Int. J. Struct. & Civil Engg. Res. 2013

International Journal of Structural and Civil Engineering Research

ISSN 2319 – 6009 www.ijscer.com
Vol. 2, No. 3, August 2013

Research Paper

NUMERICAL SIMULATION OF TWO-PHASE FLOW OVER MANDALI DAM OGEE SPILLWAY

Riyadh Zuhair Al Zubaidy[1] and Shaymaa Abdul Muttaleb Hashim Alhashimi[1]*

*Corresponding Author: **Shaymaa Abdul MuttalebHashimAlhashimi,** ✉ shyabed @yahoo.com

Spillway flows are essentially rapidly varying flows near the crest with pronounced curvature of the streamlines in the vertical direction. Spillway hydrodynamics can be obtained through physical modeling or numerical modeling. Physical modeling of spillways is expensive, time-consuming and many difficulties associated with scaling effects and measurement devices. Nowadays, using computational fluid dynamic (CFD) codes, flow behavior in the hydraulic structures can be investigated in reasonable time and expense. Due to the complexity of air-water two-phase flow over ogee spillways, a numerical study of the free surface flow on an ogee-crested is presented. A numerical model which solves the RANS equations coupled to a surface-capturing algorithm to predict the flow in air and water was developed using the finite volume module of the FLUENT software. Two types of multiphase flow models are used: a mixture multiphase flow model (MMF) and a volume of fluid model (VOF). The differences between both models are the phases interpenetrating and the phase velocities. In both models, the RNG k-ε model is chosen to simulate turbulence with the PISO arithmetic technique. The free surface numerical model is applied on a Mandali Damogee spillway as a case study to investigate the hydraulic characteristics of flow over spillway crest profiles by predicting the velocity distribution, pressure distribution and discharge characteristics..Results from several runs of software compared with corresponding experimental data. It is shown that there is a close agreement for the discharge capacity over the spillway between physical model and FLUENT predictions. The data series obtained for model comparison include; velocity profiles, pressure distribution, and characteristics of flow. Both models can satisfactorily simulate the flow pattern and the recirculation regions. The velocity profiles are more accurately simulated using the VOF model than MMF model.

Keywords: Ogee Spillway, Finite volume method, CFD modeling, FLUENT software, Flow velocity, Discharge capacity

INTRODUCTION

One of the important structures, which is required to be built at the same time of constructing dam and enables the discharge

[1] Water Recourses Engineering Department, College of Engineering, University of Baghdad, Iraq.

[2] Highway Engineering Department, College of Engineering, University of Al- Mustansiriya, Iraq.

of floodwater exceeding the capacity of dam, is called spillway. Another application of spillways is to control the height and volume of the water of the lake behind dam. In this state, the form and dimensions of spillway is a function of geographical and hydrological situation of the region. One of the most famous with highest application is ogee spillway. Due to its proper hydraulic properties, ogee spillway has been studies frequently.

Spillway models are important for evaluating and improving dam safety, as well as optimizing spillway design and economical operation. Traditionally, scaled down physical models have been used for validation and to collect hydraulic data. However the ability to efficiently evaluate a range of different spillway designs using physical models is limited by time, cost and resources. Modeling techniques using Computational Fluid Dynamics (CFD) are able to quickly evaluate different spillway designs. CFD has therefore played an increasing role in spillway modeling, with physical models used more often to supplement and validate simulations. In the following, the studies and researches conducted on ogee spillway and its form and properties, have been described:

