



HAL
open science

Computational fluid dynamics (CFD) simulation on the hydraulics of a spillway

Satria Damarnegara, Wasis Wardoyo, Richard Perkins, Eric Vincens

► **To cite this version:**

Satria Damarnegara, Wasis Wardoyo, Richard Perkins, Eric Vincens. Computational fluid dynamics (CFD) simulation on the hydraulics of a spillway. 3rd International Conference of Water Resources Development and Environmental Protection (ICWRDEP), Oct 2019, Malang, Indonesia. pp.012007, 10.1088/1755-1315/437/1/012007 . hal-03832360

HAL Id: hal-03832360

<https://hal.science/hal-03832360>

Submitted on 27 Feb 2023

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution 4.0 International License

PAPER • OPEN ACCESS

Computational fluid dynamics (CFD) simulation on the hydraulics of a spillway

To cite this article: Satria Damarnegara *et al* 2020 *IOP Conf. Ser.: Earth Environ. Sci.* **437** 012007

View the [article online](#) for updates and enhancements.

You may also like

- [Numerical study on flow over stepped spillway using Lagrangian method](#)
Junmin Wang, Lei Fu, Haibo Xu et al.
- [Characterizations of flow over stepped spillways with steps having transverse slopes](#)
A S Ali and O S Q Yousif
- [Numerical and Physical Modelling of the Performance of the Pro-vortex Vanes in Shaft Spillways](#)
Miroslav Broucek, Ladislav Satrapa, Martin Kralik et al.



244th Electrochemical Society Meeting

October 8 – 12, 2023 • Gothenburg, Sweden

50 symposia in electrochemistry & solid state science

Abstract submission deadline:
April 7, 2023

Read the call for papers &
submit your abstract!

Computational fluid dynamics (CFD) simulation on the hydraulics of a spillway

Satria Damarnegara¹, Wasis Wardoyo¹, Richard Perkins², Eric Vincens³

¹Laboratory of Hydraulic and Coastal Engineering, Department of Civil Engineering, Sepuluh Nopember Institute of Technology, Surabaya, 60111, Indonesia

²Laboratoire de Mécanique des Fluides et d'Acoustique UMR CNRS 5509 Université de Lyon, Ecole Centrale de Lyon 36, avenue Guy de Collongue, 69134 Ecully, France

³Laboratoire de Tribologie et Dynamique des Systèmes UMR CNRS 5513 Université de Lyon, Ecole Centrale de Lyon 36, avenue Guy de Collongue, 69134 Ecully, France

E-mail: damarnegara@its.ac.id

Abstract. A spillway is one of the most important features of a dam. It should be able to allow flood discharge to flow and keep its structure stable. The force impacting a spillway is great; hence, it needs a proper design. The design philosophy of a spillway is to maximize its discharge while keeping the water level in the dam low. The discharge capacity of a spillway is mostly predicted based on empirical data derived from idealized conditions with laboratory experiment. To determine actual capacity, a physical test model is usually utilized. However, a physical model needs large amounts of space and time to prepare. Computational Fluid Dynamics (CFD) offers an interesting method to predict spillway capacity with less time and resources. In this paper, a simulation of flow over an Ogee spillway was performed using the open-source CFD platform called OpenFOAM. The flow is simulated based on the Volume of Fluid (VoF) method to resolve the dynamics of free surface. The simulation results show a good agreement with experimental measurements and the USBR method of calculation.

Keywords: CFD, OpenFOAM, Spillway, Free surface

1. Introduction

A spillway is considered a critical structure in a dam. Its capability to allow flood to flow is important to avoid failure in the dam structure. Generally, a spillway is designed based on the weir principle, where the flow rate is determined by the discharge coefficient C_d , spillway width B , and upstream energy height H_e . The spillway geometry is often limited by the site conditions; hence, to optimize the flow rate, a high discharge coefficient is important. Several shapes of the spillway are designed to optimize the C_d , with one popular shape being the Ogee shape. This is designed based on the water nappe on a sharp crested weir.

Spillway design guidelines are usually based on an empirical formulation for determining the discharge coefficient, with the most often used being from USBR [1]. The discharge coefficient is mostly determined by the spillway shape, with several assumptions made for the idealized condition. However, when applied on the field, some of the assumptions could be violated, and a physical model is needed to ensure the accuracy of spillway discharge capacity.

