HYDRAULIC PERFORMANCE ASSESSMENT OF AN ORIFICE SPILLWAYS USING CFD MODELING

Zohaib Nisar[1], Muhammad Kaleem Sarwar[2], Ghulam Nabi[3]

1. *Research fellow and corresponding author, Centre of Excellence in Water Resources Engineering, University of Engineering and Technology Lahore, Pakistan. E-mail: engrzohaibnisar@hotmail.com, Fax No. 92-42 99250259*
2. *Assistant Professor, Centre of Excellence in Water Resources Engineering, University of Engineering and Technology Lahore, Pakistan. E-mail: engr_Kaleem@yahoo.com*
3. *Assistant Professor, Centre of Excellence in Water Resources Engineering, University of Engineering and Technology Lahore, Pakistan.*

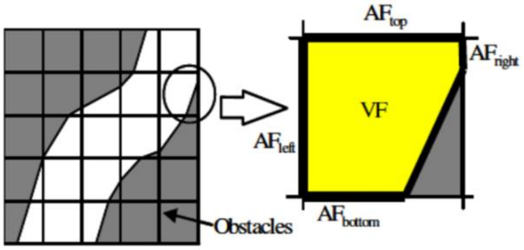
E-mail: gnabi60@hotmail.com

ABSTRACT: *An orifice spillway is normally gated and is used when substantial discharge capacity is needed at low reservoir levels. Orifice spillways are designed for dual purpose of flood disposal as well as flushing of sediments. Flow passing through the spillway shows the complex turbulent behaviour. To model the effect of turbulence, Reynolds's-averaged Navier-Stokes equations are commonly used which is an expanded form of Navier-Stokes equation. Flow physics becomes more complex in case of orifice spillways due to short lengths of spillway, large variation in reservoir levels, high flow depths and wide range of Froude numbers varying from 3 to 9. In view of this background, the present study intends to numerically investigate the hydraulic behaviour of orifice spillway. The objectives of this study include the numerical modeling of complex flows over the orifice spillways, pseudo validation of numerical model results and to assess the flow parameters at different operating conditions. Results showed that model is capable of simulating the orifice spillway flows. Model can measure flow parameters at different operating conditions with an acceptable error of 0.23 to 1.5 %.*

Key Words: Orifice Spillway, Flow Physics, Froude Number, FLOW-3D, Flow Parameters.

1. INTRODUCTION

The large capacity outlets placed below the dam crest and controlled by gates are called orifice or submerged spillway. These outlets are openings in the dam used to drawdown the reservoir level and can also serve to pass density currents. Large bottom openings serve as submerged spillways and their capacity can be used for flushing sediment from the reservoir and disposal of flood [1]. The flow is free surface flow for low reservoir levels and orifice flow for reservoir levels above the orifice opening. The crest profile of most of the orifice spillway is flat ($\alpha < 30°$). Breast walls are provided to hold 60 to 90 m of water heads behind them. Because of narrow river gorges, limited available waterways, and 6 to 8 m thick piers supporting huge breast walls, the discharge intensities range between 100 and 300 m^2/sec. The depths of flow are 10 to 12 m leading to low Froude numbers between 3 and 6 [2].Hydraulic performance of spillways can be checked by physical modelling and numerical modelling. Traditionally, reduced scale physical spillway models are used to study spillway hydraulic performance. But, it has some disadvantages such as scaling effects, time consuming, expensive, requirement of skilled labour and cannot readily capture behavious like cavitation and air entrainment effects, which occur in reality [3]. In recent years, numerical modelling is extensively being used to investigate hydraulic performance of spillways. Computational fluid dynamics (CFD) is a numerical method used to solve fluid flow problems. The computational fluid dynamics (CFD) analysis can solve the Navier-Stokes equations in three dimensions and free surface computation in a significantly improved manner. Computational fluid dynamics presents a cost effective solution that can be employed throughout the entire design process. The use of CFD modelling for spillway application is quiet recent [4]. In last decade, many physical model studies have been validated using CFD models. Most of the numerical studies used the CFD code Flow-3D to analyze the spillway flows. Savage and Johson [5] compared numerically generated discharge rating curves with physical model data and United States Bureau of Reclamation (USBR) calculations. The study found that Flow-3D slightly overestimated the discharges. Gessler [6] model the spillway flows and found 5% difference between CFD and physical model results. Savage and Johnson [5] did not confirm application of CFD model for all spillway configurations. Ho and Riddete [3] applied CFD model to evaluate the hydraulic performance of different spillways for increased flood discharges and suggested future work for cavitation, air entrainment, scour modelling, air demand and dynamic interaction. Bhosekar *et al.* [2] studied the performance of the aerator of orifice spillway by varying the discharge, gate openings and cavity sub-pressures and presented the results with respect to jet length, cavity pressure and air entrainment coefficients in the form of non-dimensional plots and developed equations for jet length and air entrainment coefficient for the orifice spillway aerator. CFD code, Flow-3D was selected for this research study due to its ability to model the free surface flow by using true volume of fluid (true-VOF) method developed by Hirt and Nichols [7] and track the sharp interface between water and air. This code also models the complex geometric region by using fractional area/ volume obstacle representation (FAVOR) technique [8]. This code overlay the mesh on imported non-flow geometry while FAVOR technique is used to determine the void or flow region within each cell as shown in (figure 1). With finer grid spacing, high resolution of the non-flow region is achieved. The use of multi block grids enable larger domains to be modelled and use of nested mesh technique enable more flow details to be captured in regions of interest [3]. Flow physics becomes more complex in case of orifice spillway due to short lengths of spillway, large variation in reservoir levels, high flow depths and wide range of Froude numbers varying from 3 to 9. In view of this background, present study intends to numerically investigate the hydraulic behaviour of orifice spillway. The objectives of this study include the numerical modeling of complex flows over the orifice spillways, pseudo validation of numerical model results and to assess the flow parameters at different operating conditions.



