COMPARISON OF PHYSICAL MODELING AND CFD SIMULATION OF FLOW OVER SPILLWAY IN THE KAVSAK DAM

Mehmet Ali Kokpinar [1], Şerife Yurdagul Kumcu[2], & Izzet Ozgun Guler [3]

[1] Hydraulic Model Laboratory, State Hydraulic Works, Turkey, 06100 Yucetepe-Ankara, Turkey
e-mail: mkokpinar@superonline.com

[2]Civil Engineering Department, Konya University, Karacigan Mahallesi Ankara Caddesi No:74 Karatay, Konya, Turkey
e-mail: yurdagulkumcu@gmail.com

[3] IOG Engineering, Turkey, Yüksel Cad. Yüksel Apt. 33/6 Kızılay-Ankara, Turkey
e-mail: info@iog.com.tr

Abstract

A comparative study was made for flow over a spillway structure using results obtained from physical modeling and computational fluid dynamics (CFD) simulation. The Kavsak Dam spillway which is located on the Seyhan River in south of Turkey, was selected to make this comparison. The dam consists of a 52 m high RCC dam body, three radial gated spillway chute located on the dam body, 80 m-long stilling basin, and 180 MW power house. The 1/50-scaled physical model was used in conducting experiments. Pressure taps were installed along the entire length of the spillway chute and stilling basin. Flow depth, discharge and pressure data were recorded for 3 different flow conditions as; low, mid, and high. A commercially available CFD program, which solves the Reynolds-averaged Navier-Stokes (RANS) equations, was used to model the numerical model setup. Discharge rating curves and pressures are used to compare the results of the physical model and the numerical model. It is shown that there is reasonably good agreement between the physical and numerical models for both pressures and discharges. The absolute percent differences between physical modeling and CFD simulation results in discharge estimation for Q_{1000} and Q_{10000} are found as 3.70% and 2.30%, respectively. It is seen that numerical methods can be used to provide an effective tool in the design and analysis of spillway flows.

Introduction

Hydraulic design of a spillway and a stilling basin has been one of the most studied subjects in hydraulic engineering. Properly designed spillways and stilling basins will be able to pass flood flows efficiently and safely to downstream of dams. A hydraulic model is still a precision device for the experimental investigation of flow over a spillway structure, which can give reliable information only if it is designed correctly.

With increasing computer processing capacity, numerical simulations for hydrodynamic processes become attractive, including flow over spillways. A comparison of these numerical results with experimental or prototype data is still required for calibration and validation. Computational Fluid Dynamics (CFD) is a branch of numerical modeling that has been developed for solving problems involving fluid flow. This includes applications involving fluid-solid interaction, such as the flow of water in a river or over and around hydraulics structures. There is therefore considerable interest on the part of hydraulic engineers into the applicability of CFD to model fluid flow at hydro-electric generating stations. Although CFD can take a significant amount of computation time, it can provide 3-dimensional flow fields around curved objects as well as other flow detail not available in more simplified 1 or 2 dimensional models. Despite the fact that CFD is being utilized for modeling flow in all areas of a generating station, this study will focus on the use of CFD to model the flow of water through spillways.

This paper provides information on how accurately a commercially available computational fluid dynamic (CFD) model can predict the spillway discharge capacity and pressure distribution along the spillway bottom surface.

Physical Model

A 1/50-scaled physical model of the Kavsak Dam spillway and stilling basin, as shown in Figures 1 and 2, was built and tested at the Hydraulic Model Laboratory of State Hydraulic Works of Turkey (DSI). The model was constructed of Plexiglas and was fabricated to conform to the distinctive shape of an ogee crest. The spillway has 45.8 m in width and 57 m long with a bottom slope of 125%. The length of the stilling basin is about 90 m.

During model tests, flow rates were measured with an ultrasonic flow meter. Pressures on the spillway were measured using a piezometers board reading provided the

average pressure reading at each pressure tap location. Both the upstream reservoir lake level and downstream tailwater elevations were measured using piezometers. A control valve was used to set the flow in the physical model. The model was operated at four different upstream reservoir elevations as given in Table 1. The 3rd and 4th runs in Table 1 belong to corresponding Q_{1000} and Q_{10000} discharge values of the project. The downstream tailwater elevations was adjusted by another control gate located far downstream of the model.

