CFD in Hydraulic Modeling

Vivek S. Chavhan[1]*, Dr. Ganesh A. Hinge[2]

[1]*Research Scholar, TSSM's Bhivarabai Sawant College of Engineering and Research, Pune 411041, India*
[2]*Research Guide, TSSM's Bhivarabai Sawant College of Engineering and Research, Pune 411041, India*
vschavhan@aissmscoe.com

Abstract

Ski-jump spillways are the most important structures using flow velocities greater than 20 m /s. With the development of computer science and a variety of computational fluid dynamics (CFD) software, the behavior of ski-jump spillways can be studied in a short period of time and without paying high expenses. This paper presents comparisons of experimental and CFD results for pressure on the surface of the ski-jump bucket. Pressure on the curve surface of the bucket is one of the most important design parameters. In these structures, pressure fluctuations can result in considerable lift force under the structure. The obtained results showed that for different flow rates, the maximum pressure is near the center of the bucket's curve. For different flow rates, the theoretical and numerical results showed a good agreement.

Keywords: *CFD, Ski-jump spillway, Pressure, Flow-3D.*

1. Introduction

Energy dissipating structures are constructed at the end of the spillway to dissipate the excess energy. In general, hydraulic jump stilling basins, roller buckets, ski jump buckets or other energy dissipating structures are applied to dissipate excess energy. If the selected option for energy dissipation is ski-jump, a curved spherical surface called a bucket will be installed at the end of the spillway chute, causing a mild transition of flow from the spillway surface to the outlet channel. Ski-jump buckets are not considered in themselves as an energy dissipator, however, it is an integral part of the energy dissipation system. Ski-jump buckets are intended to guide high velocity flows (jets) to a distant location rather than to the dam, powerhouse, spillway, and other parts of the dam structure. A small amount of water energy is transmitted due to friction through the bucket. In the hydraulic design of a ski-jump bucket, the parameters that are of high importance to designers include the geometry of the bucket, the pressure exerted on its boundaries, and the jet trajectory characteristics. Pressure fluctuations in ski-jump buckets were studied using analytical and numerical methods by Khani et al.[1]. Savage and Johnson compared the flow-3D simulation results through physical models and USBR and USAC data to a standard Ogee crested spillway [2]. Computational fluid dynamics (CFD) has been increasingly applied by engineers for modeling and analysis of complex problems related to hydraulic design [3]. Ho et al. [4] investigated the behavior of a spillway with a standard WES profile under maximum flooding. Their predicted values for pressure heads on the surface and flow rates of the spillway were in close agreement with experimental achievements. Two dimensional flow analyzes at the bottom of the dam were performed by Heidirinjad and Najibi [5]. Ecklund [6] investigated the jet trajectory of a ski-jump spillway using physical and CFD models.

2. Experimental study

In an experimental study a model of ski-jump bucket was designed as per the guidelines given in IS code. Experiments were conducted in the fluid mechanics laboratory, in a hydraulic flume for various approach flow condition. Experimental model is placed at the center of the flume. The

ISSN: 2233-7857 IJFGCN

flow depths were measured on the bucket centerline with a pointer gauge. Hydraulic forces acting on a ski-jump bucket are of interest in the structural design of the bucket. Theoretical studies, model studies and prototype data indicate that bottom pressures on the ski-jump bucket fluctuate continuously. Therefore, in this study the pressure values were estimated theoretically along the curve of the ski-jump bucket. The total pressure head on the bucket is the summation of the pressure head, due to centrifugal force and due to water depth in the bucket. The total theoretical pressure head is given by equation 1.

$$P = (\frac{V^2}{g\,r} + 1)\,?\,d_1 \qquad (1)$$

Where

P = Bucket pressure
ϒ = Unit weight of water
V = actual velocity of flow
g = acceleration due to gravity;
R = radius of bucket
d_1 = depth of the flow entering bucket.

The pressure fluctuations on bucket may cause a floor destruction and cavitation. For this purpose, the theoretical pressure was calculated, at the entry of bucket, at the mid of bucket, and at lip of the bucket for flow condition 1, 4 and 10. The theoretical pressures were calculated according to the depth and velocity of the flow on bucket by using equation (1). The results of these calculations were given in Table 1. For conventional ski-jump bucket, the maximum pressure occurs at the midpoint of the bucket.

Table 1. Experimental bucket Pressure

Type	Discharge Q (m3/s)	Pressure P (Pa)		
		At lip of bucket (1)	At center of bucket (2)	At entrance of bucket (3)
Q_1	0.00962	792.45	792.45	117.29
Q_2	0.005	404.61	404.61	77.48
Q_3	0.00255	208.99	208.99	21.47

3. CFD study

The Flow-3D model developed by Flow Science is a computational fluid dynamics (CFD) tool capable of simulating the dynamic and steady state behavior of liquids and gases in one, two or three dimensions. It applies to almost any type of flow process and is capable of simulating free surface flow, computational fluid dynamics (CFD) is a method of simulating a flow process in which standard flow equations such as Navier-Stokes and continuity. The equations are projected and solved for each computational cell. Using CFD software is similar to setting up an experiment. If the experiment is not set up correctly to simulate a real-life situation, the results will not reflect the real-life situation. In the same way, if the numerical model does not accurately represent real-life situations, the results will not reflect real-life conditions. Simulation by using Flow- 3D consists of below important steps.