VF = Open Volume/Volume of Cell AF = Open Area/ Cell Edge Area

Figure 1: Conceptual diagram of FAVOR method [5]

1.1. Mangla Dam Spillway

To study the complex flows behaviour of orifice spillways, CFD modelling of Mangla dam main spillway was carried out by operating the model at different reservoir levels and gate openings. The main spillway of dam consists of two-stage stilling basin and sloping side walls. The head works of the main spillway are 444 feet long. It consists of three monoliths separated by 24 feet wide piers. The head works is followed by parabolic chute and intermediate weir divides the chute into two and creates a stilling basin and water pool at an elevation of 1000 feet [9]. The spillway plan and the longitudinal section is shown in (Figure 2).

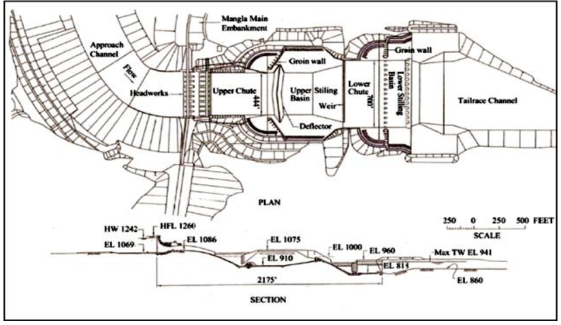


Figure 2: Plan and Longitudinal section of Mangla Dam Spillway

2 Material and Method

2.1 Data Collection

Data collection for model set up sensitivity analysis and model validation includes detail engineering drawings, physical model study results and discharge rating curve. Figure 3 shows single bay three dimensional model of Mangla dam spillway up to intermediate weir. This model was imported into CFD model.

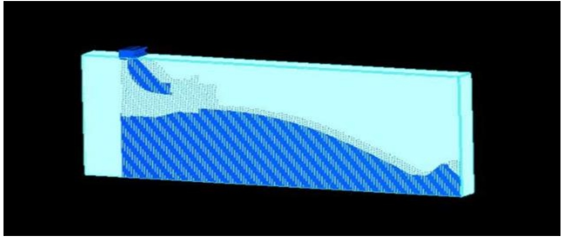


Figure 3: One Bay of Spillway 3D model

2.2 Setting Up of CFD Model

To model Mangla dam spillway, collected drawings were first converted into three dimensional drawings then imported in stereo lithographic (Stl) file extension image. Total four Stl. file images were prepare for different gate openings. These Stl. file images were then imported into CFD code (Flow-3D) for numerical set-up. In next step, meshing of imported Stl. file images was carried out. The extent of mesh domain on upstream and downstream of structure is defined in such a way that it could show the fluid movement and its impact properly. The boundary conditions applied for this model include, volume flow rate, specified pressure, symmetry, wall and out flow. After specifying boundaries of the model, fluids were added on the upstream and downstream sides of the structure as initial condition. After pre-processing explicit and fluid flow solver option was selected to solve Reynold Average Navier Stokes equation (RANS) which is prime equation used by Flow-3D for simulation.

