International Journal of the Physical Sciences Vol. 5(6), pp. 774-780, June 2010
Available online at http://www.academicjournals.org/IJPS
ISSN 1992 - 1950 ©2010 Academic Journals

Full Length Research Paper

Improvement of hydraulic stability for spillway using CFD model

Sung-Duk Kim[1], Ho-Jin Lee[2*] and Sang-Do An[2]

[1]Chung-Ang University, Department of Civil and Environmental Engineering, 221, Seoul, Korea.
[2]Colorado State University, Department of Civil and Environmental Engineering, Fort Collins, CO 80523, USA.

Accepted 04 April, 2010

Hydraulic model test was used to analyze the rapidly varied flow on the spillway. But, it has some shortcomings such as error of scale effect and expensive costs. Recently, through the development of three dimensional computational fluid dynamics (CFD), rapidly varied flow and turbulence can be simulated. In this study, the applicability of CFD model to simulate flow on the spillway was reviewed. The Karian dam in Indonesia was selected as the study area. The FLOW-3d model, which is well known to simulate a flow having a free surface, was used to analyze flow. The flow stability in approach channel was investigated with the initial plan design, and the results showed that the flow in approach channel is unstable in the initial plan design. To improve flow stability in the spillway, therefore, the revised plan design was formulated. The appropriateness of the revised design was examined by a numerical modeling. The results showed that the flow in spillway is stable in the revised design.

Key words: Spillway, FLOW-3D, approach channel, flow stability, numerical modeling, hydraulic model test.

INTRODUCTION

When a large scaled dam is constructed, hydraulic stability for spillway should be examined. The examination methods are largely subdivided into a hydraulic model and numerical model test. Since a hydraulic model test deals with the flow characteristics by minimizing prototypes, reliable results can be obtained. But, it has some disadvantages such as error of scale effects in addition to expensive costs. Meanwhile, a numerical model test calculates the mathematical governing equation through a proper numerical analysis.

The flow passing through the spillway is rapidly changed by centrifugal force unlike gradually varied steady flow and if a streamline is more steeply bent, the water particle cannot follow the streamline any longer and separately forms a discontinuous section, involving a rapid turbulence like a hydraulic jump. For the rapidly varied flows, general 1-D and 2-D numerical tests cannot be used, however, due to the recent development of the 3-D computational fluid dynamics (CFD), rapidly varied flow and turbulence can be simulated very close to the prototype. The flow passing through the spillway is an important engineering problem. Cassidy (1965) analyzed spillway flow using potential flow theory and mapping into the complex potential plane. A better convergence of Cassidy's Solution was obtained by Ikegawa and Washizu (1973), Betts (1979), and Li et al. (1989) using the variation principle and linear finite elements method. Guo et al. (1998) expanded on the potential flow theory by applying the analytical function boundary value theory. This method successfully applied spillways with a free drop. Olsen and Kjellesvig (1998) simulated numerically the flow over a spillway in 2-D and 3-D for various geometries and solved the Navier-Stokes equation with the standard k-ε equations. Modelings of flow over ogee spillway were carried out using a commercially available 3-D CFD model, FLOW-3D, which solves the RANS (Reynolds Averaged Navier-Stokes) equation (Ho et al., 2001; Savage et al., 2001; Kim and Park, 2005). Numerical simulations of water flow over stepped spillway with different step configurations are presented (Tabbara et al., 2005). They adopted two-phase solution process in order to optimize the overall simulation efficiency. Most of these studies were conducted in the ideal condition.

In this study, the Karian dam in Indonesia is selected as a study area. The flow in the spillway is simulated with the initial plan design using FLOW-3D model. The

*Corresponding author. E-mail: leehojin74@gmail.com.

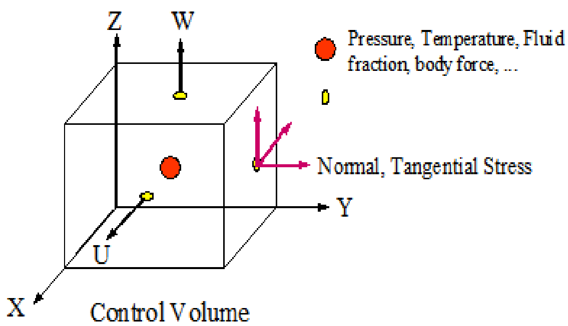

Figure 1. Grid system for a FLOW-3D.

stability of spillway is analyzed and the problems are resolved. In order to verify the accuracy of modeling the simulated results are compared with hydraulic model tests.

