Transactions of the Korean Nuclear Society Autumn Meeting
Jeju, Korea, October 21-22, 2010

A Scheme on Sump Boundary Condition in Shallow Water Equation Solver for OPR1000

Young Seok Bang [a*], Gil-Soo Lee [a], Byung-Gil Huh [a], Ju Yeop Park [a], and Sweng-Woong Woo [a]
[a] *Korea Institute of Nuclear Society, 19 Gusung-Dong, Yuseong-Gu, Daejeon, 305-335, Korea*
[*] *Corresponding author: k164bys@kins.re.kr*

1. Introduction

Two-dimensional (2-D) Shallow Water Equation (SWE) has been used to predict the transient flow field on containment floor following a loss-of-coolant accident (LOCA) in pressurized water reactor (PWR), encouraged by its fast-running capability and a reasonable predictability [1,2]. Finite Volume Method (FVM) is frequently used to solve the SWE with unstructured grid system representing the containment floor. However, the limitation from 2-D has been a problem. One of the problems was a difficulty in modeling of the recirculation sump which is a stepped-down region over the containment floor. Discontinuity in bed elevation between the floor and the sump cannot be directly modeled by the SWE, which always requires a complicated approximation during solution procedure [3]. Special boundary condition scheme suitable to simulate the falling flow into the sump pit can be applied, instead of the special technique.

To apply the SWE solver to the Optimized Power Reactor (OPR) 1000 [4], three types of phases related to the sump boundary condition should be considered: (1) water falling to the sump, (2) floor filling-up phase after sump filled, and (3) recirculation. Among the three phases, the boundary condition for the recirculation can be easily deduced. The present paper has its aim to discuss a scheme of determine the boundary condition suitable to the phases (1) and (2).

2. Analysis Model

The Shallow Water Equation (SWE) is as follows:

$$\frac{\partial \boldsymbol{W}}{\partial t}+\frac{\partial \boldsymbol{F}}{\partial x}+\frac{\partial \boldsymbol{G}}{\partial y}=\frac{\partial \boldsymbol{R_x}}{\partial x}+\frac{\partial \boldsymbol{R_y}}{\partial y}+\boldsymbol{S} \quad \text{.......(1)}$$

$$\begin{aligned}
&\boldsymbol{W}=[h, hu, hv]^T,\ \boldsymbol{F}=[hu, hu^2+1/2gh^2, huv]^T,\\
&\boldsymbol{G}=[hv, huv, hv^2+1/2gh^2]^T\\
&\boldsymbol{R_x}=[0, \nu_t(\partial hu/\partial x), \nu_t(\partial hv/\partial x)]^T,\\
&\boldsymbol{R_y}=[0, \nu_t(\partial hu/\partial y), \nu_t(\partial hv/\partial y)]^T\\
&\boldsymbol{S}=[B, -\partial z_b/\partial x - gn^2uq, -\partial z_b/\partial y - gn^2vq]^T\\
&q=\sqrt{u^2+v^2}/h^{1/3}
\end{aligned} \quad \text{.......(2)}$$

The description of variables, subscript, etc, can be found in reference [1]. The finite volume equation of the Eq.(1) for triangular mesh can be written as follows:

$$\begin{aligned}
\boldsymbol{W}_k^{n+1/2} &= W_k^n-\frac{\Delta t}{2A_k}\sum_{j=1}^{3} flux_{conv}+\frac{\Delta t}{2A_k}\sum_{j=1}^{3} flux_{diffus}+\frac{\Delta t}{2}\boldsymbol{S}_k^{\ n} \quad \text{....(3)}\\
\boldsymbol{W}_k^{n+1} &= \boldsymbol{W}_k^n-\frac{\Delta t}{A_k}\sum_{j=1}^{3} flux^*_{conv}+\frac{\Delta t}{A_k}\sum_{j=1}^{3} flux_{diffus}+\Delta t\boldsymbol{S}_k^{\ n}
\end{aligned}$$

Eq.(3) is solved at the center of all the cells using the fluxes across the cell sides and the predictor-corrector scheme was applied in order to preserve the second order accuracy.

