COMPUTATIONAL FLUID DYNAMICS SIMULATION METHODOLOGIES FOR STILLING BASINS

PANKAJ LAWANDE
CFD Engineer, Flow Science Software Private Limited, Pune, Maharashtra, India

ABSTRACT

Stilling basins are common structures designed for the dissipation of energy downstream of spillways. The dissipation of energy in these structures takes place due to turbulence in the hydraulic jump in the stilling basin. Due to the complexity of the flow in the stilling basin, physical hydraulic modelling studies can be necessary to finalize the design of the stilling basin elements such as the basin floor, training and divide walls; and appurtenances like baffle and chute blocks that are subjected to dynamic and complex hydrodynamic forces. Since no two structures are exactly alike, attempts to extrapolate from existing data for related stilling basins can create difficulties.

To fill the need for up-to-date hydraulic design information on stilling basins and energy dissipators, Computational Fluid Dynamics (CFD) simulations can be a powerful supplement for physical modelling studies. In the present study, CFD is used to investigate critical design parameters such as discharge capacity, hydraulic jump characteristics and water surface profiles for a stilling basin.

1. INTRODUCTION

For energy dissipating devices that have been designed in conjunction with spillways, outlet works, and canal structures, it is often necessary to make model studies of individual structures to be certain that these will operate as anticipated. The reason for these repetitive tests is that a factor of uncertainty exists regarding the overall performance characteristics of energy dissipators.

Stilling basins are a common type of structures designed for the dissipation of energy downstream of spillways. The dissipation of energy in these structures takes place due to turbulence in the hydraulic jump in the stilling basin. Physical hydraulic modelling studies can be necessary for finalizing the design of the stilling basin elements such as the basin floor, training and divide walls; and appurtenances like baffle and chute blocks that are subjected to dynamic and complex hydrodynamic forces. Since no two structures are exactly alike, attempts to generalize existing data can result in unreliable and inconsistent results.

To fill the need for up-to-date hydraulic design information on stilling basins and energy dissipators, Computational Fluid Dynamics (CFD) simulations can be a powerful supplement for physical modelling studies.

Use of Computational Fluid Dynamics (CFD) tools help to gain insight in studying the design of stilling basins, including basin geometry and tail-water (Shearin-Feimester et al. 2015), given the intrinsic limitations of experimental measurements. However, lack of validation and verification is still an issue noted by several researchers (Chanson and Lubin, 2010; Chanson, 2013). The VOF method described by Hirt and Nichols (1981) - used extensively for hydraulic engineering problems (Carvalho et al. 2008, Bombardelli et al. 2011, Oertel and Bung 2012) have been employed using FLOW-3D to properly reproduce turbulence and free surface effect upon the mean flow. Model geometry (as shown in Figure 1) and flow boundary conditions are based on the previous experimental study of Hinge et al. (2010).

2. DESIGN OF ENERGY DISSPATORS

The design of energy dissipators plays a crucial role in the design and construction of dams, weirs and barrages. In general, the primary objective in designing the energy dissipators is to reduce the high velocity flow to minimize the erosion of natural riverbed. IS 4997 (1968) gives a classification of different types of energy dissipators. It advises that hydraulic jump type stilling basins and bucket type energy dissipators be used depending upon conditions of downstream tail water. However, in projects where the fall is greater than 15 m, the discharge intensity is more than 30 m3/s/m, or for possible asymmetry of flow, it is recommended that performance of energy dissipation arrangement shall be tested using a physical model. Factors affecting design of energy dissipators include: the nature of the foundation, discharge magnitude (Bowers and Toso 1988), flow velocity (Cassidy 1990), and tailwater rating curve.

3. METHODS

In the present study, we considered two cases by Hinge et al. (2010) with a new design of stilling basin which includes a rectangular broad crested stepped weir geometry that would restrain the formation of clear hydraulic jump within basin for any discharge between design discharge and 20% of the design discharge and the given range of tail water submergence. FLOW-3D is used to validate hydraulic jump characteristics post jump depth (y2) and tail water depth (yt) using CFD simulations.

