

DEBRIS TRANSPORT ANALYSIS RELATED WITH GSI-191 IN ADVANCED PRESSURIZED WATER REACTOR EQUIPPED WITH INCONTAINMENT REFUELING WATER STORAGE TANK

Jeong Ik Lee¹, Soon Joon Hong^{1*}, Jonguk Kim¹, Byung Chul Lee¹,
Young Seok Bang², Deog Yeon Oh², and Byung Gil Huh²

¹*FNC Tech. Co. Ltd.,

SNU 135-308, San 56-1, Shinrim 9-Dong, Kwanak-Gu, Seoul, 151-742, S. Korea

Tel: +82-2-872-6084, Fax: +82-2-872-6084, Email: sjhong90@fnctech.com

²Korea Institute of Nuclear Safety,

19 Gusung-Dong, Yuseong-Gu, Daejeon, Korea, 305-755

Abstract

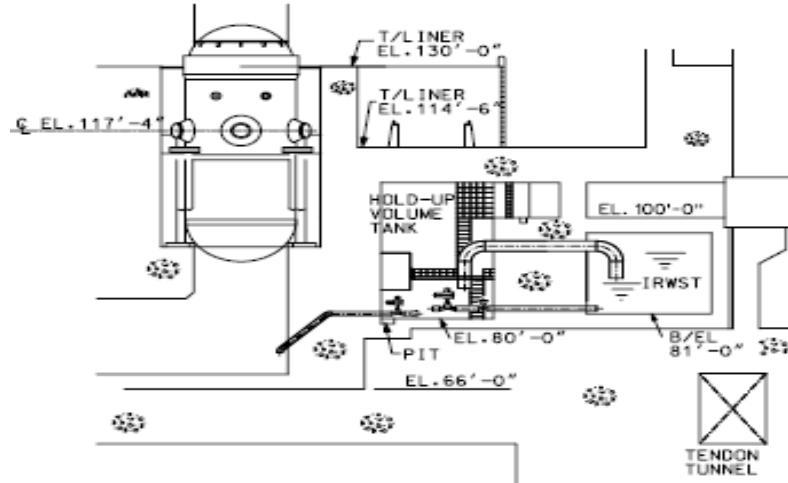
A recirculation sump blockage issue is an important and urgent problem to be resolved for ensuring nuclear power plant safety. Through a series of intensive resolution activities, a new regulatory guide and an evaluation methodology were proposed. However, these were restricted to a conventional pressurized water reactors (PWRs), and the applicability of these methodologies to a new type or advanced PWR has not been fully addressed. In this paper the applicability of suggested methodologies to a Korean advanced nuclear power plant, which is different from other PWR due to an In-containment Refueling Water Storage Tank (IRWST), is discussed. The scheme for obtaining a solution by using Computational Fluid Dynamic (CFD) is introduced. The CFD was applied to correctly simulate free surface flow. Related mathematical models and numerical analyses are explained in detail.

1. INTRODUCTION

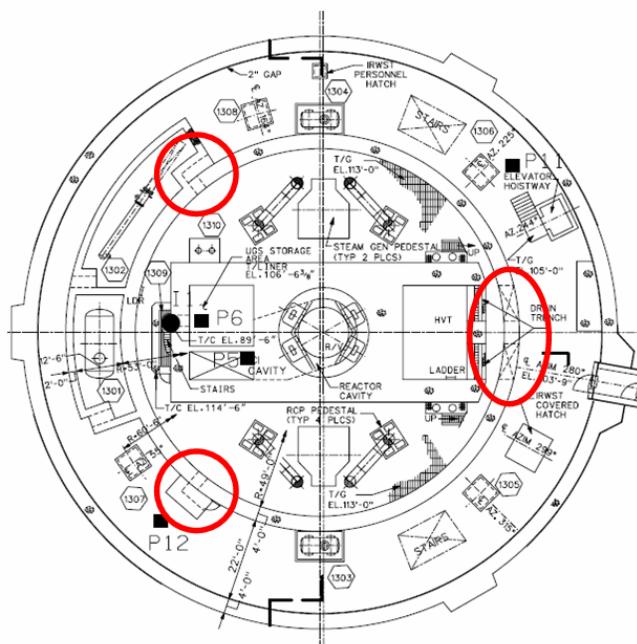
In 2003, USNRC (United State Nuclear Regulatory Committee) revised Regulatory Guide 1.82 (RG. 1.82) to revision 3, and in 2004 issued generic Letter 2004-02 for PWRs (Pressurized Water Reactors) (USNRC, 2003, 2004a). These USNRC's regulatory actions via several intensive researches made the sump blockage issue an important safety problem. As a response to these USNRC's activities, NEI 04-07 was developed in order to evaluate the post-accident performance of a pressurized water reactor's (PWR's) recirculation sump, and has been used as a standard methodology (NEI, 2004). NEI 04-07 methodology is composed of break selection, debris generation, latent debris, debris transport, and estimating head losses. The debris transport is evaluated through blowdown transport, washdown transport, pool fillup transport, and recirculation transport. Such a debris transport evaluation methodology is suitable to PWR which is equipped with Refueling Water Storage Tank (RWST) outside the containment.