An approximate Riemann solver, Harten-Lax-van Leer (HLL) scheme [5] was introduced to the corrector step, to avoid unphysical oscillation and instability of the solution especially at the wet-dry interface.

$$F^*(\boldsymbol{W}_R,\boldsymbol{W}_L)=\frac{[(s_R\boldsymbol{F}(\boldsymbol{W}_L)-s_L\boldsymbol{F}(\boldsymbol{W}_R))\cdot \boldsymbol{n}+s_Ls_R(\boldsymbol{W}_R-\boldsymbol{W}_L)]}{s_R-s_L} \quad \text{...(4)}$$

$$\begin{aligned}
&s_L=\min(\boldsymbol{q}(\boldsymbol{W}_L)\cdot\boldsymbol{n}-c(h_L),\ \boldsymbol{q}^*\cdot\boldsymbol{n}-c^*), \quad \text{.........(5)}\\
&s_R=\max(\boldsymbol{q}(\boldsymbol{W}_R)\cdot\boldsymbol{n}-c(h_R),\ \boldsymbol{q}^*\cdot\boldsymbol{n}+c^*)\\
&\boldsymbol{q}(\boldsymbol{W})=(u,v),\quad c(h)=\sqrt{gh},\\
&\boldsymbol{q}^*\cdot\boldsymbol{n}=\frac{1}{2}(\boldsymbol{q}(\boldsymbol{W}_L)+\boldsymbol{q}(\boldsymbol{W}_R))\cdot\boldsymbol{n}+c(h_L)-c(h_R),\\
&c^*=\frac{1}{2}(c(h_L)+c(h_R))+\frac{1}{4}(\boldsymbol{q}(\boldsymbol{W}_L)-\boldsymbol{q}(\boldsymbol{W}_R))\cdot\boldsymbol{n}
\end{aligned}$$

The diffusive flux term and the source term can be approximated by the central difference scheme. The time step size to solve the Eq.(3) should be limited to prevent the negative water level as follow [1]:

$$\Delta t \le Min_k\left(\frac{A_k}{K_{CFL}Max|\boldsymbol{q}(U)\cdot\boldsymbol{n}\pm c|_{kj}}\right) \quad \text{.........(6)}$$

K_{CFL} is a coefficient similar to the Courant-Fredrich-Lewy (CFL) number in CFD calculation and set to 1.3.

3. Boundary Conditions

Information on level and velocity of the water at the boundary of the solution domain is requested to solve Eq.(3). Regarding the containment of OPR1000, the boundary condition should be specified at the solid wall and at the sump. For the solid wall, the following condition can be specified:

$$V_j=0,\ \ h_j=h_k \quad \text{.........(7)}$$

where the subscripts j and k denote boundary side and the adjacent cell centre, respectively. This equation can be valid for sub-critical regime, while a special treatment may be needed for super critical situation.

As mentioned, the boundary of the sump cannot be specified by 2-D approach. As an approximation, the flow rate can be specified as a function of water level, based on the formula for the broad crested weir [6]

$$u=\frac{Q_{BCW}}{hL_x}, v=\frac{Q_{BCW}}{hL_y},\ Q_{BCW}=(\frac{2}{3})^{3/2}g^{1/2}h^{3/2}L \quad \text{.........(8)}$$

where L represent the length of boundary side, Q_{BCW} is flow rate, and h should be taken at the still water.

However, it is not clear where the h value should be taken in the course of transient calculation. To be a practical approximation, it is recommended that the water level be at the upstream cell and velocity head term be considered.

$$u_j = \frac{Q_C}{h_j L_x}, v_j = \frac{Q_C}{h_j L_y}, \ h_j = h_k \qquad \text{(9)}$$
$$Q_C = \left(\frac{2}{3}\right)^{3/2} g^{1/2} L_j \left[h_k^{3/2} + (\boldsymbol{v}_k \cdot \boldsymbol{n}_j) h_k\right]$$

where the Q_C and $\boldsymbol{n}$ denotes the corrected flow rate and the unit normal vector at the boundary, respectively. The condition will be used during the phase of water falling into sump. Once the sump pit is completely filled, then a condition simulating the flow interaction between the sump region and the surrounding floor region should be imposed. Let the water level of the sump region be h_S and the K_S be a form loss factor, then the velocity at the sump boundary can be as follows:

$$u_j = V_S n_x, v_j = V_S n_y, \ V_S = \sqrt{\frac{g|h_S - h_k|}{1 + K_S}} \qquad \text{(10)}$$

In this expression, the energy balance between the upstream cell and sump region through boundary was considered and the water level at sump boundary was assumed to be a mean value of h_S and h_k.