FLOW-3D MODEL OVERVIEW Governing equation

On the Cartesian coordinate system (x, y, z), governing equations for analysis of incompressible three dimensional flow are given by

$$V_F\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x}(uA_x)+\frac{\partial}{\partial y}(vA_y)+\frac{\partial}{\partial z}(wA_z)=\frac{RSOR}{\rho} \tag{1}$$

Where, (u,v,w) are velocity components in the coordinate directions (x,y,z), (A_x,A_y,A_z) are fractional areas open to flow in the coordinate directions (x,y,z), ρ is density and RSOR is a density source term.

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial x}+G_x+f_x \tag{2a}$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial y}+G_y+f_y \tag{2b}$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_x \tag{2c}$$

Where, V_F is a fractional volume open to flow, p is pressure, (G_x,G_y,G_z) are body acceleration in the coordinate direction (x,y,z), and (f_x,f_y,f_z) are viscous accelerations in the coordinate direction (x,y,z).

In order to model a free surface, the boundary between water and air, the VOF function (volume of fluid, *F*) should be defined to meet the following governing equation. If $F(x,y,z,t)$ is equal to 1, the control volume will be full of fluid, and if *F* is equal to 0, no fluid will exist in a control volume. Furthermore, in the case of a free water surface, *F* is shown to have the value between 0 and 1.
Applying function *F* to equation (1):

$$\frac{\partial F}{\partial t}+\frac{1}{V_F}\left[\frac{\partial}{\partial x}(FA_xu)+\frac{\partial}{\partial y}(FA_yv)+\frac{\partial}{\partial z}(FA_zw)\right]=0 \tag{3}$$

Numerical simulation algorithm

The governing equation becomes discretized using a finite difference method, whereas, a FLOW-3D adopts a finite volume method using a finite difference method plus a FAVOR (Fractional Area and Volume Obstacle Representation) method(Flow science, 2002).

After the analysis areas are divided by grids, the calculation is carried out based on a grid unit. That is, the velocity can be computed for the given pressure at each grid, and using the velocity, the value of a pressure equation in a form of a Poison equation can be calculated. Then, the velocity can be adjusted to the computed value. If it is necessary to calculate the free water surface, it is common to adopt a VOF using the fixed Eulerian method. The VOF method for FLOW-3D adopts accurate pressure and kinetic boundary conditions and describes movement between two fluids using a special numerical difference method in order to prevent the boundary face from smearing. A grid system for a FLOW-3D is shown in Figure 1.

A mesh and geometry are independent elements for a grid system composition. A hexahedral gird includes topographical information such as structures and walls (Figure 2).

A difference equation of the governing equation can be solved with an explicit method except for pressure term of momentum equation and flow velocity term of a continuity equation. Both SOR and SADI can be used in order to compute a pressure term satisfying a continuity equation.

STUDY AREA

The Karian dam site on the Ciberang River, a tributary of the Ciujung River, is located at about 10 km upstream from Rangkasbitung. The Ciberang River flows with a width of around 40 m and riverbed of about EL. 17.5 m at the dam site. The left bank shapes a relatively steep slope of about 30° up to around EL. 75 m. The right bank is a steep slope about 40° up to EL. 60 m. Hills surrounding the dam and reservoir are about EL. 70 m to EL. 100 m.

The geology of the dam site consists of tuffaceous sedimentary rocks which are generally soft rock, especially weak in pumiceous facies. Table 1 is the specification of Karian dam(KRA and KWATER, 2006).