Some advanced reactors, such as APR1400 (Advanced Power Reactor 1400 MWe), are equipped with In-containment Refueling Water Storage Tank (IRWST) instead of RWST for more reliable emergency core cooling (ECC) water supply. However, such a design feature results in a different break flow behaviour inside a containment compared to conventional PWRs. This is because IRWST, as the acronym implies, changes the ECC water injection sequence. In other words, since the RWST is situated inside the containment, no process is required to switch water source from RWST to recirculation sump. Therefore, the water source for ECC is always IRWST while conventional PWRs, which current methodology is based on, switches from RWST to recirculation sump. This fact based observation made us to review the applicability of NEI 04-07's methodology to APR1400.

In this paper we discuss the applicability of NEI 04-07's methodology to an advanced reactor type with RWST inside the containment. Moreover, a new methodology is proposed and demonstrated by applying CFD analysis. Finally, a rough debris transport estimation is presented based on the analysis result. The reference plant is APR1400 which is an advanced PWR developed by Korean nuclear industry. APR1400 incorporates several improved safety features, and one of the key features is an adoption of IRWST mentioned above and important to address the sump blockage issue.



(a) Vertical View



(b) Plane View

Fig. 1: Components on Containment Floor and Secondary Shield Wall Passages

2. REVIEW OF REFERENCE PLANT DESIGN AND DEBRIS MOVEMENT

As it is already mentioned in the previous section, APR1400 has different features from conventional PWRs, which it has IRWST instead of RWST and recirculation sump. Thus, the water source for long term cooling is solely IRWST, and there is no switching process from RWST to recirculation sump. The configuration of IRWST and other partial components, which consist APR1400, is shown in Fig.1 (a) and (b). It is noted that the red circles on Fig.1 (b) is to highlight the secondary shield wall passage, which connects the interior and the exterior of secondary shield wall. These three passages can be viewed as break water passage also during LOCA (Loss of Coolant Accident).

In case of LOCA the water from a break splashes down to the floor, and some portion of water is directed to bottom floor. Since HVT (Holdup Volume Tank) is located below (See Fig.1 (a)) the bottom floor, the break water drains to HVT through four trenches. And the water is finally collected in IRWST via spillways. The water in IRWST is provided to RCS (Reactor Coolant System) by ECC pumps. Debris is expected to move together with redistributing break flow.

During earlier phase of the accident the break flow will sweep out and spread wide on the bottom floor. The debris which is located in high velocity region will be transported to HVT. When the water level in HVT reaches the spillway, the water will flood to IRWST, and some debris will be also transported to IRWST. It is very difficult to evaluate how much debris will enter the spillway, because the flow pattern in HVT is a complex phenomenon and the relation of water and debris behavior is not clearly understood yet. Since ECC pumps and containment spray pumps operate and inflow from/through spillways exists, steady flow field is formed in IRWST. Thus, some debris will move to the suction part of pumps.

From the design features and an observation of flow behaviours, it is safe to suggest that some debris will be transported to IRWST during the initial phase of accident. This means some revision of NEI 04-07 method is required to apply the method to APR1400 case, since the method does not consider these effects due to different design. To evaluate debris transport in early phase of the accident, a free surface computational fluid dynamics (CFD) calculation is initially required to understand the water redistribution process. Current CFD skill can calculate a free surface flow using simple but robust numerical schemes, such as VOF (Volume Of Fluid) model. In this paper, the break flow from the initial release is modeled, and the redistribution of water on the floor is simulated. By utilizing calculated velocity profiles, debris transport fraction can be evaluated.

3 NUMERICAL ANALYSIS MEHODOLOGY

3.1 Description of Adopted Physical Models

Two major transport equations are solved in order to simulate a flow distribution over a containment floor. One is the continuity equation (1) and the other is the momentum conservation equation (2).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0 \quad (1)$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U} \times \vec{U}) = \nabla \cdot \left(-p \delta + \mu \left(\nabla \vec{U} + (\nabla \vec{U})^T \right) \right) \quad (2)$$

In addition to the equation set presented here, transport equations for turbulence have to be solved simultaneously. Turbulence is a chaotic motion but not a totally random motion so that statistical description is possible. Even though the true solution to the momentum equation given above is expected to describe the nature of turbulence successfully, it is widely accepted that the true solution is nearly impossible to obtain for general conditions at current level of understanding. Furthermore, Direct Numerical Simulation (DNS), a recent numerical scheme developed for solving the momentum equation directly without an aid from any empirical model, requires heavy computing resources to obtain approximate solution even with a cutting edge computational technology. Thus, an easy approximation to true turbulent motion is essential to solve an engineering problem such as ours.