FLOW-3D solves the transient form of the conservation equations so the two metrics used to determine that the flow had reached a steady state were a) that the volume of fluid within the computational domain had reached a constant value, and b) that the mass averaged mean kinetic energy of the flow had reached a constant value.

For each discharge, the free surface elevation is measured using history probes (FLOW-3D objects that allow for the measurement of flow quantities through a prescribed point) located at upstream and downstream of the weir. The computed values are then compared to measured data from the physical model.

4. PHYSICAL MODEL

The physical model used for validating the CFD model was performed by Hinge et al. (2010). A new design of stilling basin in the form of horizontal apron with a rectangular broad crested stepped weir is proposed. To support this new design, a model study of an existing spillway energy dissipator is carried out in laboratory to check the applicability of the downstream weir. To experimentally verify the performance of mathematically obtained weir geometry, a sectional model of spillway and horizontal stilling basin with rectangular broad crested stepped weir at its end is constructed as shown in Figure 1.

ABSTRACT: Stilling basins are common structures designed for the dissipation of ener
downstream of spillways. The dissipation of energy in these structures takes place due
turbulence in the hydraulic jump in the stilling basin. Due to the complexity of the flow in

Figure 1 : Definition sketch of hydraulic jump in a rectangular channel.

A stepped weir is considered to be made up of number of rectangular weirs. A mathematical procedure based on the free flow rectangular weir formulae was developed (Hinge et al. 2010). The proposed stepped weir should cater to wide range of discharge from design discharge (Qmax) to minimum discharge equal to 20% of the design discharge (Qmin). 'N' intermediate discharges between Q min and Qmax with an interval of (Qmax - Qmin) / (N+1) are considered resulting in (N+2) discharges corresponding to which there would be (N+2) steps in a stepped weir. Figure 2 shows a sketch of a rectangular stepped weir section A-A taken from figure 1.

profiles for a stilling basin.

INTRODUCTION

For energy dissipating devices that have been designed in conjunction with spillways, o
works, and canal structures, it is often necessary to make model studies of individual struct
to be certain that these will operate as anticipated. The reason for these repetitive tests is
a factor of uncertainty exists regarding the overall performance characteristics of en
dissipators.

Stilling basins are a common type of structures designed for the dissipation of en
downstream of spillways. The dissipation of energy in these structures takes place du
turbulence in the hydraulic jump in the stilling basin. Physical hydraulic modelling studies
be necessary for finalizing the design of the stilling basin elements such as the basin fl

Figure 2 . Definition sketch of geometry of rectangular stepped weir

5. BHAMA ASKHED CASE

Hinge et al. (2010) reported studies on Bhama Askhed Dam EDA (India), in which sectional model of spillway and horizontal stilling basin with rectangular broad crested stepped weir at its end is constructed. Experimental verification of the weir performance, a sectional model (scale 1:50) of spillway and an innovative design of stilling basin are constructed. Experiments show that in a stilling basin with horizontal slope, a stepped weir designed for tail water deficiency, restricted the hydraulic jump at the desired location even for discharges lower than the design discharge. The stepped weir performance is judged from the ability of weir to form free hydraulic jumps near toe of spillway from Q_{min} to Q_{max}. The ideal, observed-y1, y2 and yt values for 4-trials are given in Table 1.

Table 1 : Experimental results for the Bhama-Askhed Dam case (Hinge et al. 2010)

Q m^3/s	Ideal y_2 (m)	Experimental	
		y_2 (m)	y_t (m)
0.025	0.111	0.094	0.016
0.050	0.156	0.137	0.026
0.075	0.189	0.169	0.035
0.100	0.214	0.191	0.042

Flood damages that occurred in 2005 describe that 12 out of 35 RCC panels (of size 11m x 7m x 0.3m thick and weighing 55 Tons each and with 5-anchor bars of 25mm diameter) were dislodged from their location and few of them were thrown away outside the basin, as shown in Figure 3.