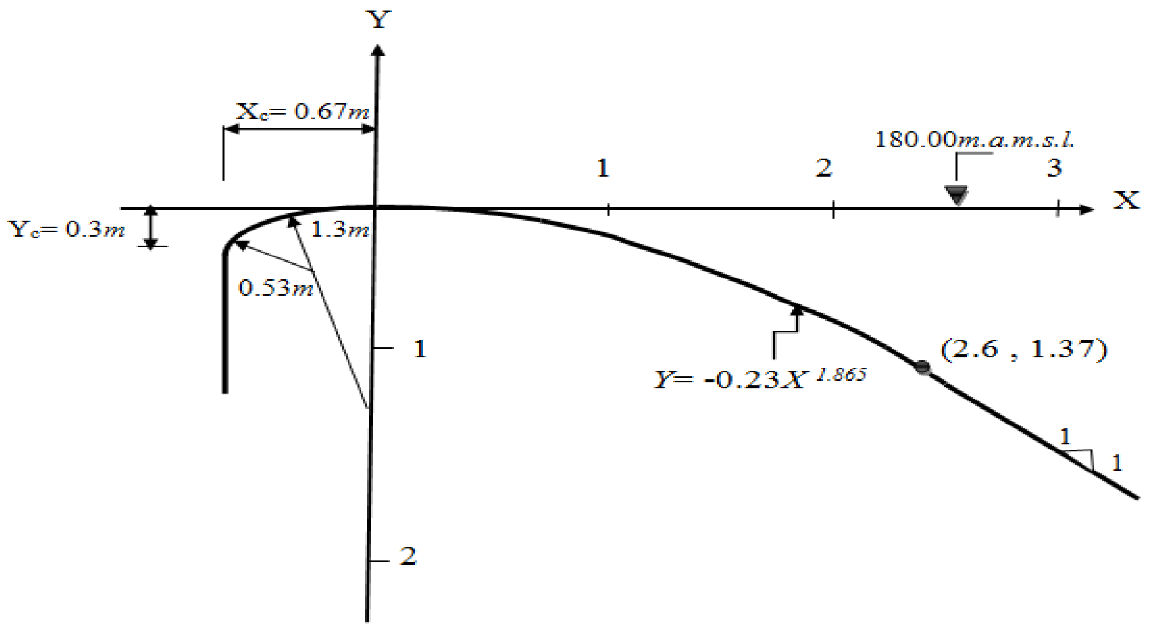
Cassidy (1965) used numerical model in a 2D space to determine the pressure on the crest of ogee spillway based on potential flow. (Olsen and Kjellesvig, 1998) applied Reynolds equations and standard k-ε model in finite volume method to analyze the flow over ogee spillway in 3D and 2D spaces. (Burgisser and Rutschmann, 1999) employed finite element method to analyze the vertical component of flow over the crest of spillway in 2D space supposing that there is incompressible and turbulent flow. (Tufi and Wilson, 2001) presented the finite difference method to analyze vertical flow over ogee spillway crest in a 2D space assuming that there is a potential flow and Neumann condition is imposed on the boundaries of flow field. (Bruce *et al.*, 2001) conducted a comprehensive study to compare the parameters of flow over standard crested ogee spillways using a physical model, numerical model and existing studies. A numerical study on turbulent flow over spillways in a 3D space was presented by (Bouhadji, 2002). In (2003, Chen *et al.*) modeled turbulent flow over stepped spillway using finite volume method. (Ho *et al.,* 2003) studied the maximum impacts of contingent floodwater over spillways. (Jean and Mazen, 2004) was modeled flow over ogee spillway numerically. (Kim and Park, 2005) used the commercial numerical model of computational fluid dynamics (Flow-3D) software to study the properties of flow including flow rate, water surface profile, pressure imposed on the crest of ogee spillway, pressure vertical distribution and speed based on the scale of model, the impact of surface roughness and details. Bhajantri*et al*. (2006) studied the hydraulic model of flow over ogee spillway numerically considering downstream and the information obtained from two physical models have been compared with the results obtained from numerical study of two crested ogee spillways. Ferrari (2009) simulated numerically the flow with free surface over a spillway with sharp crest. Daneshkhah and Vosoughifar (2011) presented a study to investigate the sensitivity analysis of meshing dimension to calculate the properly profile of flow over ogee spillway using finite volume method.Numerical analyses of different turbulent models using Fluent are applied to estimate the free surface flow over ogee spillway by Daneshkhah and Vosoughifar (2012). In this research, numerical analyses of different multiphase flow models are compared to calculate properly the profile of free surface flow over ogee spillway using finite volume method. The model was extensively validated against experimental data for flow rates at different headwater elevations, free surface elevations, and velocity distribution at the ogee surface.

Physical Model of Mandali Spillway

Mandali spillway was designed as an uncontrolled ogee weir as shown in Figure 1, with a length of 250 m and height of 10 m with a crest level at elevation of 180.0 m.a.m.s.l. Its maximum design discharge is 1724 m^3/s and the heading up over the crest at this discharge is about 2.5 m. (Rafidain State Company for Dams Construction, 2008).