One of the most widely applied approximation methodologies to describe true turbulent motion is Reynolds averaging scheme. By applying this scheme and Boussinesq's approximation the momentum equation presented in eq. (2) can be transformed to eq.(3)

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu + \mu_t) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] - \frac{\partial}{\partial x_j} \left(\frac{2}{3} \delta_{ij} \rho k \right) \quad (3)$$

To close equation (3) we require physical model for turbulent viscosity (μ_t) and turbulent kinetic energy (k). For our particular problem, RNG (Renormalized Group) $k-\epsilon$ is used. The reason why this

model was chosen for the calculation is because the model was used in NEI 04-07 SER, and it is also recommended in the other report (USNRC, 2004b). RNG k- ϵ is a model that has all the features of the standard k- ϵ model with added terms to describe swirl, buoyancy and compressible nature of a fluid flow. These supplemented terms enable RNG k- ϵ model to simulate radically strained flow and flow over a curved surface better than the standard k- ϵ model, which is necessary features to gain more accuracy of our solution.

In addition to the turbulent model, a two-phase model has to be considered too. This is because physical mechanism of how the break flow redistributes on top of the containment floor surface is equivalent to tracking a boundary surface between air and water, which interaction between air and water needs to be considered simultaneously. The model adopted here is VOF model. This model is one of the simplest two phase model and it also recommended for treating free surface flow problem such as ours. It hypothesizes that multiple phases in the system can be viewed as one fluid mixture so no additional momentum or energy equation has to be solved. Therefore, only continuity equation for each component or phases is satisfied in the problem domain, and other physical properties are based on the mixture at a certain time and geometrical point. Hypothesis of VOF model shows satisfying performance for stratified flow, free-surface flow (such as our case), filling, sloshing, large bubbles in liquid flow, jet breakup, steady or unsteady tracking of gas-liquid boundary surface etc. For the surface reconstruction, the Euler explicit method was chosen over other surface tracking schemes, since it is based on the simplified physical model of how volume fraction of a cell transports to the other cell rather than calculating the surface by a pure mathematical treatment.

3.2 Determining Problem Domain and Boundary Conditions

In the previous section, geometrical characteristics of the containment interior of the reference plant were reviewed and developed flow passages during a situation when reactor coolant flows into the containment, such as LOCA, were identified. To successfully model the problem, the domain of our interest is specified and computational space is constructed in this subsection.

The containment floor can be divided into three regions.

- Region 1: Inside the Primary Shield Wall
- Region 2: Between the Primary Shield Wall Exterior and Secondary Shield Wall Interior
- Region 3: Between the Secondary Shield Wall Exterior and Containment Outer Wall Interior

Region 1 is where the reactor core is located and the floor of Region 1 is completely isolated from free space in the containment. Thus, this region does not coincide with our interest. Region 2 and Region 3 are divided by the secondary shield wall. Two regions are interconnected through narrow passage ways indicated as circles in Fig. 1. Initial break flow collides with the containment floor with a high velocity and spreads out over the secondary shield wall. Since the interconnecting passages are narrow, it is expected that only a small fraction of water will be relocated to Region 3 during our calculation, which is less than 10sec. Also, since Region 3 covers large area and water moves into Region 3 is not going to flow with a high velocity, including Region 3 to our problem domain will end up costing large computing resources without gaining any new insight. Therefore, only Region 2 is set as our problem domain and this is predicted to be more conservative than including Region 2 and 3 together in the viewpoint of debris transport, since higher flow velocity is expected during the calculation.

Height from the containment floor is determined to minimize computing resource and maximize accuracy and conservatism of the calculation. Since water that flows through the break will hit the ground floor and splash back or crawl over the wall, it is imperative to set the height near the break region precisely to simulate water that is not participating on water level rising on the floor but falls down later from upper structures like hot leg, cold leg, grating and other pipelines. However, the height that is far from the break region has less significance for determining the water distribution on the floor since the effect, which occurs near the break region, does not play a major role here. Therefore, the height far from the break region is lower than the height near the break region, and this is shown in Fig. 2. Other structures considered inside the problem domain is foundations for a steam generator and reactor coolant pumps. The geometry modelled into CAD program is based on the Preliminary Safety Analysis Report (PSAR) of the reference plant.