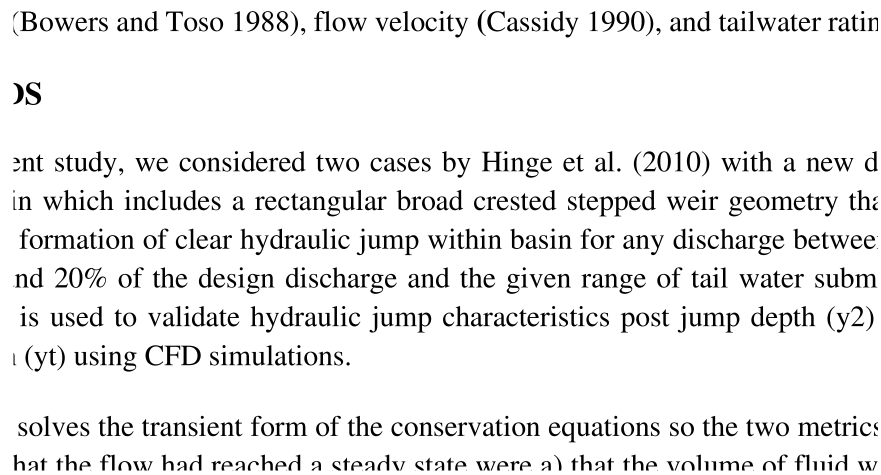

(Bowers and Toso 1988), flow velocity (Cassidy 1990), and tailwater ratin
)S
ent study, we considered two cases by Hinge et al. (2010) with a new d
in which includes a rectangular broad crested stepped weir geometry tha
formation of clear hydraulic jump within basin for any discharge betwee
nd 20% of the design discharge and the given range of tail water subm
is used to validate hydraulic jump characteristics post jump depth (y2)
(yt) using CFD simulations.
solves the transient form of the conservation equations so the two metrics
hat the flow had reached a steady state were a) that the volume of fluid w

Figure 3 : View of dislodged RCC panels in Bhama Askhed dam EDA.

6. PAWANA CASE

Hinge et al. (2010) performed model studies on a single bay of Pawana dam spillway at 1:40 scale. The performance of stepped weir geometry is experimentally verified as the location of front of hydraulic jump is restricted near the toe of spillway for different discharges (ranging from Q_{min} to Q_{max} and Fr1 > 4.5) and specific range of submergence ratio . Parameters including the percentage energy dissipation and residual energy of tail water are compared between the existing broad crested weir and new stepped weir. The experimental y2 and yt depth with the ideal y2 depths given by the Belanger momentum equation (Subramanya, 1986). are entered in Table 2.

Table 2 : Experimental results for the Pawana Dam case (Hinge et al. 2010)

Q m^3/s	Ideal y_2 (m)	Experimental	
		y_2 (m)	y_t (m)
0.0200	0.210	0.195	0.054
0.0181	0.200	0.186	0.052
0.0160	0.188	0.175	0.049
0.0130	0.170	0.158	0.045
0.0095	0.146	0.135	0.040
0.0077	0.132	0.120	0.037
0.0065	0.122	0.110	0.035
0.0043	0.100	0.090	0.030

7. COMPUTATIONAL FLUID DYNAMICS MODELS

FLOW-3D is a commonly used commercial CFD software specializing in the simulation of complex free surface flows using the volume of fluid (VOF) method and solves the fully 3D Reynolds-averaged Navier Stokes Equations (Eq. 1).

gure 1. Definition sketch of hydraulic jump in a recta

...(1)

is considered to be made up of number of rectar

A full description of the modeling capabilities and methods can be found in the FLOW-3D User Manual (Flow Science, 2018). The physical model geometry from both the case studies were recreated in CAD and imported into FLOW-3D at the model scale (Figure 3). This includes accurate representation for the shape of the spillway crest, height, pier, and stepped weir.