In recent times, the advancement of CFD code and computation resources provides an interesting alternative to aid spillway design by using numerical models to give a more accurate representation.



However, this approach is challenging, because the flow in the spillway will consist of supercritical flow and hydraulic jump, which will need a correct calculation of free surface dynamics. Despite its difficulties, previous studies have shown great results in employing numerical models to predict flow discharge. Numerical simulation of an Ogee spillway in 2D using the commercial code of Flow3D [2, 3] based on the multiphase Volume of Fluid method shows a good agreement of water surface elevation and pressure compared with the experiment. Another study implemented the ADINA numerical model [4], wherein the free surface dynamics are calculated using a mesh deformation method. Although it showed promising results, tracking the free surface using mesh deformation was difficult for a rapidly varying flow problem.

In this paper, an open source CFD toolbox called OpenFOAM® is applied to an Ogee-shaped spillway model. The multiphase *interFoam* solver is applied to the model for free surface dynamics with a shape-fitted mesh to represent the spillway. The numerical simulation is compared with the rating curve from experimental measurements and the USBR calculation method in order to see its capabilities as a designing tool.

2. Materials and Methods

The spillway discharge is calculated based on the general discharge formulation for a weir, given as:

$$Q = CBH^{1.5}$$

where Q is the spillway discharge (m^3/s), B is the width of the spillway (m), and H is the water head behind the spillway (m). C is the discharge coefficient which dependent on the spillway characteristic ($\text{m}^{0.5}\text{s}^{-1}$). The empirical discharge coefficient for an Ogee spillway is given in the manual by United States Bureau of Reclamation [1]. The Ogee spillway is designed using a planned water level of 10 cm.

2.1 Experimental Setup

An experimental measurement was conducted in the Laboratory of Hydraulic and Coastal Engineering of ITS in a flume with a width of 80 cm and a height of 50 cm (**Figure 1**). The discharge flow was supplied using two pumps with capacities of 13 l/s and 20 l/s respectively. The discharge was measured using a suppressed rectangular weir with a width of 47 cm. Water level was measured using a point gauge along the spillway crest.



Figure 1. Experimental set up

2.2 Numerical Model

The numerical model used in this work is based on the Volume of Fluid (VoF) method [5] for solving a free surface dynamics problem. The governing equation is based on the Reynolds-Averaged Navier-Stokes (RANS) equations for incompressible flow:

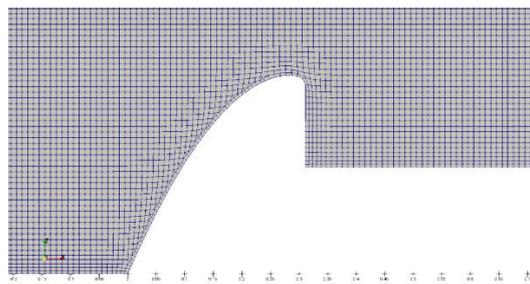
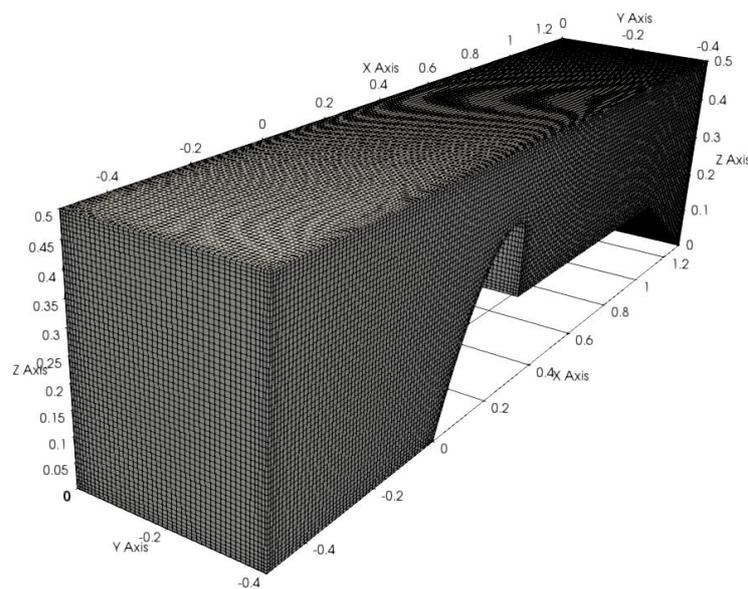
$$\nabla \cdot \mathbf{U} = 0 \quad (1)$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot \rho \mathbf{U} \mathbf{U}^T = -\nabla p + \nabla \cdot [\mu \nabla \mathbf{U} + \rho \mathbf{R}] + \rho \mathbf{g} + \mathbf{f}_\sigma \quad (2)$$