For the purpose of making comparisons between the physical and numerical model results, Mandali Dam Spillway System was selected to be modeled.

Figure 1: Standard Design of Mandili Dam Weir
(*General Directorate for Dams and Reservoirs, 2006*)



SOLUTION OF NUMERICAL MODEL

In this research, Finite volume method of FLUENT software version 6.3 has been used to study the numerical performance of free surface flow over ogee spillway. This software is able to determine the free surface of flow by applying VOF model (volume of fluid model) or a MMF (mixture multiphase flow model) and assuming the effect of turbulence. VOF model was introduced by (Nichols and Hirt, 1981) to determine the common surface of air-water two-phase flow model in many hydraulic problems. In this research, 90% of the volume of cell is water and the remaining part is air. Before using this software, auxiliary software like GAMBIT was used to plot the geometry of this model. Thereafter, the output file of GAMBIT is inserted through the read case of FLUENT. This software is based on finite volume method, which is a very strong method for solving computational fluid dynamics. The equations governing flow in this software include continuity equation (law of mass conservation), and Navier-Stokes equation (momentum conservation law). In this research, Reynolds-Average method was used to solve Navier-Stokes equation in such a way that turbulent flow is inserted into the equations indirectly. Thus, continuity equations and momentum are determined.

BOUNDARY CONDITIONS

The boundaries are shown in Figure 2. The inlet section consisted of the inlet of water this software lower part and the inlet of air on the higher part upstream of the dam. The water inflow velocity can be calculated according to the discharge and the water depth at the water inlet. The air boundary was set as pressure-inlet conditions on which atmospheric pressure was assumed. Because the boundary between water and air at the downstream outlet could not be distinguished, it was defined as a pressure boundary, or free-flow condition. All of the walls were set as the stationary, non-slip wall. The viscosity layer near to the wall dealt with the wall function. Then, the segregated solver was used because it is multiphase flow with two materials, water and air, each with different velocity. The VOF and/or MMF models were used to deal with the multiphase flows.

FREE SURFACE MODELS

The Volume of Fluid Model (VOF)

The VOF formulation relies on the fact that two or more phases are not interpenetrating. In each control volume, the volume fractions of all phases sum to unity. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values.In the geometric reconstruction approach, the standard interpolation schemes are used to obtain the face fluxes whenever a cell is completely filled with one phase or another. When the cell is near the interface between two phases, the geometric reconstruction scheme is used. It assumes that the interface between two fluids has a linear slope within each cell, and uses this linear shape for calculation of the advection of fluid through the cell faces. The first step in this reconstruction scheme is calculating the position of the linear interface relative to the center of each partially-filled cell. The second step is calculating the advection amount of fluid through each face using the computed linear interface representation and information about the normal and tangential velocity distribution on the face. The third step is calculating the volume fraction in each cell using the balance of fluxes calculated during the previous step. (Yeoh and Tu, 2010).

A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum equation, Equation (1), is dependent on the volume fractions of all phases through the properties ρ and μ.

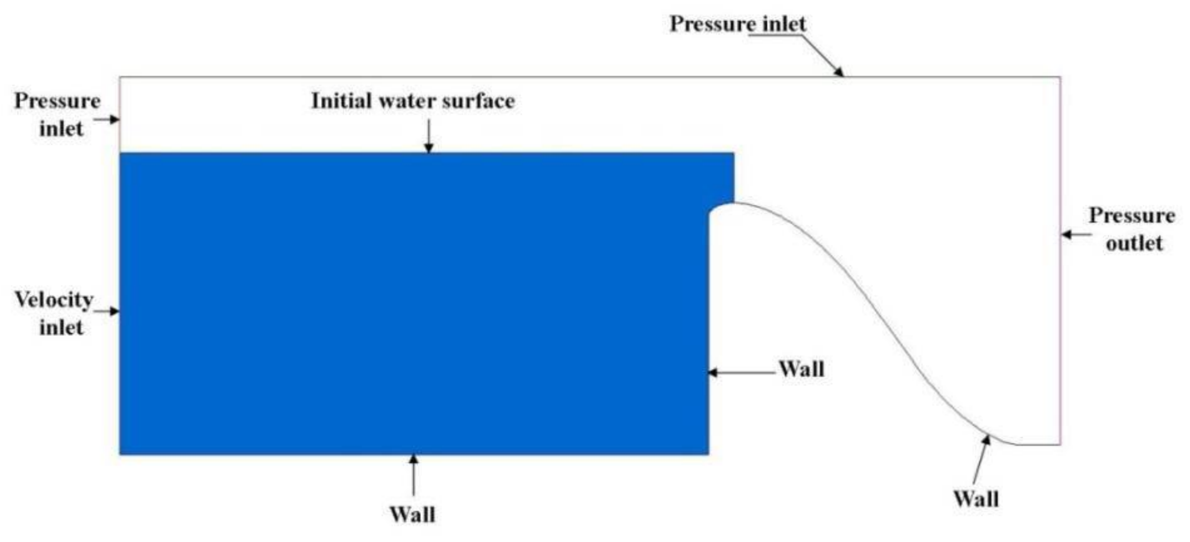
$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla.(\rho \vec{u}\,\vec{u}) = -\nabla p + \nabla\left[\mu\left(\nabla \vec{u}^{T}\right)\right] + \rho\,\vec{g} + \vec{F} \quad (1)$$