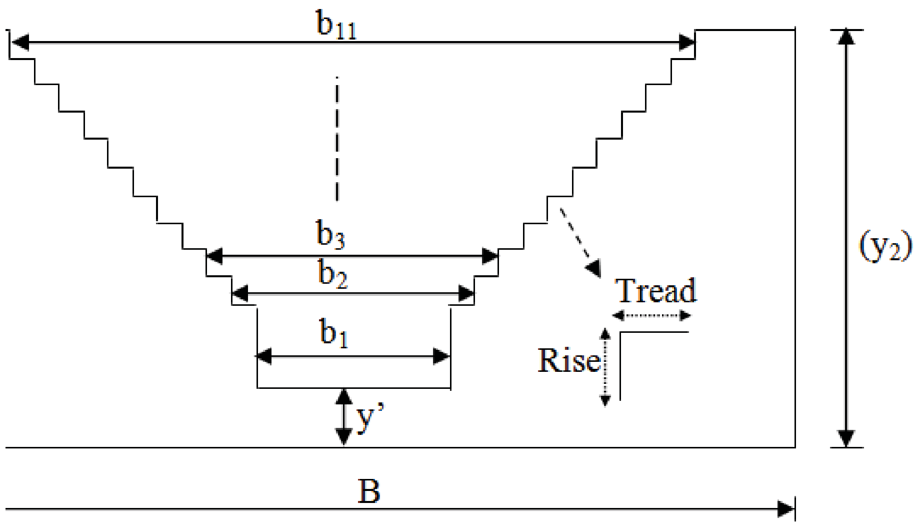


Figure 4 : Geometry of the Bhama-Askhed Dam in CFD model

8. MESHING

CFD requires that the solution region be discretized into small control volumes; this set of control volumes is commonly called the mesh. In FLOW-3D, the mesh is structured and composed of rectangular elements defined by a set of planes perpendicular to each of the coordinate axes (x,y,z). Additionally, FLOW-3D incorporates geometry into the mesh using a technique called FAVOR (Hirt and Sicilian, 1985). Using this method, the geometry is incorporated into the conservation equations via the open volume fraction and open area fractions in each cell (Flow Science, 2018). This approach offers a simple and accurate method to represent complex surfaces without requiring a body-fitted grid, though features smaller than the mesh are not represented. This implication is important if there are thin objects or sharp breaks in the geometry that are important to be resolved. Under refining the mesh may result in poor geometry representation that can have significant negative effects on the solution. In these situations,, the mesh may be locally refined to better resolve these objects. Alternatively, locations of fixed grid lines can be defined with mesh planes to resolve specific geometric features.

Consideration should also be given to how mesh design (e.g., cells aspect ratios and adjacent cell size ratios) and resolution effect the hydraulic solution since accurately capturing the spatial variation of the solution is critical to the quality of the simulation results. A poorly designed mesh can lead to poor pressure solver convergence performance, high run-times, and possible inaccuracies in the solution.

In the models of the Bhama Askhed and Pawana dams, mesh planes were fixed at the sharp edges of the stepped weir to ensure the open area fractions and volume fractions are represented accurately in the FLOW-3D simulations.

To further reduce the computational expense, we assumed that the flow solution was symmetric across the center of the physical domain. The symmetry assumption allows for a reduction of the domain size by a factor of 2. To test the validity of this assumption, initial testing was completed to compare full width versus half width results. Discharges differed by less than 2%, indicating that symmetry is a valid assumption for this case. All subsequent simulations were conducted on the reduced symmetric domain shown in Figure 5.

Figure 3. View of dislodged RCC panels in Bhama Askhed dam EDA.

WANA CASE

Figure 5 : Use of symmetry in the FLOW-3D CFD model.