SIMULATION CONDITION

Topography and spillway

In this study, the topography and the spillway of the Karian dam were determined by locally measured data and a digital map drawn by aerial photogrammetry. The topography was determined to fully cover the range of a modeling including the upstream and downstream (Table 2). The vertical range extended to the range of exceeding the maximum water level of the reservoir in PMF. Figure

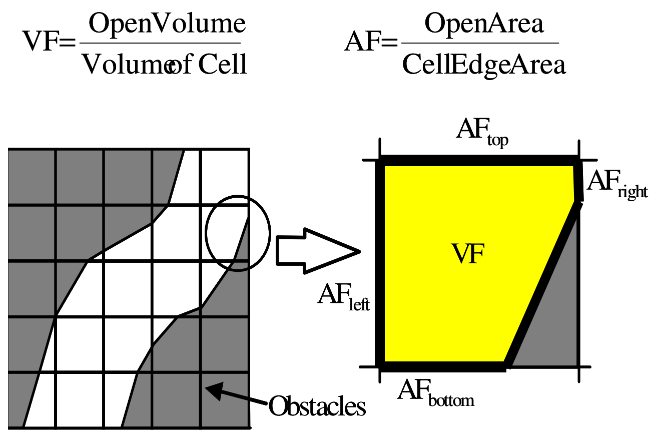

Figure 2. Conceptual diagram of a FAVOR method.

Table 1. Specification of Karian dam.

Maximum water level	EL. 71.22 m
Flood water level	EL. 70.85 m
Normal high water level	EL. 67.50 m
Peak inflow	3,671 m^3/s(PMP)
	1,850 m^3/s(1/2 PMP)
Peak discharge	3,190 m^3/s(PMP)
	658 m^3/s(1/2 PMP)
Spillway type	gated spillway + side channel spillway
Spillway elevation	EL. 57.5 m(gated spillway)
	EL. 67.5 m(side channel spillway)
Gate dimension	B12.5 m × H13.4 m × 2EA
Gate type	radial gate

3 depicts the reproduced topography, whereas, Figure 4 does the spillway and the approach channel part.

Grid composition

The rectangular coordinate system was used as the coordinate system for the simulation. By using the digital map drawn by aerial photogrammetry, the structures were combined with the three dimensional topography that were reproduced.

The grid was used to examine stability of the approach channel and discharge capacity of the spillway. The modeling range was made to reproduce accurate flow conditions using a coarse grid near the structures such as the weir. Table 3 shows the grid composition which is used in the simulation.

Boundary and initial condition

The reservoir level for the upstream was set as the boundary condition. For the downstream, by estimating accurate discharge-water level relation equation, the downstream river level according to discharge was determined to be the boundary condition (Figure 5). Because the rest are the solid boundaries, they were set as the no slip condition in order to reflect roughness of the wall face. The discharge through the side channel and the whole opening of gates were examined in order to look into the spillway stability. Table 4 is the detailed boundary condition.

In the case of the upstream, the initial condition was defined as having the same water level ranging from the upstream reservoir to the overflow crest and all dependent variables such as the flow velocity were fixed as zero. The initial water level of the downstream level was determined by a stage-discharge relation curve to the downstream river.

ANALYSIS AND RESULTS

As to the stability of the approach channel for 3,190 m^3/s, PMF discharge, and flow at the approach channel were examined when the gates were fully opened. Figure 6 depicts the flow at the approach channel and the figure's brightness represents the size of flow velocity. The flow should not be disruptive at the approach channel and the approach channel for design discharge should not exceed 4 m/s. According to the numerical analysis results, the flow flows over the wall of the approach channel due to high flood level in PMF and it is disrupted. However, the approach velocity ranges from 2.0 - 3.5 m/s and in design flood discharge. According to the examination results of the existing plan design, the biggest problem is that due to high flood level, the flow was likely to overflow the wall of the approach channel and be disrupted at the approach channel. Figure 7 shows flow phenomena of the approach channel in PMF discharge and flow overflowing the guidewall occurs by uniting approach guidewall with upstream slope elevation of the dam. Unstable inflow occurs due to the approach guide wall overflow. Hence, appropriateness of the revised design was examined by using a numerical test after the wall of the approach channel was modified. To solve this problem, the initial plan design was revised. The modified items are as follows:

(a) Crest of approach guide wall was changed to 72.5 m, dam crest.
(b)The radius of curvature and length remain the same.
Figure 8 shows the revised approach channel section.