Table 1: Physical and CFD models upstream and downstream operating conditions (prototype scale)

Run	Upstream reservoir elevation (m)	Downstream tailwater elevation (m)
1	216.98	168.00
2	221.58	174.50
3	224.79	178.90
4	227.36	182.55

Figure 1: View of the 1/50-scaled physical model (original design)

Figure 2: The physical model under operation (Q=3856 m^3/s-original design)

Numerical Simulation

The commercially-available CFD package FLOW-3D Version 10.0 was used in the simulation of the flow field. The CFD package applies finite-volume method to solve the RANS equations. Free surfaces are modeled with the Volume of Fluid (VOF) technique, which was first reported in Nichols and Hirt (1975), and more completely in Hirt and Nichols (1981). Trademarked as TruVOF, this technique is one of the defining features of the program and provides three important functions for free surface flow: location and orientation of free surfaces within computational cells, tracking of free surface motion through cells, and a boundary condition applied at the free surface interface.

The location of the flow obstacles is evaluated by the program implementing a cell porosity technique called the fractional area/volume obstacle representation of FAVOR method (Hirt 1992). The free surface was computed using a modified volume-of-fluid method (Hirt and Nichols 1981). For each cell, the program calculates average values for the flow parameters (pressures and velocities) at discrete times using staggered grid technique (Vesteeg and Malalasekera 1996).

FLOW-3D version V10.0 was used to simulate flow over the Kavsak Dam along with the renormalized group

turbulence model. A rectangular grid was defined in the computation domain shown in Figure 3. Total number of grid cells was approximately 6.24E+06 in which only 4.36E+06 of them were active. The corresponding uniform mesh size used in meshing was $\Delta x=\Delta y=\Delta z=0.5$ m.

To simulate given flow, it is important that the boundary conditions accurately represent what is physically occurring. Because the flow is defined in Cartesian coordinates, there are six different boundaries on the computational mesh domain. The boundary conditions on the mesh were set as follows: sidewalls y-no slip/wall; top z-pressure boundary with gauge pressure equal to zero (atmospheric); bottom z-no slip/wall; left x-local stagnation pressure based on upstream reservoir elevation; and right x-local static pressure based on downstream tailwater elevation.
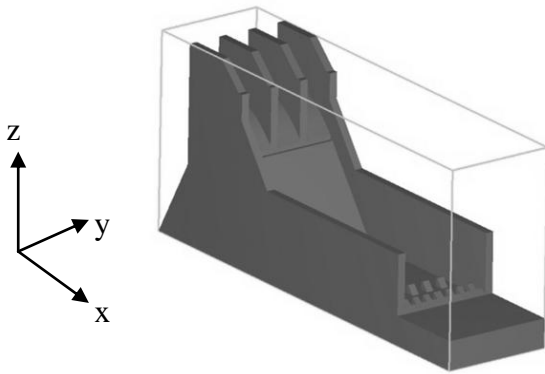

Figure 3: Solid model of the Kavsak Dam Spillway used in the CFD simulations (final design)

The inflow and outflow boundary conditions (left and right boundaries, x-direction) can be computed by employing a hydrostatic pressure distribution throughout the flow depth. In running the FLOW-3D CFD software, computation modules of *viscosity and turbulence*, *gravity*, *air-entrainment*, and *density evaluation* were activated for all cases studied. Since there are no prototype data available for comparison to the CFD solution, the data from the physical model have been scaled to prototype dimensions.

Discussion of Results

The main purpose of this study was to compare results from a physical model with that of a CFD model for flow over an ogee crest spillway and through stilling basin. The flow rates over the spillway crest and free surface elevations, depth-averaged velocity distributions, and the pressures acting on the crest and on the stilling basin are used to compare the differences between the physical model and the CFD model. The existing Kavsak Dam physical model data have been used as a baseline of this comparison (Kumcu, 2010).