8. BOUNDARY CONDITIONS

The inflow boundary condition was defined as a specified inlet discharge. The water surface elevation at the upstream boundary was set using the natural inlet option in FLOW-3D, which automatically sets water depth at the boundary to be equal to that in the adjacent mesh cell inside the domain. With this treatment, the water surface elevation near the inlet boundary rises as water surface rises naturally inside the domain due to the downstream hydraulic control. This condition is very helpful for sub-critical flow conditions at the inlet boundary when the fluid elevation is not known. All the results in this article are obtained using this natural inlet setup option.

The downstream boundary is located downstream of the stepped weir where flow is in a subcritical regime. For a subcritical outflow we defined an outflow boundary equal to gauge pressure to allow flow to exit the domain. A symmetry boundary was defined at the center of the physical domain as previously discussed. All solid objects in the domain were assumed to be hydraulically smooth no slip surfaces.

10. TURBULENCE MODEL

The Renormalization group k-epsilon (RNG) was selected as the turbulence model for all simulations presented in this study since it is one of the most robust, versatile and computationally efficient turbulence models in FLOW-3D. The user can manually control the maximum turbulence mixing length scale, but the results were not found to be sensitive to this and were instead obtained using a dynamically computed length scale computed by the solver.

11. RESULTS

Figure 5 provides a graphical illustration of the solution in the case where the half width of the weir was modelled.

includes accurate representation for the shape of the spillway crest, height, pier, and ste
weir.

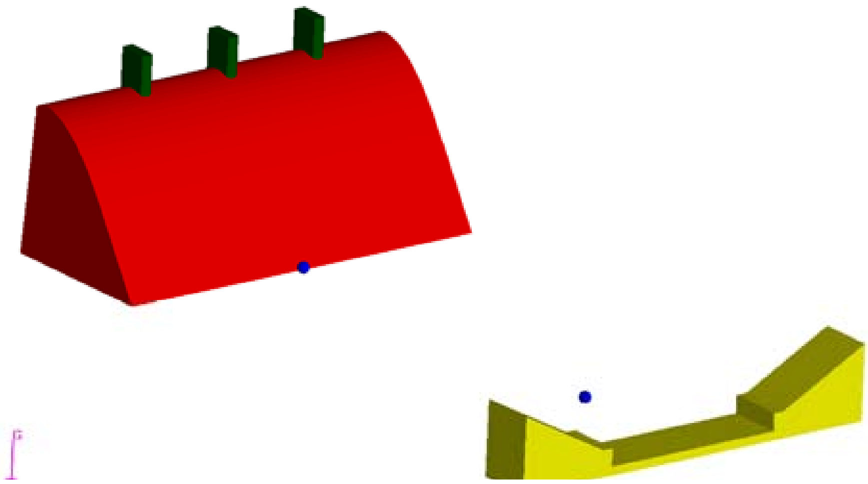
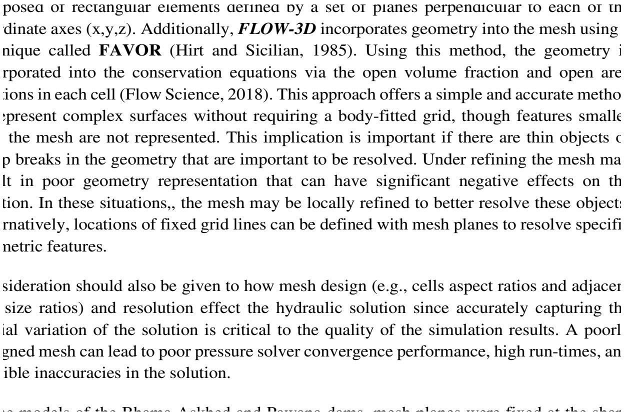


Figure 6 : Iso surface rendering of flow over a stepped weir for Bhama-Askhed Dam case.