Based on PMF discharge conditions, flow at the approach channel was examined. Figure 9 shows the flow at the approach. Figure 10 shows flow phenomena of the approach channel in PMF discharge. The result found that flow disruption at the approach channel disappeared and stable flow at the approach channel was formed.

Conclusion

The flow in the spillway and hydraulic jump is very

Table 2. The range of modeling**.**

Section	x			y			z		
	Min.	**Max.**	**Dis. (m)**	**Min.**	**Max.**	**Dis.(m)**	**Min.**	**Max.**	**Dis. (m)**
Model range	732	940	208	238	744	506	12	76	64

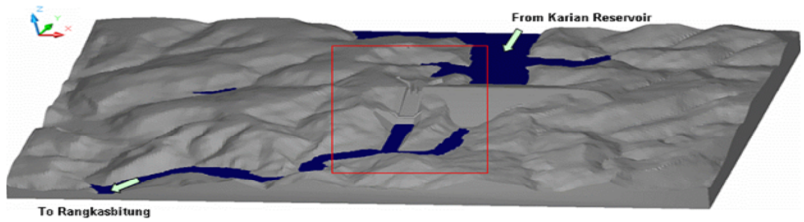

Figure 3. Reproduction results of topography, dam, and spillway.

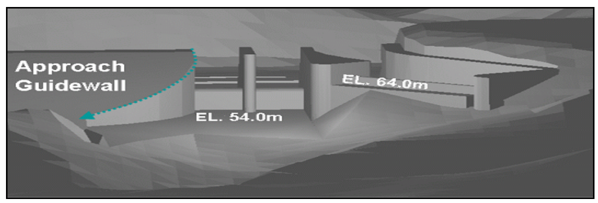

Figure 4. Spillway and approach channel.

Table 3. Grid composition.

Section	Δx			Δy			Δz			The number of grid
	Min. size (m)	**Max. size (m)**	**EA**	**Min. size (m)**	**Max. size (m)**	**EA**	**Min. size (m)**	**Max. size (m)**	**EA**	
Grid	1.35	2.70	130	1.88	8.30	130	0.63	3.70	50	845,000

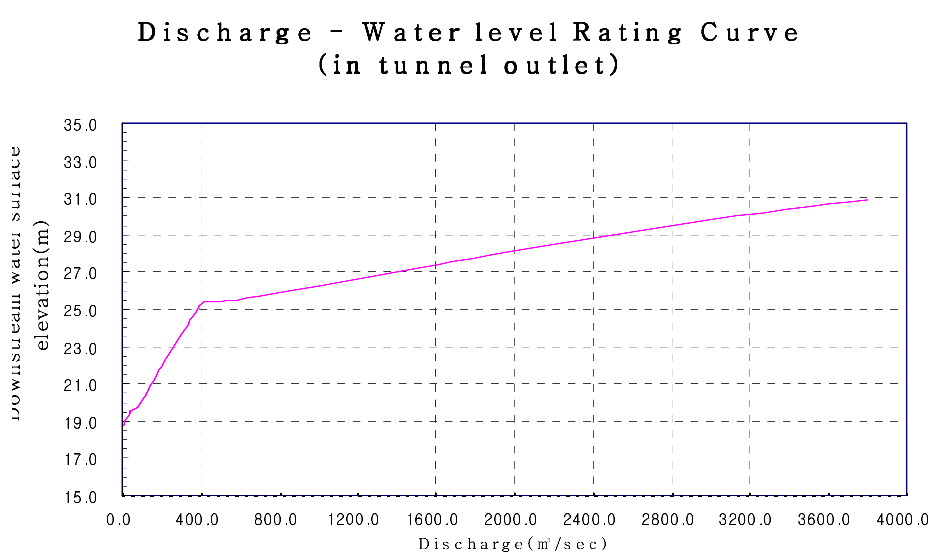

Figure 5. Stage-discharge relation curve of the downstream river.