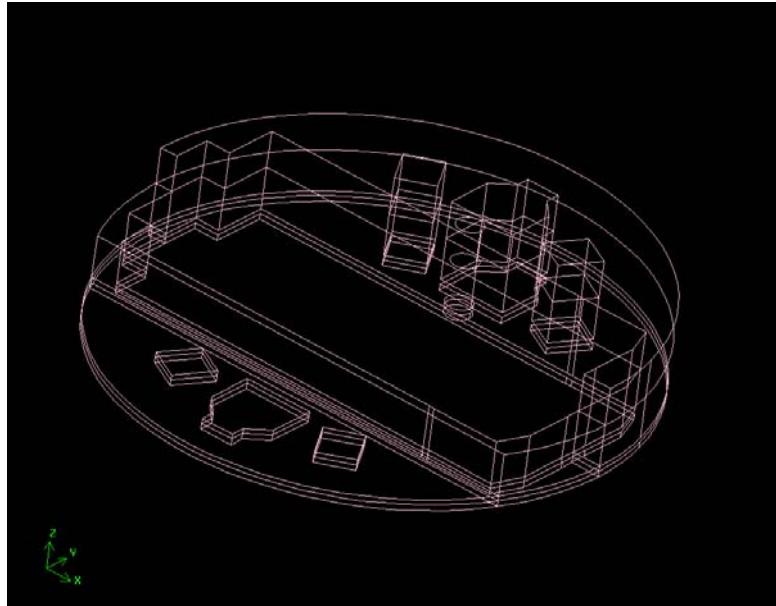


Fig. 2: Isometric View of Problem Domain

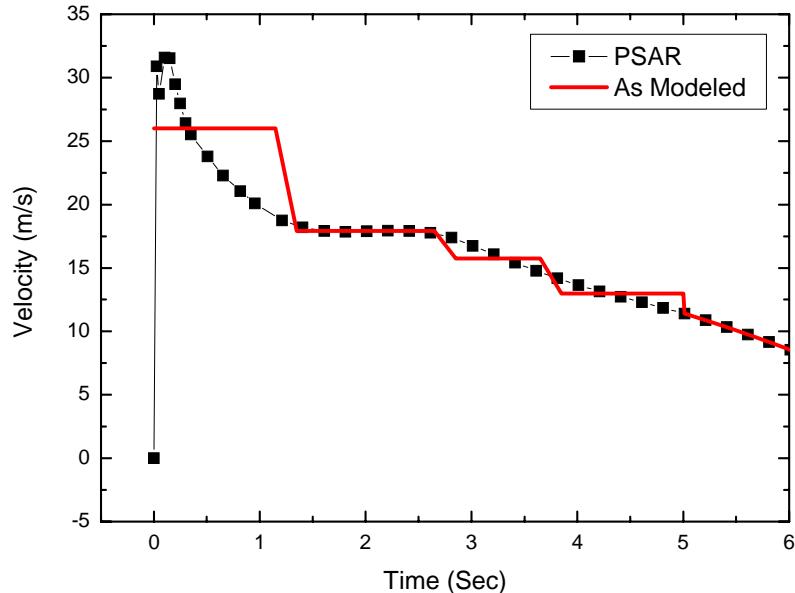


Fig. 3: Velocity of the Break Flow vs. Time

Imposed boundary conditions are one flow velocity inlet condition (the break) and four pressure outlet conditions, which simulate four trenches that connects ground floor of the containment to the Holdup Volume Tank (HVT) located at lower level as shown in Fig.1. It is noted that HVT is not considered as a part of the problem since we are only interested in how initial phase of LOCA generated break flow redistributes on the containment floor. For setting the break flow condition, water at room temperature and pressure is used. The amount of water entering into the domain is calculated from the PSAR (Preliminary Safety Analysis Report) mass release table and assuming isentropic process for vaporization to calculate liquid water mass only. The assumption of isentropic process actually leads to a smaller generation of vapor phase and since more liquid water means more debris transport to HVT, the assumption is fairly conservative. Velocity of the break flow is determined from the liquid water mass flow rate calculated above and the density of water at room condition. This is represented in Fig.3 as a line with squares. However, violent release of water at the beginning phase of LOCA is predicted to hinder the convergence of the calculation, average value over a unit second is used in the

analysis instead. The modeled value is shown in Fig.3 as a solid line. It is clearly shown that the total amount of water entering the containment is same between a value calculated from PSAR and the model, and the total amount of water is more important than the velocity of the water for determining the water level in the containment and the debris transport. This is because the tumbling velocity defined as the velocity for debris to be transported is very low (0.1 ft/sec), which means as long as the fluid velocity is above the tumbling velocity the debris will be moved. As it will be presented in the next section, the fluid velocity of an area covered with break water is always higher than the tumbling velocity during the initial phase, the containment floor area fraction covered by water is more important than the liquid velocity. The containment is initially assumed to be completely filled with air at room condition.