Table 1: Different Scenario of Boundary Conditions

Scenario 1 X min. Specified Pressure X max. Outflow Y min. Wall Y max. Wall Z min. Wall Z max. Specified Pressure	**Scenario 2** X min. Volume Flow Rate X max. Outflow Y min. Wall Y max. Wall Z min. Wall Z max. Specified Pressure
Scenario 3 X min. Volume Flow Rate X max. Specified Pressure Y min. Symmetry Y max. Symmetry Z min. Symmetry Z max. Specified Pressure	**Scenario 4** X min. Specified Pressure X max. Specified Pressure Y min. Wall Y max. Wall Z min. Wall Z max. Specified Pressure

2.3 Sensitivity Analysis

CFD code Flow-3D does not require calibration [10] but it is sensitive to the turbulent models and boundary conditions. Commonly used turbulence models include large eddy simulation, RNG, K-ꜫ, RSM and DSM. Code user manual declares that RNG model is robust and most accurate model [11]. So, it was selected for further simulations. Four scenarios of boundary conditions were used for sensitivity analysis as shown in Table 1.

2.4 Model Validation

Validation of numerical model is extremely important. Validation process indicates the degree of accuracy of the model. A true validation of model involves the comparing of the model results with those from actual structure [5]. But for this study model validation was carried out by using physical model study result due to the lack of actual performance measurements at prototype.

2.5 Gated Flow Modelling

For sensitivity analysis gated flow simulations were performed at vertical gate openings of 3.05 m, 6.10 m, 9.14 m and 10.67 m to find out water levels by maintaining reservoir level at 378.54 m and 384 m. Then flow rate was measured for model validation at same operating conditions. Further vertical gated flow simulations were performed to find out the water surface profiles, pressures and velocity at gate opening of 6.10 m and 9.14 m and at reservoir level of 380.07m, 381.59 m and 383.11 m respectively. Initially simulation was performed with 3 m × 3 m uniform mesh but in order to get more refined results size of mesh was reduced to 1 m. Figure 4 shows gated flow modelling.

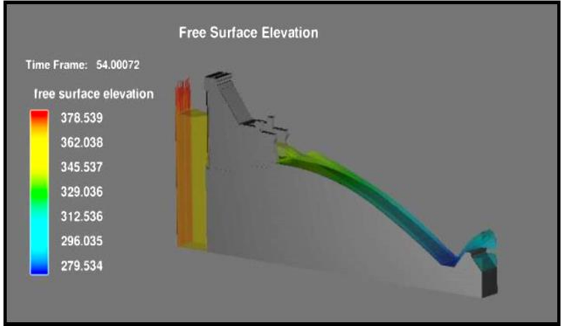


Figure 4: Flow Simulation at 10.67 m Gate Opening

3 RESULTS AND DISCUSSION

3.1 Sensitivity Analysis

Model was run at different operating conditions by applying all set of boundary condition for the sensitivity analysis of the model. The results of sensitivity analysis are discussed below.

3.1.1 Sensitivity Analysis of Model for Boundary Conditions

Model was operated at reservoir level of 378.54 m and 384 m with different gate openings to compare the water levels with physically observed one for all scenarios of boundary conditions. Comparison of results shown in tables 2, 3, 4 and 5 indicate that water levels computed with boundary condition scenario 3 are more close to the physical model results as compared to other scenarios. Hence boundary conditions mentioned under scenario 3 were selected for further simulation.

3.2 Model Validation

After sensitivity analysis, model validation was carried out for gated and free flow condition which is discussed as under.

3.3.1 Flow Rate

Discharge passing through spillway was calculated by operating the model at different operating conditions for validation of model. A comparison between CFD model discharge and physically observed one at various gate openings and reservoir level is shown in Table 6. Comparison shows that percentage difference is in the range of 0.43% to 0.63% which is quite acceptable. Similarly, percentage error at free flow condition is within 0.40 % as shown in Table 7 Which indicates that model has successfully validated for Mangla dam spillway.