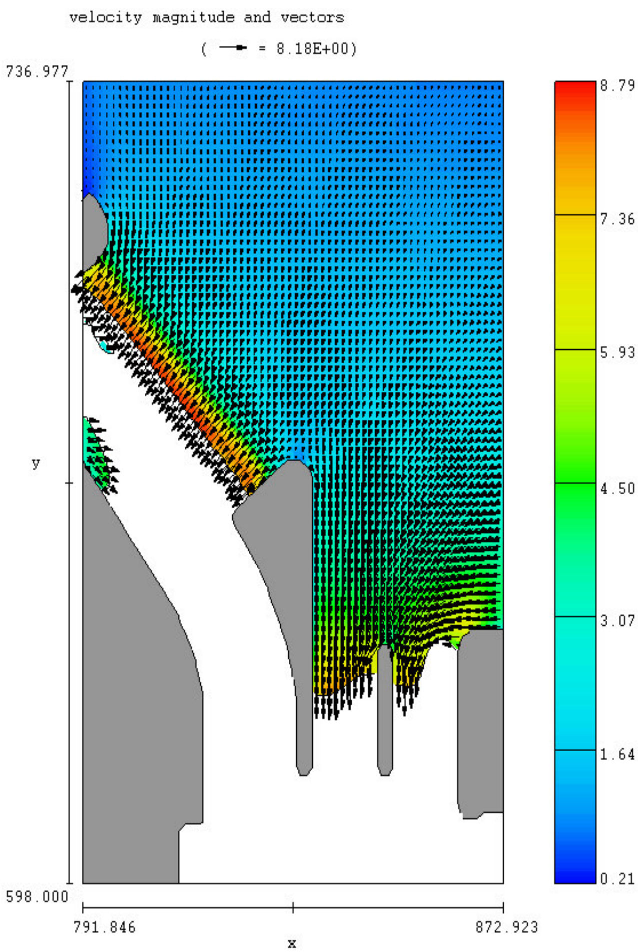

Figure 6. Two dimensional flow velocity distribution at the approach channel (Flow velocity distribution at depth EL. 68.12 m).

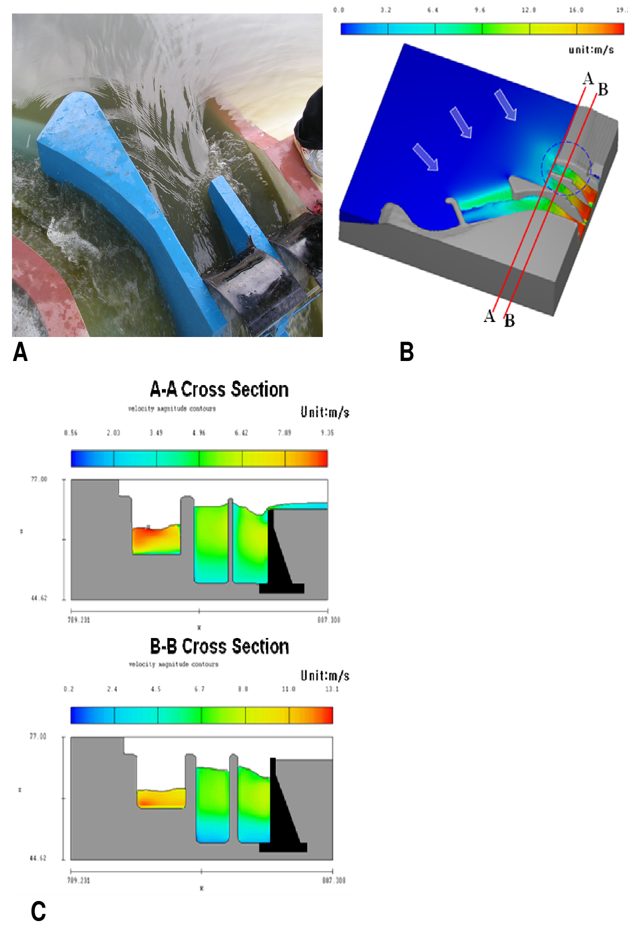

Figure 7. Flow distribution at the approach channel in PMF. A. Hydraulic model test; B. Numerial simulatio C. Cross section view.

complex phenomenon. So, hydraulic model test was used to analyze these flows. But, because hydraulic model test had a disadvantage such as the error of scale and expensive costs, there are many efforts to alternate hydraulic model test.