3.3 Procedures for Obtaining Numerical Solution

Throughout the process of obtaining numerical solution, commercial CFD code, FLUENT 6.3 is used. Fig. 4 shows meshed domain by GAMBIT 2.0. 1,357,576 hexahedral cells and 3,981,536 quadrilateral interior faces exist in the domain and it is consisted of 1,449,929 nodes.

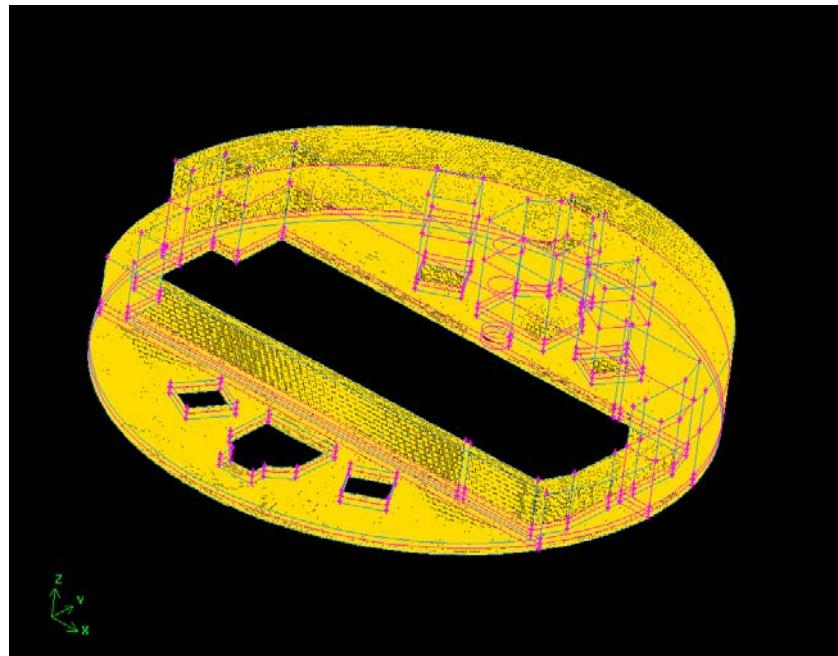
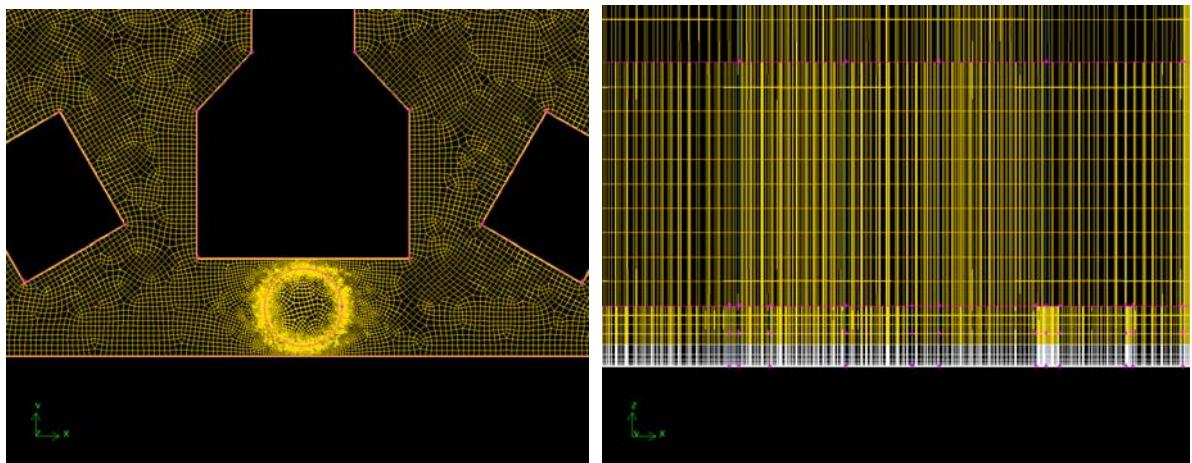