3.3 Water Surface Profiles

On successful validation of model, water surface profile at different vertical gate openings were computed at reservoir level of 380.07 m, 381. 59 m and 383.11 m as shown in (figures 5, 6 and 7). Flow surface profiles are important to ensure that flood water is not interfering with other structures such as bridges at crest or raised gates or overtopping the chute walls. The fluctuation in water surface is negligible along the chute but show slight fluctuation due to variation of discharge beyond 150 m at all operating conditions. Further, continuous drop in water surface level is noted in all cases. Fluctuations in water surface levels are negligible and it is not interfering the bridge at crest or raised gates or overtopping the chute walls throughout the length of spillway chute.

3.4 Pressure Distribution

The computations of pressure along spillway chute are used to examine the potential for cavitation damage due to excessive sub atmospheric pressure. Pressure distribution for gated operation of the model at reservoir level of 380.07 m, 381.59 m and 383.11 m are shown in figures 8, 9 and figure 10 respectively. Significant, pressure variation is observed due to variation in flow depth and velocity between 50 m to 125 m length of the chute from dam axis at large gate opening and at all reservoir levels. Further, pressure remained high up to 175 m distance at large gate opening (9.14 m) in all cases. Beyond 175 m, little higher pressure values than large gate opening are noted at 6.10 m against all reservoir levels. At all operating conditions, gradual increasing trend in pressure value is observed due to high flow depth from 175 m to the end of the chute. Generally, pressure remained positive throughout the length of spillway chute which shows that Mangla dam spillway chute is safe against the cavitation damages.

3.5 Velocity

Velocity indicates the potential for erosion damage downstream of the spillway. The erosion assessment depends on the accurate prediction of flow velocity. CFD modelling can reliably predict the velocities provided the model is properly prepared. Velocity profile for gated operation of the model at different operating condition is shown in figures 11, 12 and 13 respectively. Figures shows that velocity values are

Table 2: Numerical and Physical Model Results for Scenario 1 of Boundary Condition

Sr. NO	Reservoir Level (m)	Avg. Physically Observed Water Level (m)				Avg. Numerically Calculated Water Level (m)				Avg. % Difference
		G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	
1	378.54	324.15	324.59	325.37	327.13	321.97	322.49	323.25	324.77	0.67
2	384	-	-	-	327.17	-	-	-	324.46	0.83

Table 3: Numerical and Physical Model Results for Scenario 2 of Boundary Condition

Sr. NO	Reservoir Level (m)	Avg. Physically Observed Water Level (m)				Avg. Numerically Calculated Water Level (m)				Avg. % Difference
		G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	
1	378.54	324.30	324.59	325.37	327.13	320.42	321.24	322.90	324.10	0.98
2	384	-	-	-	327.17	-	-	-	324.26	0.89

Table 4: Numerical and Physical Model Results for Scenario 3 of Boundary Condition

Sr. NO	Reservoir Level (m)	Avg. Physically Observed Water Level (m)				Avg. Numerically Calculated Water Level (m)				Avg. % Difference
		G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	
1	378.54	324.15	324.59	325.37	327.13	322.57	322.93	323.67	324.74	0.56
2	384	-	-	-	327.17	-	-	-	325.36	0.55

Table 5: Numerical and Physical Model Results for Scenario 4 of Boundary Condition

Sr. NO	Reservoir Level (m)	Avg. Physically Observed Water Level (m)				Avg. Numerically Calculated Water Level (m)				Avg. % Difference
		G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 10.67 m	
1	378.54	324.30	324.59	325.37	327.13	319.03	320.93	322.75	323.91	1.13
2	384	-	-	-	327.17	-	-	-	324.45	0.83

Table 6: Comparison of results between CFD model and Physical Model (Gated Flow)

Sr. NO	Reservoir Level (m)	Physical Model Discharge (Cumec)			CFD Model Discharge (Cumec)			Avg.% Difference
		G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	G/O 3.05 m	G/O 6.10 m	G/O 9.14 m	
1	378.54	523.45	1044.09	1657.88	523.88	1058.58	1628.96	0.43 %
2	384	555.92	1117.22	1790.38	556.49	1117.93	1810.91	0.63%

Table 7: Comparison of results between Physical and CFD Model (Free Flow)