In this study, the Karian dam in Indonesia was selected study area to examine the applicability of CFD model.

The flow in spillway based on initial plan design was simulated by FLOW-3D model. According to the simulation result, the flow flowed over the wall of the approach channel and was disrupted at the approach channel in PMF. The revised plan design suggested that the height of the approach channel of the guide wall is upwardly adjusted to EL. 72.5 m, the same as the crest of the dam. The flow in the spillway was simulated with the revised plan design. The simulation result shows that the flow in the spillway is stable.

In all cases, an excellent agreement has been made between the hydraulic model test and the numerical simulation. It shows that the simulation of spillway flow using CFD model is possible.

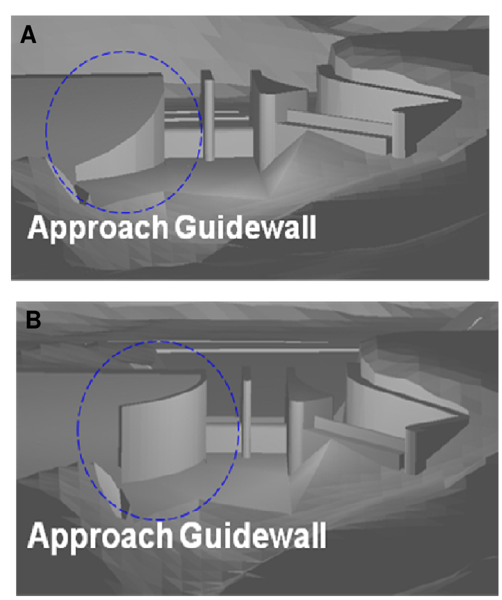

Figure 8. Revised approach channel section. A. Initial plan design; B. Revised plan design.

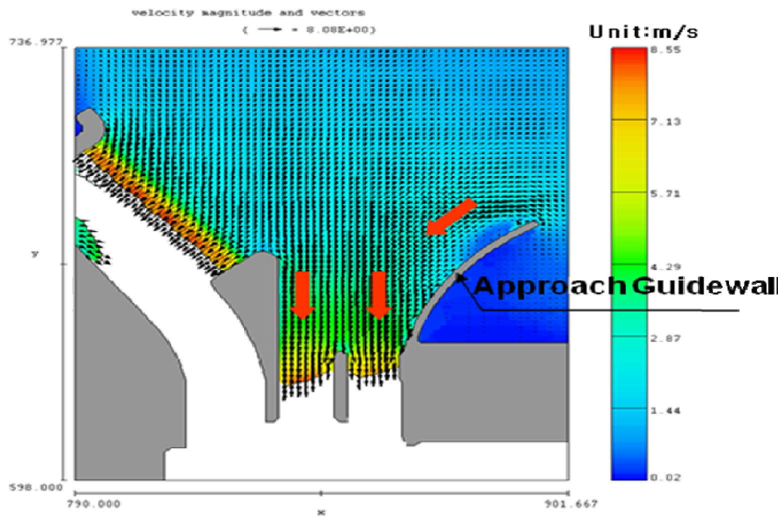

Figure 9. Two dimensional flow velocity distribution at the approach channel based on revised plan design (Flow velocity distribution at depth EL. 68.12 m).