Fig. 4: Isometric View of Meshed Domain



(a) Magnified View of Break Flow Area

(b) Magnified View of Floor Area

Fig. 5: Close View of Meshed Domain

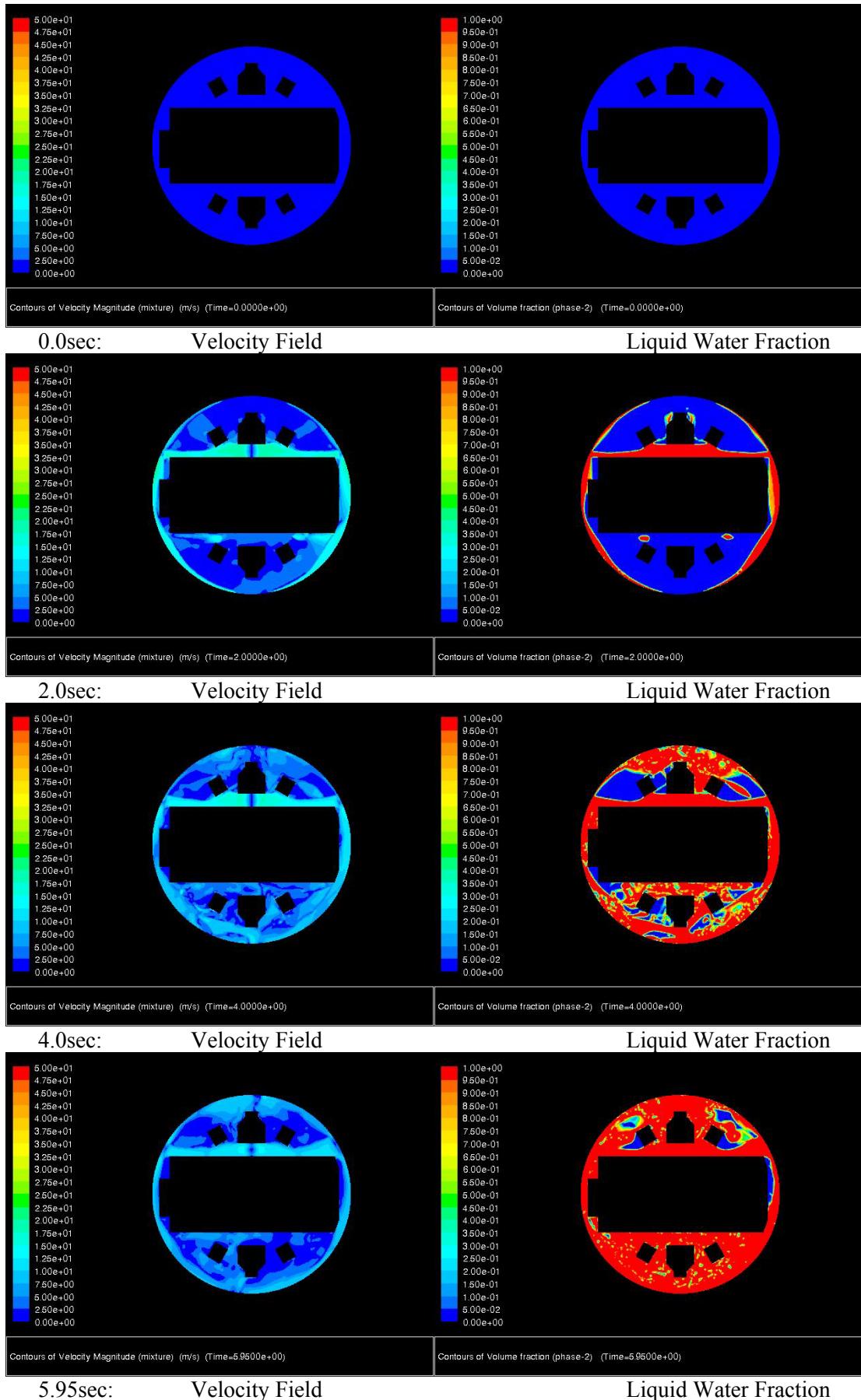


Fig. 6: Velocity Field and Water Fraction at a Surface 1cm above the Floor

Fig. 5 shows magnified views near the break area and the floor. Since the break flow is modelled as water falling down from the top to the bottom, water column is going to be created from the break to the floor. Thus, fine meshes are required from the break to the floor where water column is created to capture the boundary surface between air and water and to minimize over diffusion of information due to large mesh size near the boundary. The break area is twice the cold leg pipeline flow area, since water releases from both side of the break. Furthermore, boundary layer meshes were utilized for meshing near the floor to accurately model generated turbulence from the floor fluid interaction.

First order upwind scheme is used for discretizing partial differential. Even though the lower order discretization can induce some numerical diffusion, the preliminary analysis result showed that the numerical diffusion was not that significant. For instance, the time for break water to collide with the floor was estimated to be around 0.2 sec, and the calculation result also showed similar result. Therefore, it was concluded that lower order discretization was fair enough for the calculation. Upwind implies that the convection term calculated at the cell interface is derived from the upstream cell average value. For coupling velocity and pressure to satisfy mass conservation and obtaining pressure field, SIMPLE algorithm is used. SIMPLE algorithm adds a pressure correction term to the flux at a cell interface. This is necessary measure since both air and water are considered as an incompressible fluid. In other words, since the fluids' density is not a function of pressure, a modification to the original equation is necessary to obtain pressure field while satisfying mass conservation. PRESTO (PRESSure STaggering Option) is used for pressure interpolation. The PRESTO scheme uses the discrete continuity balance for a "staggered" control volume about the face to compute the face pressure. For flows with high swirl numbers, high-Rayleigh-number natural convection, high speed rotating flows, flows involving porous media, and flows in strongly curved domains, the PRESTO scheme is preferred over other pressure interpolation scheme. These options and numerical schemes can be found in more detail in FLUENT 6.3 manual.