Sr. No	Reservoir Level (m)	Discharge (Cumec)		Avg. % Difference
		Observed	CFD	
1	378.54	2077.91	2069.81	0.39
2	384	2222.43	2227.48	0.23

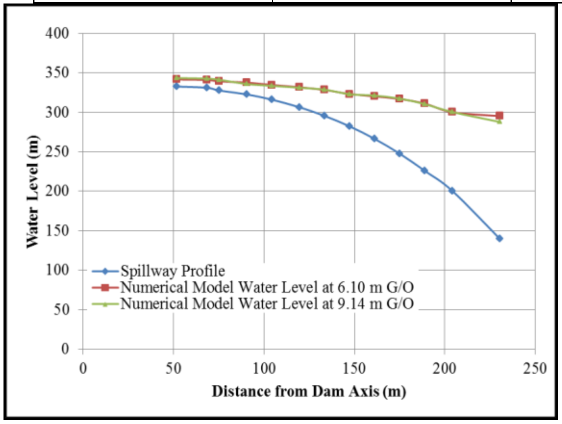


Figure 5: Water Level for Gated Flow at 380.07 m of Reservoir Level

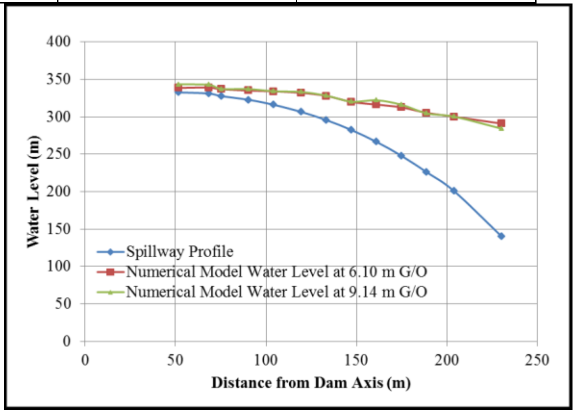


Figure 6: Water Level for Gated Flow at 381.59 m of Reservoir Level

increasing with distance but near the entry point of stilling basin, velocity values drops abruptly due to increase of water depth. Near the crest of spillway, flow velocity is in the range of 10 to 15 m/sec but when water moves further downward it attains maximum value of 30 m/sec on all operating conditions. So, velocity varies from 10 m/sec to 30 m/sec along the chute of Mangla dam spillway. Water depth increases due to the presence of small height weir at end of spillway chute which reduces the velocity 10 m/sec. The existence of two stage stilling basin at spillway site will further reduced this velocity value which will ultimately reduce the potential for the downstream bed erosion.