Figures 7 & 8 shows comparisons of experimental results (blue line) and FLOW-3D simulation results (red markers) for the Bhama-Askhed and Pawana case respectively. The results show that FLOW-3D predictions of free surface elevation for a given discharge are generally within 5% of the experimental data for all cases that were studied.


posed of rectangular elements defined by a set of planes perpendicular to each of th
dinate axes (x,y,z). Additionally, ***FLOW-3D*** incorporates geometry into the mesh using
nique called **FAVOR** (Hirt and Sicilian, 1985). Using this method, the geometry i
rporated into the conservation equations via the open volume fraction and open are
ions in each cell (Flow Science, 2018). This approach offers a simple and accurate metho
present complex surfaces without requiring a body-fitted grid, though features smalle
the mesh are not represented. This implication is important if there are thin objects c
p breaks in the geometry that are important to be resolved. Under refining the mesh ma
lt in poor geometry representation that can have significant negative effects on th
tion. In these situations,, the mesh may be locally refined to better resolve these object
rnatively, locations of fixed grid lines can be defined with mesh planes to resolve specifi
netric features.

sideration should also be given to how mesh design (e.g., cells aspect ratios and adjacer
size ratios) and resolution effect the hydraulic solution since accurately capturing th
ial variation of the solution is critical to the quality of the simulation results. A poorl
gned mesh can lead to poor pressure solver convergence performance, high run-times, an
ible inaccuracies in the solution.

Figure 7 : Hydraulic jump characteristics for Bhama-Askhed Dam experimental and FLOW-3D data

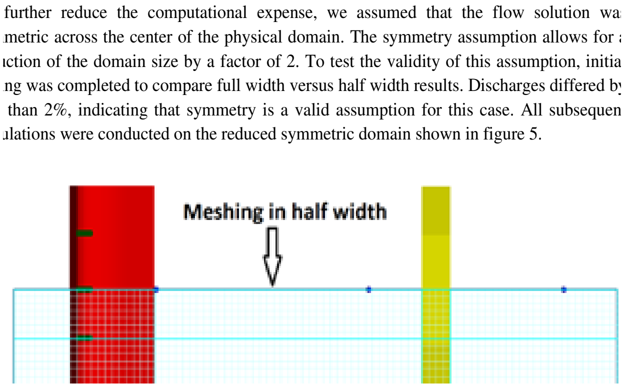


Figure 8 : Hydraulic jump characteristics for Pawana Dam experimental and FLOW-3D data

12. DISCUSSION

FLOW-3D provides excellent predictions of the post jump depth and tail water depth. In general, estimates were within a 5 percent of measured data for all simulated conditions. These results are consistent with previous studies evaluating FLOW-3D for other types of hydraulic controls (Isfahani, 2018; Wendelbo and Fox, 2017). We found the two primary factors affecting accuracy are, unsurprisingly, the proper specification of geometry and mesh resolution.

Accurate CAD representation of the geometric features is mandatory for any type of numerical simulation. Specific challenges for accurate geometry representation in this study are ensuring the stepped weir is accurately resolved with the FAVOR method in FLOW-3D. As mentioned earlier, this was addressed by adding fixed grid lines at the sharp edges to ensure it was resolved by the mesh.

In addition to accurate definition of the geometry, the mesh resolution was shown to be an extremely important user input. The results of the mesh sensitivity test show that the solution converges to within a 5 percent of measured data when the mesh is resolved to a level of 15-20 cells above the weir height. Extremely accurate predictions (<1% error) can be obtained by further increasing the mesh resolution to 40 cells above the weir height. However, care must be taken to avoid using an unnecessarily fine mesh resolutions in locations away from the area of interest to avoid intractable simulation times (Lawande, P. 2019). Additionally, this study benefited from the use of a symmetry assumption to reduce the domain size. In practical cases, it is likely that non-uniform approach conditions may be present and, in these cases, the symmetry assumption may not be valid and can potentially result in adverse effects on accuracy of the simulation. However, it is simple to relax this constraint in the CFD model and it should result in a relatively modest 2x increase in the run time.