Table 2 shows the physical model measured flow rates (Q_{PM}) and the numerically calculated flow rates from the CFD model (Q_{CFD}). The results have been normalized to allow a comparison in their simplest form in Figure 4. The 10000 years return period parameters, $(H_0)_{10000}=16.46$ m and $Q_{10000}=5053$ m^3/s, from physical model are used as the basis. In Figure 4 the static head above crest, H_0, is normalized by the $(H_0)_{10000}$ and shown in the abscissa. The discharge Q is normalized by Q_{10000} and shown on the ordinate. Using the physical model and its discharge as observed standard, the relative percent difference in discharge is calculated in Table 2. The relative percent difference at a given $(H_0)/(H_0)_{10000}$ is defined as $(Q_{CFD}-Q_{PM})/Q_{PM}\ x100$ and shows that the CFD model agrees within 3.2% in average with the physical model.
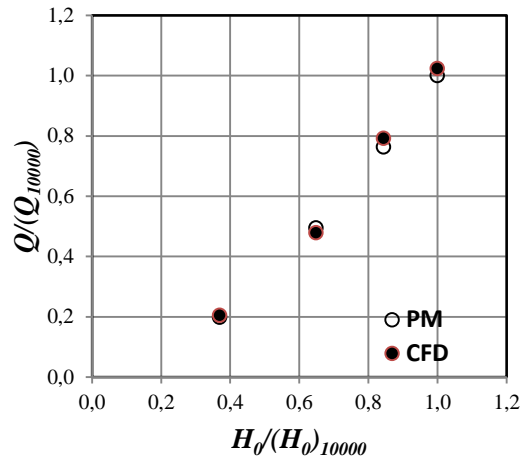

Figure 4: Comparison between the physical model (PM) and the numerical model (CFD) predictions for flow rates over spillway

Table 2: Comparison of observed flow rate versus computed flow rate (prototype scale)

Run	Q_{PM} (m^3/s)	Q_{CFD} (m^3/s)	Percent difference
1	1000	1034	3.4
2	2500	2415	3.4
3	3856	4001	3.7
4	5053	5170	2.3

The data for $Q_{10000}=5053$ m^3/s in the physical model was used for the comparison of free surface elevation between the physical model and the CFD model as seen in Figure 5. Since similar results were obtained, other simulation plots and comparison with physical model data will not be given here. In the comparison, free surface data is plotted in elevation where the crest is at 300 m above the sea level. As

seen in the figure, the majority of the points overlap exceptionally well, while only the hydraulic jump roller region on the profile seems to exhibit any notable error. This is due to the difficulties of both CFD modeling and measurement in physical model in accounting effects of strong turbulence at the hydraulic jump region and flow aeration with related consequences on bulking of flow depth.
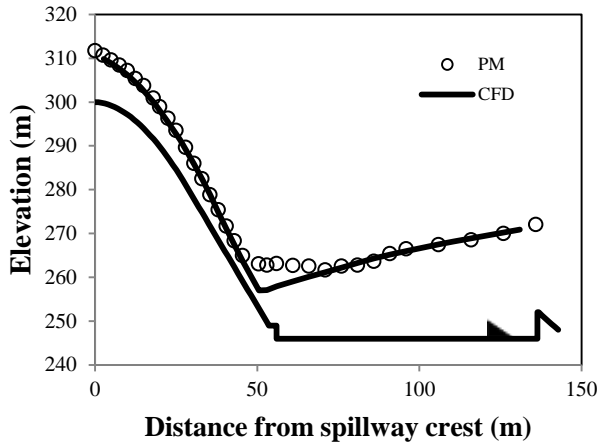

Figure 5: Comparison of free surface elevations between physical model and CFD model for Q=5053 m^3/s

Figures 6 shows 2D view of depth-averaged velocity contours obtained from the CFD model for the flow rate of Q=5053 m^3/s. Since most of the free surface elevation data of physical model overlaps the CFD model data, the depth-averaged velocity values of both models also show similarities. The maximum value of depth-averaged velocity was found as approxiamtely 32 m/sec which creates a potential risk for cavitation damage.
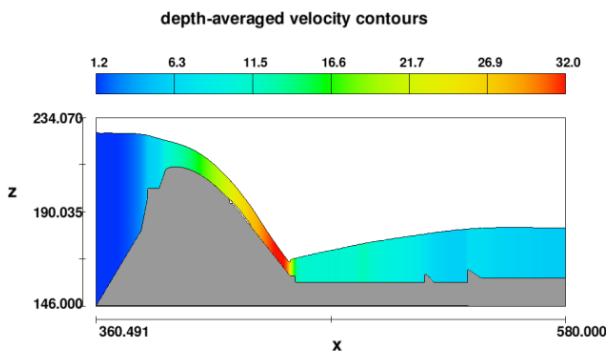

Figure 6: CFD solution of 2D depth-averaged velocity distribution along the spillway structure for Q_{10000}=5053 m^3/s (velocity values are in m/s)