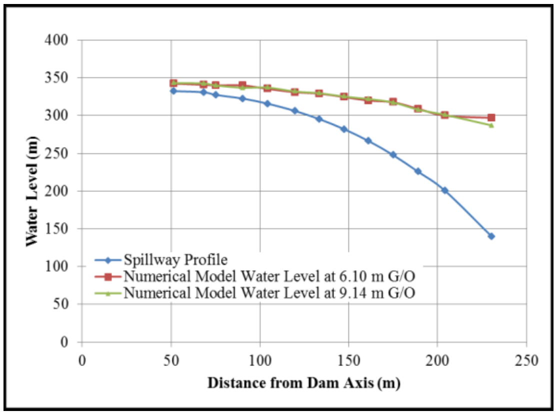


Figure 7: Water Level for Gated Flow at 383.11 m of Reservoir Level

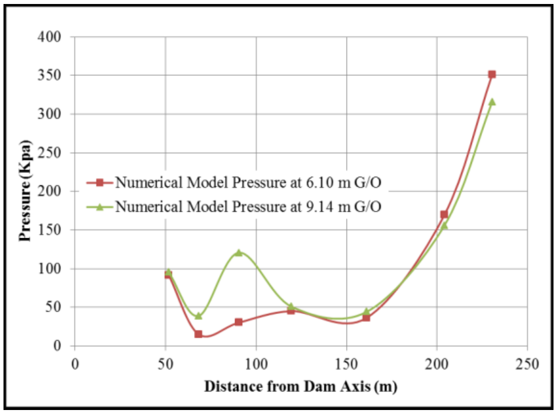


Figure 8: Pressure Distribution at Reservoir Level of 380.07 m

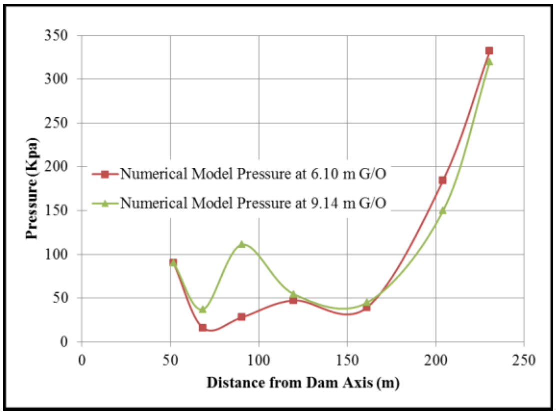


Figure 9: Pressure Distribution at Reservoir Level of 381.59 m

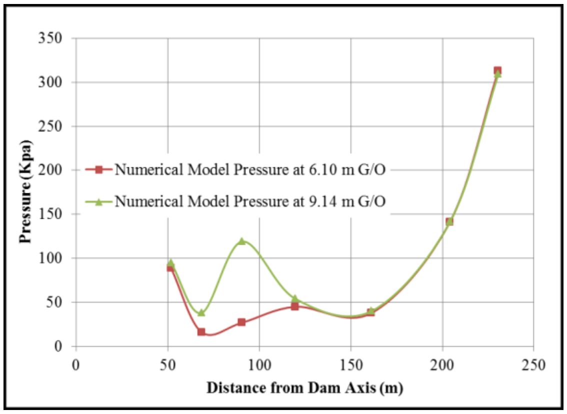


Figure 10: Pressure Distribution at Reservoir Level of 383.11 m

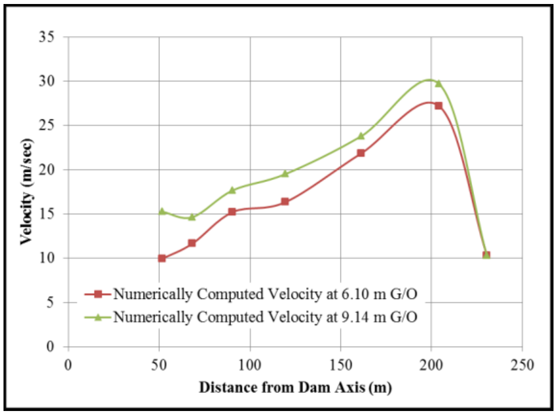


Figure 11: Velocity Profile at 380.07 m of Reservoir Level

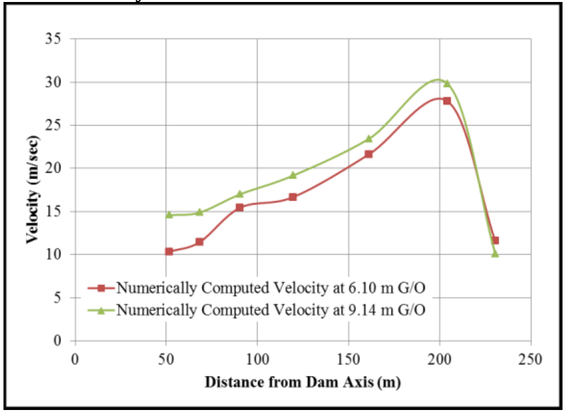


Figure 12: Velocity Profile at 381.59 m of Reservoir Level

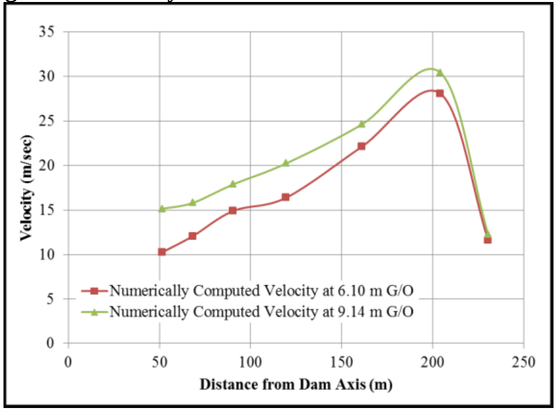


Figure 13: Velocity Profile at 383.11 m of Reservoir Level

4 CONCLUSION AND RECOMMENDATIONS

Computational Fluid Dynamics (CFD) model of Mangla dam spillway was operated by varying the reservoir levels and gate openings. Model computed the pressures at different gate openings and reservoir levels. The computed pressures were found in range of 10 KPa to 350 KPa at all opening conditions. Pressures over the chute remain positive which indicates no danger of cavitation. Variation in velocity over the profile was noted between 10 m/sec to 30 m/sec which reduce to 10 m/sec near the entry point of stilling basin. Flow depth computed near gate slot was 6.02 m which raise to 48.81 m at the end of chute. The maximum mean level of water surface computed at a distance of 51.72 m from dam axis was 343.24 m which is well below top level of the chute wall 376.10 m indicating the no danger of overtopping.