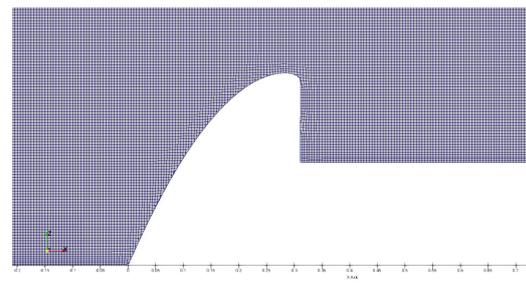
where \mathbf{U} is fluid velocity, p is pressure, μ is dynamic viscosity, \mathbf{g} is acceleration due to gravity, \mathbf{R} is the Reynolds tensor stress, and \mathbf{f} is surface tension. The VOF model takes into account two phases of fluid – water and air – by means of the volume fraction α that is advected based on fluid velocity, following the work from Rusche [6]:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \mathbf{U} \alpha + \nabla \cdot \mathbf{U}_r \alpha (1 - \alpha) = 0 \quad (3)$$

\mathbf{U}_r is the artificial compression velocity to avoid a mixing between two phases. The turbulence was modelled using the k- ϵ model as utilized by one study [7]. The numerical model was based on experimental observations. The numerical model domain is 3D, but only uses half of the spillway, assuming the flow is symmetrical. Two types of mesh were tested with maximum cell size $\Delta x = \Delta y = \Delta z = 1$ cm (coarse) and 0.5 cm (fine). The mesh was generated using *cfMesh* [8] as shown in **Figure 2**. For this test, an inflow discharge of $Q = 15$ l/s was applied in the inlet patch of domain. The outlet patch was set using a zero gradient velocity condition. A smooth wall condition was used for the wall function. The simulation was performed in a Newton cluster with a capacity of 40Tflops at *Pôle de Modélisation et Calcul en Science de l'Ingénieur et de l'Information (PMCS2I)* in 'École Centrale de Lyon.



(a)



(b)

Figure 2. Mesh configuration near the spillway for coarse mesh (a) and fine mesh (b)

3. Results and Discussion

The simulation was performed for 10 s using 32 CPU cores. The coarse mesh configuration took 1.61 hours to finish, while the fine mesh configuration took 39.13 hours. The location of free surface from the simulation was not explicitly calculated in the model; instead, the free surface location was taken as the location where volume fraction = 0.50. The water surface elevation results showed a good agreement with the experimental measurement, although the experimental measurement gave a higher water level than the CFD simulation (**Figure 3**). The difference between coarse mesh and fine mesh was not significant, even though the fine mesh configuration took 24 times longer than the coarse mesh configuration.

Varying the discharge in the experiment gave a consistent higher water level compared to the method of the USBR manual. The CFD simulation showed a better agreement compared to the calculation by the USBR manual (**Figure 4**).