The distiribution of air antrainment rate obtained from the CFD model data along with the flow over spillway and through stilling basin for the flow rate of Q=5053 m^3/s was given in Figure 7. According to the Figure 7, the bottom surface of the spillway downstrean of the aerator structure, where potentially under the risk of cavitation damage, is sufficiently aerated. Since the value of 1-3% of air concentration can be generally accepted as a critical value for the prevention of cavitation damage, CFD results promise always more than 10% of air conenration value.
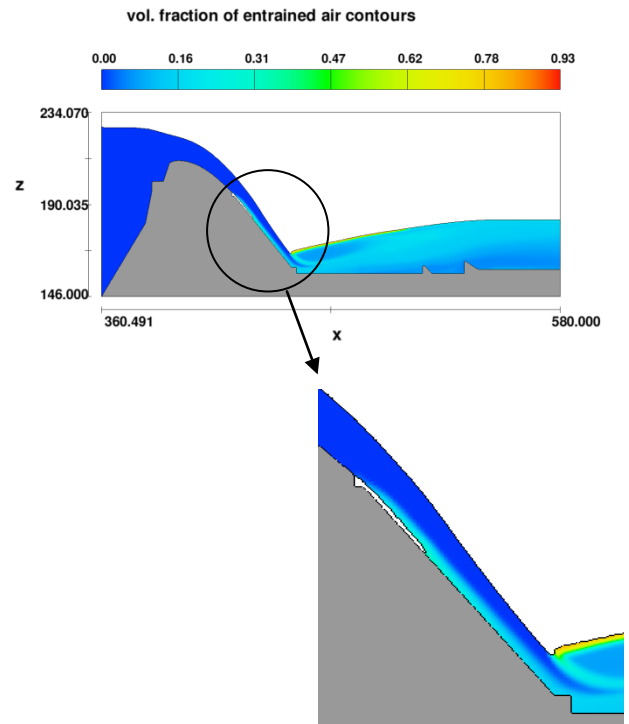

Figure 7: 2D-CFD solution of volume fraction of entrained air contours along the spillway structure for Q_{10000}=5053 m^3/s.

With the horizontal distance starting from the crest axis The bottom pressure distributions (in Pascal) along the spillway and stilling basin have been shown on Figures 8 to 11. Figures 8 to 11 provide a comparison of spillway and stilling basin average pressures for four different flow rate conditions on the physical model as; Q=1000 m^3/s, 2500 m^3/s, 3856 m^3/s, and 5053 m^3/s. Pressures from the CFD model compared quite favorably with the scaled physical model data with the exception of pressure data obtained around baffle blocks located at the longitudinal distances of 50.5 m and 86 m from the spillway crest. On prototype scale, the maximum absolute pressure difference was predicted at the longitudinal distance of 86 m from the crest as ΔH=19.2 m for Q_{10000}=5053 m^3/s (Fig. 11). The possible source of error was considered from the selection of

uniform mesh size as $\Delta x = \Delta y = \Delta z = 0.5$ m throughout the computation domain. For a finer meshing with nested mesh blocks (e.g. $\Delta x = \Delta y = \Delta z = 0.25$ m or finer) better predictions around baffle blocks could be expected from the CFD model that will be part of the another subsequent research study.

The data presented in Figs. 8 to 11 demonstrate that CFD modeling is capable of reasonably predicting pressures on spillways and stilling basins. The concern of modeling supercritical flow transitioning to subcritical flow has been still a difficult problem to solve, however numerical advances are rapidly reducing the inherent difficulties of this problem (Johnson and Savage, 2006).
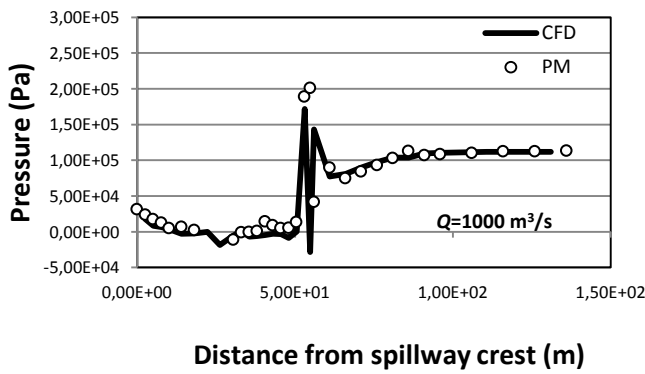

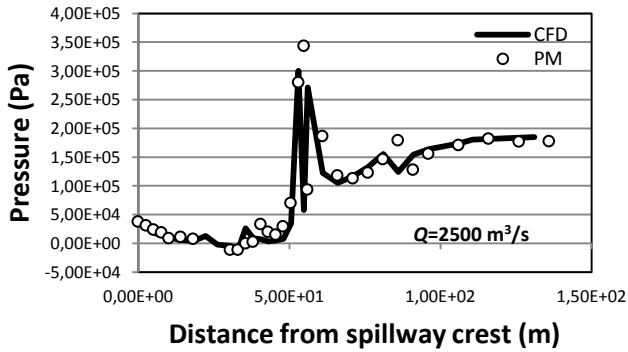Figure 8: Comparison of CFD and physical model pressures for $Q=1000$ m³/s.