Model computed the discharge both for vertical gated and free flow (vertical full gate opening) conditions. For gated flow conditions, maximum flow passed through single bay of spillway was 1810.91 m^3/sec while at full vertical gate opening, it was 2227.48 m^3/sec which confirms the design discharge capacity 3173.04 m^3/sec at maximum flood level of 384 m. Computed flow parameters with low percentage error difference in comparison with observed one confirms that model is able to assess the flow behaviour successfully. Simulations of flows with nested mesh technique, other turbulence model, and numerical options are required especially for the smaller gate openings. Moreover, air entrainment effect on flow parameters is also required to be model for refinement of results. Modelling of flows in stilling basin of spillway is recommended for analysis of flow behaviour and to check the potential for erosion damage.

5 ACKNOWLEDGEMENTS

The author (Zohaib Nisar) would extremely like to thank Engr. Kaleem Sarwar (Assistant Professor) for their guidance, help in this research and writing this research paper. Author also would like to thank Dr. Ghulam Nabi (Assistant Professor) and Center of Excellence in Water Resources Engineering, University of Engineering & Technology, Lahore, Pakistan for doing research and completion of my specialization in Water Resources Engineering.

6 REFERENCES

[1] Novak, P, A.I.B, Moffat, C. Nalluri and R. Narayanan, Hydraulic Structures, Taylor & Francis Co. 4th Edition, ISBN 0-203-96463-2, 2007.

[2] Bhosekar, V.V, V. Jothiprakash and P.B, Deolalikar, Orifice Spillway Aerator: Hydraulic Design, Journal of Hydraulic Engineering, Vol. 138, No. 6, pp: 563-572, 2012.

[3] Ho, D. K.H. and K.M, Riddette, Application of computational fluid dynamics to evaluate hydraulic performance of spillway in Australia, Australia Journal of Civil Engineering, Vol. 6, No.1, pp: 81-104, 2010.

[4] Chanel, P.G and J.C, Doering, Assessment of spillway modelling using computational fluid dynamics, Canadian Journal of Civil Engineering, pp: 1481-1485, 2008.

[5] Savage, B.M. and M.C, Johnson, Flow over ogee spillway: Physical and Numerical Model Case Study, Journal of Hydraulic Engineering, Vol. 127, No. 8, pp: 640-649, 2001.

[6] Gessler, D, CFD Modelling of Spillway performance, Proceedings of world Water and Environmental Resources Congress 2005, American Society of Civil Engineer, Anchorage, Alaska, 2005.

[7] Hirt, C.W and Nichols, B.D, Volume of fluid method for dynamic of free boundaries, Journal of Computational Physics, Vol. 39, pp. 201-225, 1981.

[8] Hirt, C.W. and Sicilian, J.M, A Porosity technique for the definition of obstacles in rectangular cell mesh, Proceedings of 4th international conference of ship dynamic, National Academy of Science, Washington D.C, 1985.

[9] WAPDA Model Study Cell, IRI, Hydraulic Model Studies for Mangla dam Raising Project, Final Technical Report, Hydraulic Research Station, Nandipur, 2004.

[10] USBR, Folsom Dam Joint Federal Project, Existing spillway modeling, Discharge Capacity Studies, California, USA, 2009.

[11] Flow Science, Inc., Flow-3D user manuals, Version 9.2. Flow Science, Inc., Santa Fe, N.M, 2007.

7 APPENDIX – NOTATION

Following symbols used in this paper:

CFD = Computational Fluid Dynamics
RNG = Re-normalized Group Model
RANG = Reynold Average Navier Stokes Equation
Stl. = Stereo lithographic
G/O = Gate Opening
Cumec = Cubic meter per second (m3/sec)