The limitation of the shared-fields approximation is that in cases where large velocitydifferences exist between the phases, the accuracy of the velocities computed near the interface can be adversely affected.

Mixture Multiphase Flow Model (MMF)

The MMF model is suggested to use for bubbly flows and pneumatic transport. It is a simplified multiphase model that can be used where the phases move at different velocities, but assume local equilibrium over short spatial length scales. The coupling between the phases should be strong. It can also be used to model homogeneous multiphase flows with very strong coupling and the phases moving at the same velocity. The mixture model can model *n* phases by the continuity equation for the mixture, the momentum equation for the mixture, and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities.

Figure 2: The Boundary Conditions of the Ogee Spillway of Simulated Numerical Model



The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\rho_m) + \nabla.(\rho_m \vec{u}_m) = 0 \qquad ...(2)$$

where the mixture density and the mixture velocity are defined as

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \qquad ...(3)$$

$$u_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \vec{u}_k \frac{\partial}{\partial t}}{\rho_m} \qquad ...(4)$$

where α_k and ρ_k are the volume fraction and density of phase *k*, respectively. The mixture velocity u_m represents the velocity of the mass center of the mixture flow. Note that ρ_m may vary even though the component densities keep constant.

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases.

$$\begin{aligned}&\frac{\partial}{\partial t}(\rho_m \vec{u}_m) + \nabla.(\rho \vec{u}_m \vec{u}_m) \\ &= -\nabla p + \nabla\left[\mu_m\left(\nabla \vec{u}_m + \vec{u}_m^{T}\right)\right] \\ &+ \nabla.\left[\sum_{k=1}^{n} \alpha_k \rho_k \vec{u}_{dr,k} \vec{u}_{dr,k}\right]\end{aligned} \qquad ...(5)$$

where $u_{dr,k}$ is the diffusion velocity for the mixture flow. The two stress tensors represent respectively the average viscous stress and diffusion stress due to the phases slip.

The differences between the two models are the manner in which they handle phase interpenetration and the phase velocities. With these two differences, the initial boundary condition must be different. The air velocity in mixture model should be set at zero and then reduced to homogeneous multiphase model while the air velocity in VOF model should be the same as water velocity. Flow over different kinds of spillways produce different patterns and have different effects.

In two models, the control-volume-based technique was used to solve the equations. The calculation domain was divided into discrete control volumes by the unstructured grid which has a high flexibility to fit the complex geometry and boundary of ogee spillway. For operating conditions, the specified operating density, 1.225 kg/m^3, was used with gravitational acceleration, –9.81 m/s^2, and operating pressure 101,325 Pa. The boundary conditions were set by using water velocity at water inlet. The RNG k-ε model, with non-equilibrium wall functions, was used to simulate turbulence. The default values of model constants were used.

It is a relatively recent development from the standard *k*-model is RNG *k*-model.The RNG turbulence model solves for turbulent kinetic energy (k) and turbulent kinetic energy dissipation rate (ε). The RNG-based models rely less on empirical constants while setting a framework for the derivation of a range of parameters to be used at different turbulence scales (Yakhot and Smith, 1992).