Figure 9: Comparison of CFD and physical model pressures for $Q=2500$ m³/s.
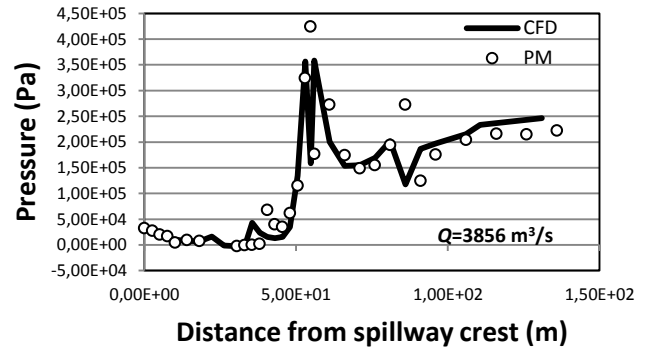

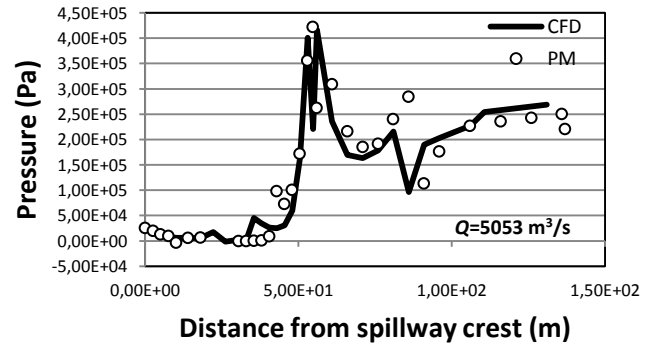Figure 10: Comparison of CFD and physical model pressures for $Q=3856$ m³/s.

Figure 11: Comparison of CFD and physical model pressures for $Q=5053$ m³/s.

Conclusions

In this study an attempt was made to simulate flow over a spillway structure using commercially available CFD software. Obtained results from the CFD model was compared to existing physical model data of the Kavsak Dam and HPP.

The flow rate results show that the CFD model provided a reasonable solution. The average relative percent difference between the CFD model and the physical model was obtained as 3.2 %.

The CFD results obtained for free surface elevation and depth-averaged velocity fit generally the physical model data, whereas some difficulties observed at the flow transition from supercritical to subcritical through the hydraulic jump region mainly due to effects of high turbulence and flow bulking.

Although numerical methods offer a potential to provide solutions with increasing accuracy, physical model studies are still considered as the basis from which all other solution methods used.

References

Hirt, C.W. and Nichols, B.D. (1981). "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," Journal of Computational Physics 39, 201.

Hirt C.W. (1992). "Volume-fraction techniques: Powerfull tools for flow modeling," Flow Science Rep. No. FSI-92-00-02, Flow Science, Inc., Santa Fe, N.M.

Johnson, M.C. and Savage, B.M. (2006). "Physical and numerical comparison of flow over Ogee spillway in the presence of tailwater". ASCE-Journal of Hydraulic Engineering, Vol. 132, No. 12, 1353-1357.

Kumcu, S.Y. (2010). "Hydraulic Model Studies of Kavsak Dam and HPP," Report No: Hi-1005. DSI-TAKK Publications (in Turkish).

Nichols, B.D. and Hirt, C.W. (1975). "Methods for Calculating Multi-Dimensional, Transient Free Surface Flows Past Bodies," Proc. First Intern. Conf. Num. Ship Hydrodynamics, Gaithersburg, ML, Oct. 20-23.

Savage, B.M. and Johnson, M.C. (2001). "Flow over Ogee spillway: Physical and numerical model case study, " ASCE-Journal of Hydraulic Engineering, Vol. 127, No. 8, 640-649.

Vesteeg and Malalasekera (1996). *An Introduction to Computational Fluid Dynamics*, Longman Scientific and Technical, New York.