4. Model Test

A simple conceptual problem considering the actual containment situation was tested to confirm the validity of the boundary condition scheme described above. Fig. 1 shows the problem. The reservoir (6x10m) has a sump pit at a point (6, 3m) of a depth 0.5 m. At the beginning, water is injected into a circle at the point (1, 2m) in 5 m^3/s and then is reduced to 0.3 m^3/s until 100 seconds. The solution domain was modeled by 841 cells. The solid wall boundary condition and the sump boundary condition were properly imposed. Currently, the K_S was assumed 0.0. Fig. 2 shows a snapshot of a water level and velocity vectors at 7.5 seconds. From this figure, one can find the flow pattern around the sump. Fig. 3 shows a comparison of water level between a point (4.95, 3) on floor and the sump region. At 10 sec, the sump was filled with water and the effect of the change of boundary condition can be identified. Also the balance between two levels reached 25 seconds.

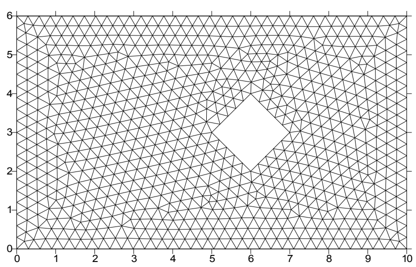

Fig. 1. Computational meshes of the problem

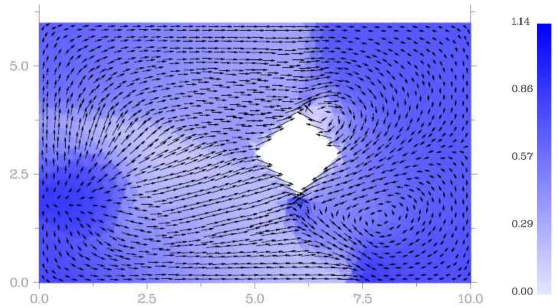

Fig. 2. Flow field at 7.5 sec.

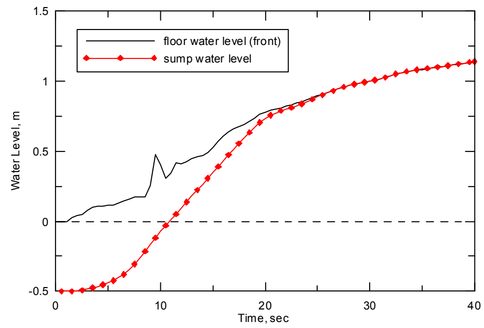

Fig. 3. Comparison of water levels

5. Summary

To apply the Shallow Water Equation solver to the calculation of flow field on containment floor of the OPR1000 plants, a scheme to treat the sump boundary condition was discussed. For the phases of water falling into the sump and floor filling-up after sump filled, a boundary condition scheme describing a flow rate as a function of water level was proposed considering the energy balance between the upstream cell and the sump. Through the testing with a conceptual problem, it was found the reasonable behavior can be predicted by the proposed scheme. The scheme will be applied to the actual containment calculation for OPR1000.

REFERENCES

[1] Bang, et al., "Prediction of Free Surface Flow on Containment Floor Using Shallow Water Equation Solver," Nucl. Eng. Tech., **41** (2009).
[2] Bang et al. "A Particle Tracking Model to Predict the Debris Transport on the Containment Floor," Nucl. Eng. Tech., **42** (2010).
[3] Liang, et al, "Adaptive Quadtree Simulation of Shallow Flows with Wet–dry Fronts over Complex Topography," Computers & Fluids, (2008).
[4] KHNP, Final Safety Analysis Report of Shinkori Units 1, 2, Seoul, Korea (2008).
[5] Harten et al. "On Upstream Differencing and Godunov-type Schemes for Hyperbolic Conservation Laws", SIAM Review, **25**, (1983).
[6] Shames I.H., Mechanics of Fluids, McGraw-Hill, New York, USA (1962).