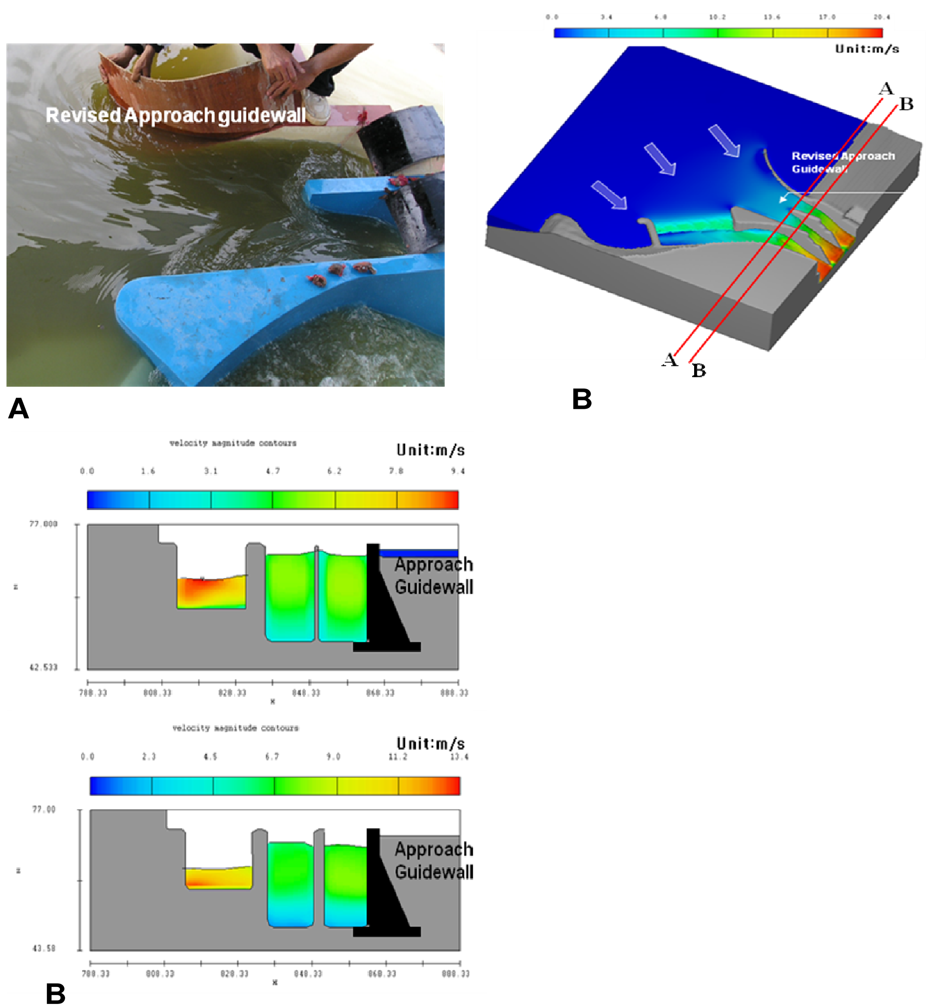

Figure 10. Flow distribution at the approach channel in PMF based on revised plan design. A. Hydarulic model test; B. Numerical simulation; C. Section view.

REFERENCES

Betts PL (1979). A variation principle in terms of stream function for free surface flows and its application to finite element method. Comp. Fluids, 7(2): 145-153.

Cassidy JJ (1965). Irrotational flow over spillways of finite height. J. Eng. Mech. Div. ASCE., 91(6): 155-173.

Flow Science (2002). FLOW-3D -Theory manual. Los Alamos, NM.

Guo Y, Wen X, Wu C, Fang D (1998). Numerical modeling of spillway flow with free drop and initially unknown discharge. J. Hydraulic Res. IAHR, 36(5): 785-801.

Ho DKH, Donohoo SM (2001). Investigation of spillway behavior under increased maximum flood by computational fluid dynamics technique. Proceeding 14th Australasian Fluid Mech. Conference, Adelaide University, Adelaide, Australia, pp. 10-14.

Ikegawa M, Washizu K (1973). Finite element method applied to analysis of flow over a spillway crest. Int. J. Numerical Methods Eng., 6: 179-189.

Kim DG, Park JH (2005). Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model. J. Civil Eng. KSCE., pp. 161-169.

KRA, KWATER (2006). Feasibility study and detail design of the Karian dam project. Indonesia.

Li W, Xie Q, Chen CJ (1989). Finite analytic solution of flow over spillways, J. Eng. Mech. ASCE, 115(2): 2645-2648.

Olsen NR, Kjellesvig HM (1998).Three-dimensional numerical flow modeling for estimation of spillway capacity. J. Hydraulic Res. IAHR., 36(5): 775-784.

Savage BM, Johnson MC (2001). Flow over ogee spillway: Physical and numerical model case study. J. Hydraulic Eng. ASCE., 127(8): 640-649.

Tabbara M, Chatial J, Awwad R (2005). Computational simulation of flow over stepped spillways. Comput. Structure, 83: 2215-2224.