First order implicit time integration was chosen for the time discretization. Implicit scheme has an advantage compared to explicit scheme for its good stability. Since implicit scheme was chosen for the time integration, iteration was done for each time step until it converges. The time step was set to 0.0005 sec and convergence criteria was set to less than 0.001 for all residuals of discretized equations. Calculation was performed for 5.95 sec in problem time. The result was obtained from supercomputer and 8 processors were utilized for the calculation.

4. RESULTS AND DISCUSSION

4.1 Results

Fig. 6 shows the calculation result of velocity field and water fraction at a surface 1 cm above the floor for every two seconds. It is noted that velocity field is for the mixture. Since the two phase model we have selected is VOF model, there is no velocity difference between water and air. The figure shows how the break flow initially distributes itself over the containment floor sequentially. Next, the analysis for setting upper and lower bounds of debris transport to HVT will be presented by utilizing the CFD analysis result. However, it should be emphasized that these bounds are rough estimates for debris transport, and more detailed analysis is being carried out.

4.2 Estimation of Debris Transport Fraction

Method 1:

This method follows NEI 04-07's with additional conservatism. The decision of debris transport is dependent on the local velocity. If the velocity is higher than the tumbling velocity of specific debris, the debris is assumed to transport.

Mixture velocity at a surface 1cm above the floor which is higher than the minimum velocity required for moving debris 0.03 m/s (tumbling velocity) covers about 100% of the floor. This can be interpreted that upper bound for fibrous debris transporting to HVT, which falls into fine and small category, is

100%. In other words, all the small and fine fibrous debris generated by LOCA will be relocated to HVT.