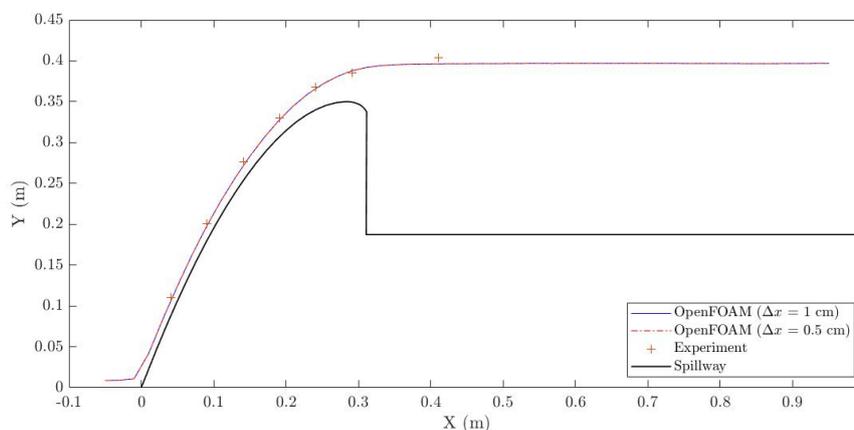


Figure 3. Water level on the spillway for $Q = 15$ l/s

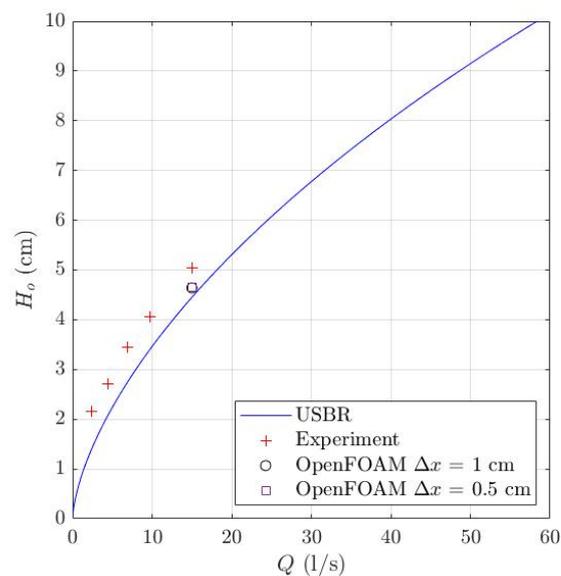


Figure 4. Ogee spillway rating curve

The velocity pattern for simulation shows that the maximum velocity reach 2.4 m/s (**Figure 5**). The experiment did not measure velocity, which makes a comparison impossible. The critical velocity U_c , for $Q = 15$ l/s with 0.8 m flume width gave $U_c = 2.35$ m/s, located near the end of spillway. The pressure acting on the surface of spillway shows low pressure on the spillway body (**Figure 6**).