The RNG-based k-ε turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called "renormalization group" (RNG) method. The *k*-ε equations are as follows:

$$\frac{\partial}{\partial t}(\rho k) + \nabla.(\rho \vec{u} k) = \nabla.\left[\frac{\mu_{eff}}{\sigma_k} \nabla_k\right] G_k - \rho \varepsilon \qquad ...(6)$$

$$\frac{\partial}{\partial t}(\rho\,\varepsilon)+\nabla.(\rho\,\vec{u}\,\varepsilon)$$

$$=\nabla.\left[\frac{\mu_{eff}}{\sigma_{\varepsilon}}\nabla_{\varepsilon}\right]+C_{1s}-G_k-C^{*}_{2\varepsilon}\rho\frac{\varepsilon^{2}}{k} \qquad ...(7)$$

where k is the turbulence kinetic energy and ε is referred to dissipation rate of k.

$$\mu_{eff}=\mu\left[1+\sqrt{\frac{C_{\mu}}{\mu}\frac{k}{\sqrt{\varepsilon}}}\right]^{2} \qquad ...(8)$$

where μ is the molecular viscosity of mixture. G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as:

$$G_k=\mu_t\left[\nabla\rho\,\vec{u}+(\nabla\,\vec{u})^{T}\right]\nabla\,\vec{u} \qquad ...(9)$$

C^{*}_{2s} is given by

$$C^{*}_{2s}=C_{2s}+\frac{C_{\mu}\rho\eta^{3}(1-\eta/\eta_0)}{1+\beta\eta^{3}} \qquad ...(10)$$

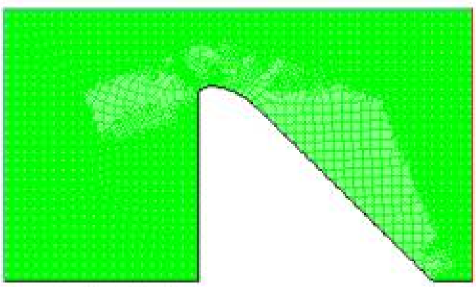
$$\eta=S\frac{k}{\varepsilon},\; s=\sqrt{2S_{ij}S_{ij}}$$

is the modulus of mean rate of strain tensor

The values of the constants in above equations are $C\mu$ = 0.0845, $C_{1\varepsilon}$ =1.42, $C_{2\varepsilon}$ = 1.68, $\sigma_k = \sigma_\varepsilon$ = 0.75, η_0 = 4.38, β = 0.012

The convective fluxes in the mean volume fraction, momentum and turbulence closure equations were discretized by employing a conservative, second-order accurate upwind scheme. The pressure-velocity coupling algorithm is the Pressure-Implicit with Splitting of Operators (PISO), which is based on the higher degree of the approximate relation between the corrections for pressure and velocity and may also be useful for transient calculations on highly skewed meshes (Ferzieger and Peric,1996). The under-relaxation factors are used in the pressure-based solver to stabilize the convergence behavior of the outer nonlinear iterations by introducing selective amounts of ϕ in the system of discretized equations. Four under-relaxation factors, i.e., pressure coefficient, momentum coefficient, k and ε are set to default values, 0.3, 0.7 0.7, and 0.8, respectively. The default values are near optimal for the largest possible number of cases. For the time step size, 0.5 s was used

Figure 3: Meshing and Its Distribution in the Model of Ogee



for every case. The implicit equation can be solved iteratively at each time level before moving to the next time step. The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size. However, the Courant number that is used for stability checking is set at 0.25 which is less than 1.0 and stable.

Validation of Numerical Model

In this research, it is required that the accuracy of the results obtained from numerical study to be verified. Before such verification, the validity of the selection of multiphase flow model, the accuracy of meshing and the fact that they have no impact on the results shall be ensured.Since the geometry was too complex to generate a fully structured grid with good quality complex geometry regions were generated using unstructured grids with quad-pave meshing as it is shown in the Figure 3.