ISSN: 2233-7857 IJFGCN

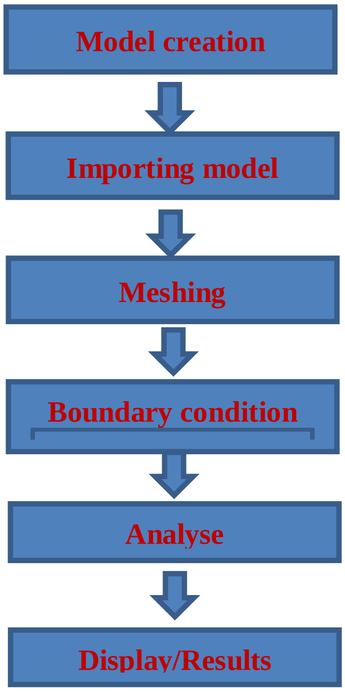

Figure. 1. Flow chart for CFD simulation

A Flow-3D takes STL files that are universally exported by all CAD software available in the market. For present study 2-D model of the spillway is created in AutoCAD. After 2-D modeling is done, a 3-D model is created and then 3-D model file is converted into stl file for importing in Flow-3D. Fllow-3D uses a structured and orthogonal mesh with rectangular (2D) and hexahedral cells (3D). To have an accurate result, an appropriate condition should be selected for boundary based on the nature of the flow. In the present study, Inlet condition is set as specified velocity and specified pressure outflow Boundary condition is used for outlet. After performing the flow simulation for various discharges in Flow-3D code, the numerical results have been obtained. The results are given in Table 2.

Table 2. CFD Bucket Pressure

Type	**Discharge Q (m3/s)**	**Pressure P (Pa)**		
		At lip of bucket (1)	**At center of bucket (2)**	**At entrance of bucket (3)**
Q_1	0.00962	974.70	974.70	121.45
Q_2	0.005	429.54	429.54	76.15
Q_3	0.00255	229.52	229.52	21.77

ISSN: 2233-7857 IJFGCN

4. Result and discussions

Figure 2 and Figure 3 plots the pressure variation on the ski-jump bucket for different values of the discharge. It is observed that the value of pressure decreases as the discharge decreases. For various discharges, the maximum pressure occurs near the midpoint of the bucket length. The difference between the theoretical and CFD results varies from 8.94% to 180.69% for various discharges. It can be seen that the theoretical and CFD results are in agreement with each other.

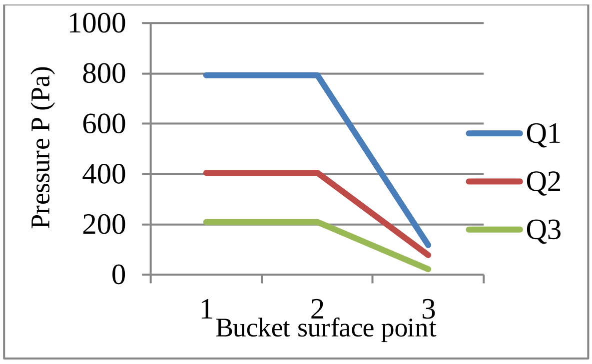

Figure 2. Experimental bucket pressure

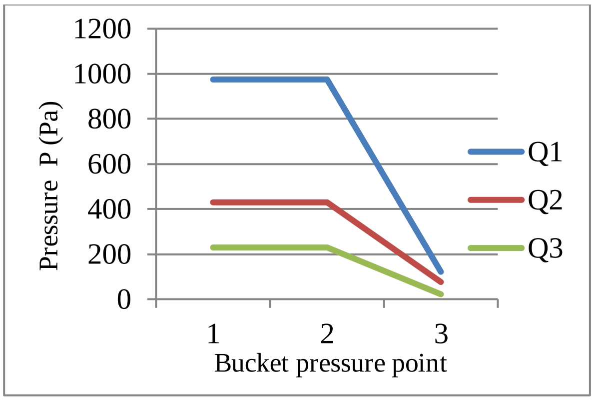

Figure 3. CFD bucket pressure

5. Conclusions

The CFD results have been compared with those achieved via the experimental study. This comparison indicated that the results of these two approaches are closer for all discharges. Also, the numerical results obtained via Flow-3D code for the pressure variation diagram in the bucket's length are in close agreement with the available experimental achievements in the literature.

References

[1] S. Khani, M. A. Moghadam and M. Nikookar, "Pressure Fluctuations Investigation on the Curve of Flip Buckets Using Analytical and Numerical Methods." CRPASE., **(2017**), 03(04), 165-171.

[2] B. M. Savage, and M. C. Johanson, "Flow over ogee spillway: physical and numerical model case study," J. of Hydraulic Engineering., ASCE 127 **(2001),** 640–649.

[3] F.A. Bombardelli, I. Meireles, J. Matos, "Laboratory measurements and multi-block numerical simulation of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways," Environ Fluid Mech., Springer 11, **(2011),** 263–288.

ISSN: 2233-7857 IJFGCN

[4] D. K. Ho, H.K.M. Boyes, S.M. Donohoo, "Investigation of spillway behaviour under increased maximum flood by computational fluid dynamics technique," 14th Australian FluidMechanics Conference., Adelaide University, Adelaide, Australia, **(2001).**
[5] G. Heidarinejad, and A. Najibi, "Two-dimentional analysis of flow at the toe of a dam," Proceedings of Intl. Conf. on Hydraulic Structures., Shahid Bahonar Kerman University, Kerman, **(2001).**
[6] S. Eklund, "CFD modelling of ski-jump spillway in Stornnforsen," Master's Thesis, Royal Institue of Technology, Sweden, **(2017).**

ISSN: 2233-7857 IJFGCN