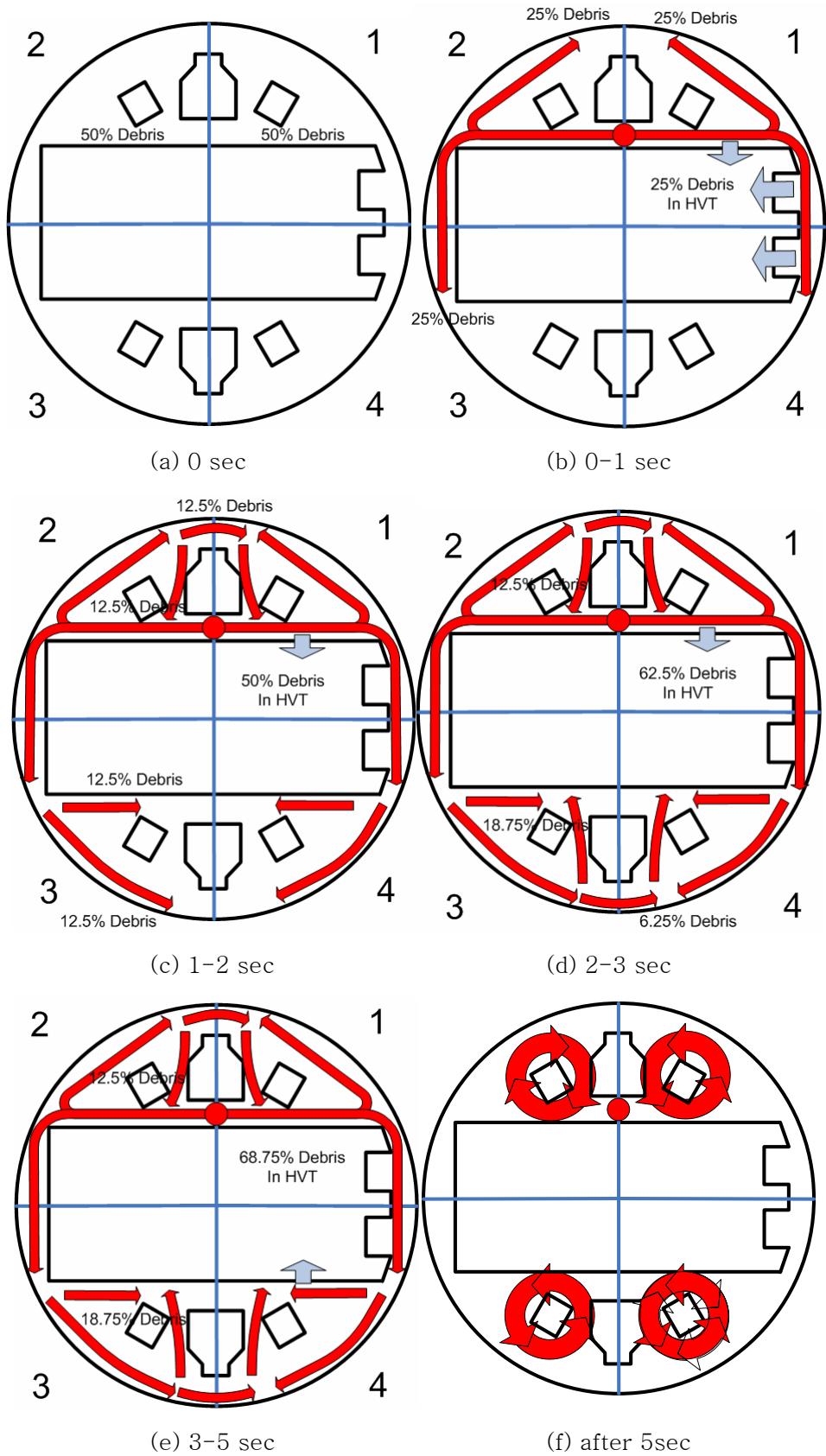


Fig. 7: Debris Transport during Liquid Water Redistribution

Method 2:

This method is one that debris transport is decided by flow pattern in each quadrant rather than local velocity. As it was already mentioned in method 1, after the break flow hits the ground and redistributes over the floor the velocity of the flow is mostly higher than the minimum tumbling velocity. Thus, by analyzing how water distributes during 6 seconds can provide an insight for amount of debris relocated to HVT during 6 seconds of flow formation. The analysis is demonstrated in Fig. 7.

First the floor of the reference plant is divided into quadrants (refer to Fig. 7). This is a basis for tracking debris moving from one area to the other area. When LOCA is initiated, we can hypothesize that the LOCA generated debris is uniformly distributed among the first and second quadrants (Fig. 7 (a)) because the break is between the first and second quadrants. In Fig. 7 (b) the flow breaks into two streams by a collision of the flow with the secondary shield wall. Hence, a stream that started out from the first quadrant divides into two streams that flows to the second and the fourth quadrants and the same flow split occurs for the stream that begun at the second quadrants. Each flow split is simply assumed to carry 25% of debris. This assumption is very rough but necessary since the interaction between debris and water flow is not clearly characterized yet as it was stated in the previous section. During the analysis any flow split is viewed as a uniform split of debris among the branch streams for simplification. Since, the streams that flows from the first to fourth quadrant passes three trenches, it is assumed that 25% of debris is lost to HVT. The same process occurs through Fig. 7 (c) through (e). And finally after quasi-steady state is achieved (Fig. 7 (f)) around 70% of debris is relocated to HVT and 30% of debris is left on the floor and moves along with the quasi-steady flow created on the top of the floor.

5. Conclusions

A review of the applicability of NEI 04-07 method to a Korean advanced reactor, APR1400, which the major design difference from a convectional reactor is having IRWST instead of RWST and a recirculation sump, was performed. The NEI methodology is revised by considering debris transport during the early phase of LOCA, which was not part of the methodology previously due to different reference reactor type. The necessity of considering debris transport during the early phase of accident was verified by CFD free surface flow analysis, since it was found that major portion of generated debris has a potential to be transported and relocated to other locations than the containment floor. Upper bound for debris transport during the first 6 sec of LOCA can be viewed as 100% and the lower bound is around 70%. Even though these values carry some uncertainties due to the analysis method and many hypothesized assumptions, the numbers still suggest that during the initial phase of LOCA, significant amount of debris will be transported to HVT. Thus, it is not safe to ignore the amount of debris transport at the initial phase of LOCA in PWR which is equipped with IRWST by simply applying NEI 04-07 method without revision. As future work, the problem domain will be extended to include HVT and spillway for more information how flow will behave during an accident, and a system code that can treat free surface flow will be utilized to compare the result to CFD analysis.

REFERENCES

- FLUENT6.3 Code Manual, Fluent Inc., 2006 & GAMBIT
- KHNP, APR1400 SSAR and PSAR
- NEI, "Pressurized Water Reactor Sump Performance Evaluation Methodology", NEI 04-07 (2004)
- USNRC, "Water Sources for Long-Term Recirculation Cooling Following a Loss-of-Coolant Accident", Regulation Guide 1.82 Rev. 3 (2003)
- USNRC, "Potential Impact of Debris Blockage on Emergency Recirculation During Design Basis Accidents at Pressurized-Water Reactor", Generic Letter 2004-02 (2004a)
- USNRC, "Safety Evaluation by The Office Of Nuclear Reactor Regulation Related to NRC Generic Letter 2004-02,Nuclear Energy Institute Guidance Report (Proposed Document Number NEI 04-07), 'Pressurized Water Reactor Sump Performance Evaluation Methodology' (2004b)