To have access to an appropriate two-phase flow model for calculating the parameters of flow over ogee spillway, numerical model with meshing of 0.05 m was taken into consideration within 70 s and step time of 0.5 s. By studying different models of two-phase under the same conditions, the best and worst simulated flow profile can be accessed in

Figure 4: Flow Profile in VOF Multiphase Model

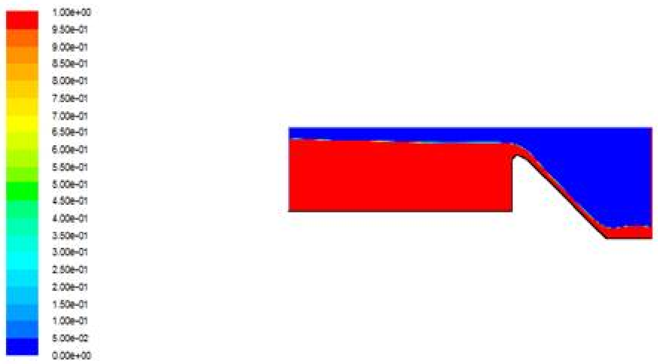
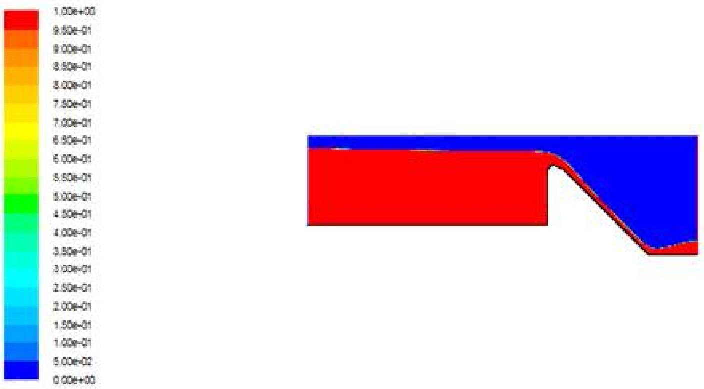


Figure 5: Flow Profile in MMF Multiphase Model



comparison with physical model, which have been shown in Figures 4 and 5.

The study of the flow profile at different multiphase models based on Figures 4 and 5 indicate that the flow profile obtained from VOF model is more similar to physical model, and in contrary the MMF model has the minimum similarity with physical model.

The profile of flow over spillway resulted from VOF matches the physical model very well. It can be claimed that this model is an appropriate choice for modeling flow over ogee spillways.

Simulation Conditions

The numerical calculations were performed for different headwater elevations to control the discharge. The numerical model was constructed at prototype scale in order to obtain valid predicted stresses. Seven different headwater elevations were considered,including the minimum elevation (180.20 m), the highest expected elevation (182.4 m), and five elevations in between (182.03 m, 181.95 m, 181.78 m, 181.4 m and 180.85 m).

The grid quality was critical to achieve convergence. Thus, grid quality requirements forced the use of a specifically designed grid for each condition. This was achieved by initially running a coarse, high-quality grid to estimate the location of the free surface and then redesigning the grid to match the free surface location. For all cases, the headwater elevation was fixed by setting the inlet boundary conditions. The flow rate and the overall free surface topology were then predicted based solely on the operational conditions.

NUMERICAL RESULTS AND ANALYSIS

The numerical model was extensively validated against the experimental data taken on

Table 1: Mandali Discharge Comparison

Water Elev. (m)	Phys. Model (m^3/s)	Numerical Model (VOF)		Numerical Model (MMF)	
		Flowrate (m^3/s)	Diff. (%)	Flowrate (m^3/s)	Diff. (%)
182.3	1803.12	1856.15	2.9	1891.51	4.7
182.03	1473.08	1511.44	2.5	1549.63	4.9
181.95	1359.7	1397.51	2.7	1425.12	4.5
181.78	1178.57	1201.21	1.9	1221.96	3.6
181.4	813.17	831.08	2.2	840.5	3.2
180.85	367.7	372.9	1.4	379.07	2.9
180.53	163.52	166.17	1.6	168.95	3.2

thephysical model of Mandali spillway. The experimental data was scaled to prototype equivalents (based on Froude scaling laws) and compared to numerical model predictions. Validation included comparing discharge, free surface elevation, and velocity distribution over