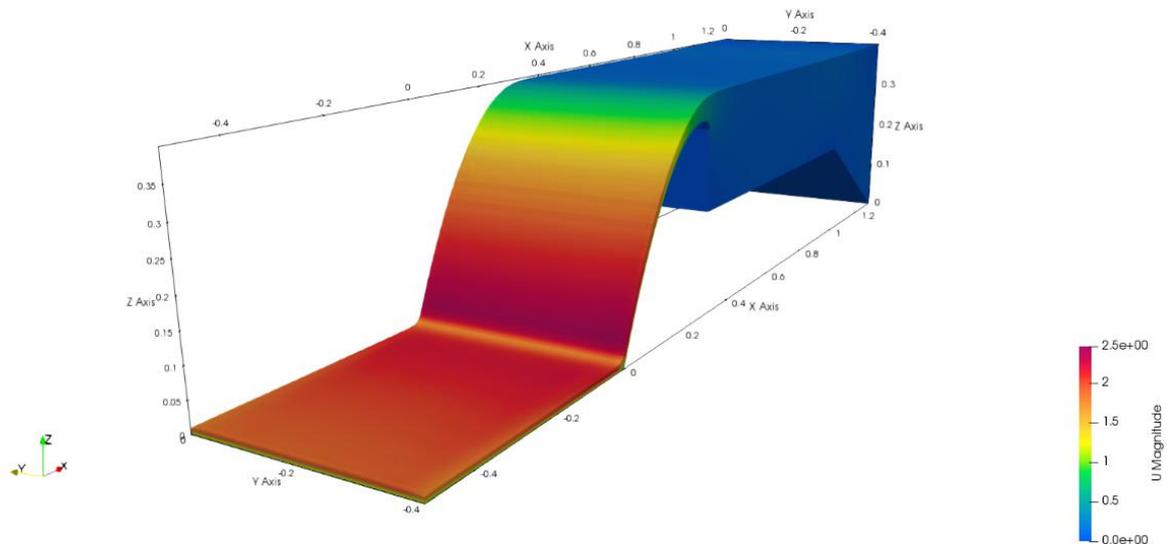


Figure 5. Velocity pattern on the spillway

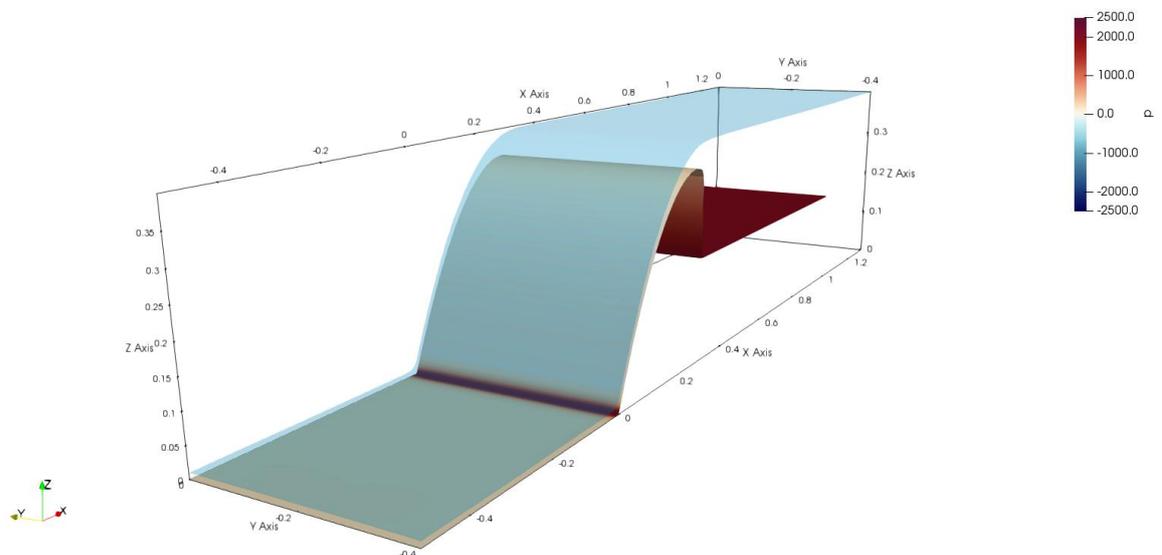


Figure 6. Pressure pattern on the spillway body

4. Conclusion

A numerical simulation for Ogee spillway was performed using the open source CFD code OpenFOAM®. The simulation is based on the VoF method to take account of the free surface dynamics. The results show a good agreement in the spillway rating curve compared to experimental measurements and the USBR calculation method. The mesh configuration shows no significant influence on the spillway rating curve. The numerical simulation is sufficiently accurate to be used in

design tools, especially in the design iteration process. This may save time before testing the design as a physical model.

Acknowledgments

The authors are grateful for financial support from the Institute of Research and Community Service (LPPM) of the Sepuluh Nopember Institute of Technology from the Novice Researcher program. The support from PMCS2I École Centrale de Lyon is also gratefully acknowledged.

References

- [1] USBR. 2014 .Appurtenant Structures for Dams (Spillways and Outlet Works) Design Standard (Washington: USBR)
- [2] B. M. Savage and M. C. Johnson .2001. Flow over ogee spillway: Physical and numerical model case study *Journal of hydraulic engineering* 127 640 .
- [3] M. C. Johnson and B. M. Savage .2006. Physical and numerical comparison of flow over ogee spillway in the presence of tailwater *Journal of hydraulic engineering* 132 1353 .
- [4] J. Chatila and M. Tabbara. 2004. Computational modeling of flow over an ogee spillway *Computers & structures* 82 1805 .
- [5] C. Hirt and B. Nichols .1981. Volume of fluid (VOF) method for the dynamics of free boundaries *Journal of Computational Physics* 39 201.
- [6] R. Henrik. 2002. *Computational fluid dynamics of dispersed two-phase flows at high phase fractions* (London: Imperial College).
- [7] B. Launder and D. Spalding .1974. The numerical computation of turbulent flows *Computer Methods in Applied Mechanics and Engineering* 3 269.
- [8] F. Juretić .2015. *cfMesh User Guide v. 1.1. 1* (Zagreb, Croatia: Creative Fields, Ltd)