13. CONCLUSIONS

In this study we evaluated how accurately FLOW-3D could predict the post-hydraulic jump depth, and tail water depth by comparing the results from numerical simulations against measured data obtained from a physical model. The results show that FLOW-3D predicts these hydraulic jump characteristics within a few percent of measured data. Critical elements of the setup include proper definition of structure geometry and mesh resolution. Further studies are planned to evaluate the ability of FLOW-3D to simulate the pressures fluctuations in a stilling basin and aeration, recirculation near the stepped weir.

REFERENCES

Achour, B., Debabeche, M. (2003). “Control of hydraulic jump by sill in triangular channel”. J. Hydr. Res., 41(3), 319-325.

Bombardelli, F.A., Meireles, I., and Matos, J. (2011). “Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways”. Environmental Fluid Mechanics, 11(3), 263-288.

Bowers, C.E., Toso, J. (1988). “Karnafuli project, model studies of spillway damage”. J.Hydr.Engg., ASCE, 114(5), 469-483.

Cassidy,J.J. (1990). “Fluid mechanics and design of hydraulic structures”. J.Hydr.Engg., ASCE, 116(8), 961-977.

Carvalho, R.F., Lemos, C.M., and Ramos, C.M. (2008). “Numerical computation of the flow in hydraulic jump stilling basins”. Journal of Hydraulic Research, 46(6), 739-752.

Chanson, H. (2013). “Hydraulics of aerated flows: qui pro quo?”. Journal of Hydraulic Research, 51(3), 223-243.

Chanson, H. and Lubin, P. (2010). Discussion on "Verification and validation of a Computational Fluid Dynamics (CFD) model for air entrainment at spillway aerators". Can. J. Civil Eng., 37(1), 135-138. DOI:10.1139/L09-133

Chow, V.T. (1959). "Open channel hydraulics". McGraw-Hill book company, Inc., New York, 1959.

Flow Science Inc., 2018. FLOW-3D v12.0 user's manual.

Hinge G. A., Balkrishna S., Khare K.C. (2010), Improved Design of Stilling Basin for Deficient Tail Water, Journal of Basic and Applied Scientific Research, Vol.1(1), pp. 31-40

Hinge G. A., Balkrishna S., Khare K.C. (2010), Pawana Dam Energy Dissipation – A Case Study, Australian Journal of Basin and Applied Sciences, Vol.4(8), pp. 3261-3267.

Hirt, C.W., and Nichols, B.D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of computational physics, 39(1), 201-225.

Hirt, C.W. and Sicilian, J.M., 1985, September. A porosity technique for the definition of obstacles in rectangular cell meshes. In International Conference on Numerical Ship Hydrodynamics, 4th.

Isfahani, A. (2018). CFD modeling of piano key weirs using FLOW-3D, International Dam Safety Conference, January 23-24, 2018, Thiruvananthapuram, Kerala, India.

Lawande, P. (2019). Three Dimensional CFD Modeling of Flow Over a Tainter Gate Spillway, International Dam Safety Conference 2019, Bhubaneswar India.

Oertel, M. and Bung, D.B. (2012). "Initial stage of two-dimensional dam-break waves: Laboratory versus VOF." Journal of Hydraulic Research, 50(1), 89-97.

Shearin-Feimster, L.E., Crookston, B.M. and Felder, S. (2015). "Design approaches and numerical modeling of a stepped spillway under high tailwater conditions." Proc. of the 35th Annual USSD Conference, Louisville, Kentucky. CD-ROM.

Subramanya, K. (1986). "Flow in open channels". Tat a McGraw-Hill Publishing Co. Ltd., New Delhi, India.

Upadhyay S. (2017). "Restricting Hydraulic Jump Location inside stilling Basin for Maximum Energy Dissipation" International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 04 Issue: 12.

Vittal, N., Al-Garni, A.M. (1992). "Modified type III stilling basin – new method of design". J. Hydr. Res., 30(4), 485-498.

Wendelbo J, Fox B. (2017), CFD modeling of Piano Key weirs: validation and numerical parameter space analysis, 2017 Dam Safety, San Antonio