Figure 6: The Rating Curve of MandaliWeir Spillway

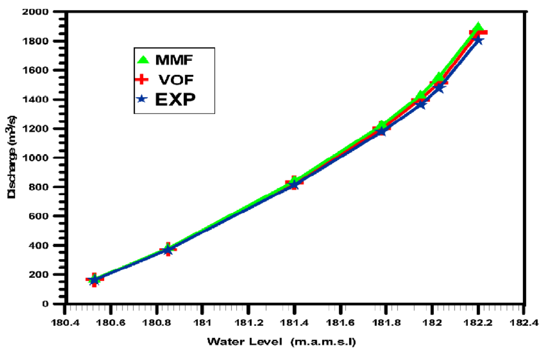


Figure 7: VelocityDistribution overOgee Spillway

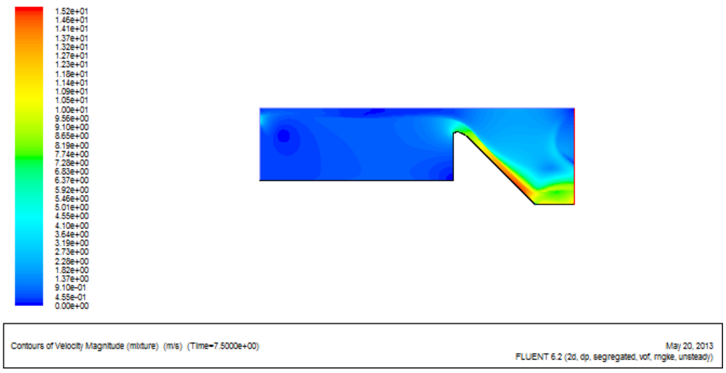
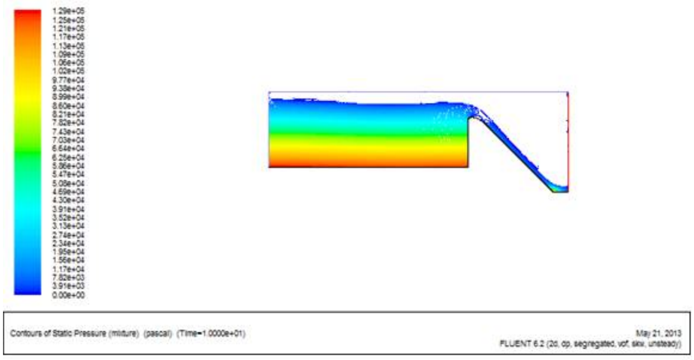


Figure 8: Pressure ContoursoverOgee Spillway



ogee spillway between numerical model and experimental measurements.

The rating curves combined different headwater elevations each headwater elevation resulted in a converged discharge (to within 0.03%) predicted by the VOF model and (to within 0.05%) predicted by MMF model. Figure 6 shows a comparison between the experimental data and the numerical model predictionswith relative differences provided in Table 1.

As seen in Figure 6, the agreement between the measured and predicted discharges was excellent, with the difference between predicted and measured discharges never exceeding 3% for VOF and 5% for MMF model. These differences were likely due to some approximations made on the numerical model, and to small errors in head water elevation control, caused by the spill region height at the inlet boundary.

Figure 7 shows the calculated velocity distribution along the ogee spillway for the discharge of 1178.57 m^3/s. Actually, the flow velocities increase in the upper parts of the ogee spillway but tend to a constant value within the uniform flow region towards the end of the chute.

The capability of the software to produce pressuremeasurements along the spillway that follow the general trend of physical model experimental results has also been displayed. Figure 8 indicate the pressure contours over spillway for discharge of 1359.7 m^3/s.

CONCLUSION

A finite-volume code, which solves two dimensional Navier-Stokes equations, was used in the numerical simulations. This code has been widely used in several studies that involve CFD modeling of hydraulic structures such as spillways. The grid size with quadrilateral-pave meshes of 0.05 × 0.05 m^2 was used.A numerical model using different multiphase flow models, VOF and MMF model together with RNG k-ε turbulence model was implemented to develop a numerical model to simulate free surface flow over an ogee-crested spillway.The data from large-scale experiments of Mandali spillway are used to calibrate and verify the model.

The numerical model was validated against measured discharges and free surface elevations at seven headwater elevations for flows over spillway. The comparison of the numerical model results with experimental data showed excellent agreement. The free surface flow fromVOF numerical model proved to be useful as a design aid tool.Also, velocity and pressure distributions were evaluated over spillway accurately.

Furthermore, this study shows that CFD can be used as a design tool of hydraulic structures together with proper experimental analysis for validation. Many more cases could be easily tested with the numerical simulations, which provide detailed information of the flow velocity, pressure among other characteristics. Thus, the numerical model showed to have a significant advantage in practice, in terms of parametric studies.

REFRENCES

1. Bhajantri M R and Eldho T I and Deolalikar P B (2006), "Hydrodynamic Modeling of Flow Over a Spillway Using

a Two Dimensional Finite Volume Based Numerical Model", *Sadhana*, pp 743-754.

2. Bouhadji Latif (2002), "Three Dimensional Numerical Simulation of Turbulent Flow Over Spillways", *Asl-Aqflow Inc., Sidney*, British Columbia, Canada.

3. Bruce M, Savage Michael, Johnson C Members and Asce (2001), "Flow Over Ogee Spillway:Physical and Numerical Model Case Study", *J.Hydr.Eng.Asce*, Vol. 127, No. 8, pp. 640-649.

4. Burgisser M F and Rutschmann P (1999), "Numerical Solution of Viscous 2D Vertical Free Surface Flows:Flow Over Spillway Crests", *Proc, 28th Iahr Congr.*, Technical University Graz, Graz, Austria.

5. Cassidy (1965), "Irrotational Flow Over Spillways of Finite Height," *J. Eng. Mech. Division*, Vol. 91, No. 6, pp. 55-173.

6. Chen Q and Dai G and Liu H (2003), "Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Overflow", *J. hydr. Eng.*, Vol.128, No. 7, pp 683-688.

7. Daneshkhah A and Vosoughifar H (2011), "Numerical Investigation of Passed Flow Different Parameters over a Standard Ogee Spillway to Satisfy Flow Profile in CFD Method", *5th Symposium on Advances in Science and Technology*, Iran.

8. Daneshkhah A and Vosoughifar H (2012), "Solution of Flow Field Equations to Investigate the Best Turbulent Modelof Flow over a Standard Ogee Spillway in Finite Volume Method", *The First International Conference on Dams and Hydropower*, Iran.

9. Ferrari A (2009), "SPH Simulation of Free Surface Flow Over a Sharp Crested Weir", Department of Civil and Environmental Engineering, University of Trento, Via Mesiano 77, I 38100 Trento, Italy.

10. Ferzieger J L and Peric M (1996), *Computational Methods for Fluid Dynamics*, Heidelberg: Springer-Verlag,.

11. Fluent Inc, (2006), "User s guide", Version, 6.3.26 for Estimation of Spillway Capacity", *J. Hydr. Res.*, Vol. 36, No. 5, pp. 775-784.

12. General Directorate for Dams and Reservoirs (2006), "Mandali Dam Project", *Design Report*, Ministry of Water Resources.

13. Ho D, Boyes K, Donohoo S and Cooper, B (2003), "Numerical Flow Analysis for Spillways," 43rd Ancold Conference, Hobart, Tasmania, pp. 24-29.

14. Jean C and Mazen T (2004), "Computational Modeling of Flow Over an Ogee Spillway,"Department of Civil Engineering, School of Engineering and Architecture, Lebanese American University, P O Box 36, Byblos, Lebanon.

15. Kim D G and Park J H (2005), "Analysis of Flow Structure Over Ogee-Spillway in Consideration of Scale and Roughness Effects by Using CFD Model", *Journal of Civil Engineering*, Vol. 9, No.2, pp 161-169.

16. Nichols B D and Hirt C W (1981), "Volume of fluid (VOF) method for the dynamics of free boundaries", *J. Comp. Phys.*, Vol. 39, pp. 201-225.

17. Olsen N R B and Kjellesvig H M (1998), "Three Dimentional Numerical Flow Modeling for Estimation of Spillway Capacity", *J. of Hydr. Research*, Vol. 36, No. 5, pp. 775-784.

18. Rafidain State Company for Dams Construction, (2008), "Hydraulic Model Study ofMandali Dam Spillway", Final Report, Ministry of Water Resources.

19. Tufi M Assy (2001), "Solution for Spillway Flow by Finite Difference Method", *J. Hydr. Res.*, Vol. 39, No. 3, pp. 241-247.

20. Yakhot V and Smith L M (1992), "The Renormalization Group, the E-expansion and Derivation of Turbulence Models", *Journal of Scientific Computing*, Vol. 7, pp. 35-61.

21. Yeoh G H and Tu J (2010), *Computational techniques for multiphase flows*, Elsevier Ltd., USA, pp. 407-500.