Fluid/structure interaction study on the variation of radial gate’s gap height in dam

Fei Chong Ng\textsuperscript{1}, Aizat Abas\textsuperscript{2*}, Ismail Abustan\textsuperscript{1}, Z. Mohd Remy Rozainy\textsuperscript{3}, MZ Abdullah\textsuperscript{4}, Ali b. Jamaludin\textsuperscript{5}, Sharon Melissa Kon\textsuperscript{6}

\textsuperscript{1,2}School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, Nibong Tebal, 14300, Penang.
\textsuperscript{3}School of Civil Engineering, Universiti Sains Malaysia, Engineering Campus, Nibong Tebal, 14300, Penang.
\textsuperscript{4}School of Civil Engineering, Universiti Sains Malaysia, Engineering Campus, Nibong Tebal, 14300, Penang.
\textsuperscript{5}TNB Research Sdn. Bhd., No. 1, Lorong Ayer Itam, Kawasan Institusi Penyelidikan, 43000 Kajang.
\textsuperscript{6}School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, Nibong Tebal, 14300, Penang.

*Corresponding author: aizatabas@usm.my

Abstract. Through fluid/structure interaction (FSI) simulation and experimental work on a scaled-down dam model, the influence of radial gate height to the force exerted to the gate wall during water flow is studied. Both numerical and experimental findings gave results of great consensus, with their deviation not exceeding 14.66%. The water flow across the radial gate of the dam is visualized as modelled based on volume of fluid (VOF) scheme. It is found that the force exerted by water flow on the gate wall is the highest with intermediate gap height of 10 mm as compared to 8 mm and 10 mm.

1. Introduction

Precise planning and rigid monitoring are required for the design and construction of dams. This is because without proper handling, they can turn hazardous as they store huge volumes of water with high amount of potential energy and subsequently leads to catastrophic failures [1]. Issues that are often faced by dams here are old age, earthquakes, climate change, urban sprawl and landslides [2].

There are various works that particular research the flow and reliability in dam. Among the choice of numerical software used are: MIKE 11 1-D Hydrodynamic Model [3, 4], Smoothed Particle Hydrodynamics (SPH) model [5 – 8] and fluid/structure interaction (FSI) [9, 10]. Several researches have been done on the overall dam structure in terms of pressure and geometry. These studies are conducted using different types of models which have displayed certain inaccuracies and inflexibilities. The FSI model has proven to be one of the most reliable and commonly utilized model based on the studies done on a variety of scenarios and objects. Very little is focused on determining the flow of water as well.
To the best knowledge of the author, the research was rarely carried out to study the amount of force acting on a physical model of a dam, to ensure the structural integrity is not affected which will then be reflected on the real dam model. Therefore, the current study is a worthy research investment that can offer a new perspective to FSI simulation of concrete gravity dams.

2. Simulation

The dam model and its corresponding fluid domain after mesh are depicted in Figure 1. The length of the fluid domain is 3021.4 mm, width is 2000 mm and height is 600 mm. Meshing is done based on proximity and curvature size function with the relevance center and span angle center adjusted to the fine option. The mesh on both fluid and structural domains were optimized in terms of computational cost and numerical accuracy.

The zones of the boundary conditions in the FLUENT set-up are FSI, inlet, interior-fluid, outlet and wall-fluid. For the mixture phase of the inlet, velocity inlet boundary conditions are used to define the velocity and scalar properties of the flow. The magnitude of the velocity at inlet, \( v_{\text{inlet}} \), is 0.01 m/s so that it is nearly negligible and does not have a huge effect on the fluid flow while the wall-fluid velocity is 0. For the inlet’s water phase, the volume fraction under multiphase section is set at 1. Moreover, the pressure at the inlet and outlet equal the atmospheric pressure of 0 Pa (gauge). The general boundary conditions used are summarized in Figure 2.

![Figure 1. Meshed structural domain (left) and fluid domain (right).](image1)

![Figure 2. Schematic diagram of boundary conditions used in the numerical simulation.](image2)

The system coupling system facilitates utilizing multidisciplinary simulations between coupling participants which transfer and receive data in a coupled analysis. For this case, the analysis systems are Transient Structural and Fluid Flow (FLUENT). System coupling service manages the execution of analyses involves couplings between both the participants and two-way data transfers are conducted.

Firstly, data transfer is formed between the fluid solid interface (FSI) of the Transient Structural and Fluid Flow (FLUENT). When the data transfer happens, force from the fluid domain surface is
sent to the solid domain surface. At the same time, displacement variable is transferred from solid domain to the fluid domain surface. Next, the time step size is set at 0.1 s and the end time is 5 s consistent with settings provided in Transient Structural; while the minimum iteration is 1 and maximum iteration is 7. Finally, the coupled analysis is solved by updating the system coupling.

3. Experiment

The experiment is conducted on the physical dam model as shown in Figure 3 that located in the Civil School Research Lab to verify the simulation force data. When water flows through the gap underneath the radial gates, force is exerted on the gate surface. The purpose of the experiment is to find out the value of force acting on the surface. This is to ensure that the gate can withstand high pressure without undergoing permanent deformation.

In the experimental setup, the IEPE force sensor and accelerometer (see Figure 4) are fixed onto the steel bar by using magnets. The force sensor measures dynamic forces over a wide frequency range while the accelerometer is used to receive and monitor vibration. Both of them are connected to the LMS SCADAS Mobile (see Figure 5), a data acquisition hardware which collects signal in real time and transmits it to the Test.Xpress software platform. The software analyses and displays the vibration and force results. Then, the steel bar is held in place by a metal holder fixed on top of the radial gate. The sensor and accelerometer are pressed against the outer radial gate surface to receive incoming force and vibration from the water flowing pass the gate. All the data is sent to the software to be displayed and recorded. The experiment is carried out for dam model with varied gap heights, \( h = 8, 10, 12 \) mm.

![Figure 3. Experimental setup for the study of dam flow.](image)

![Figure 4. Force sensor of model 1053V1 IEPE (left) and accelerometer of model 3055B2 IEPE (right).](image)
4. Results and Discussions
In Table 1, the average value of the strains on the radial gate were calculated and used to determine the stress, using the Young’s modulus equation. From stress, the force component is obtained through normal pressure formula. Meanwhile, Figures 6, 7 and 8 display the gate strain data taken from each gap heights at time of 5.0 s. The red points on each figure indicated the one of the three points where the strain values were taken. Hence, Figure 5 shows the differences in visual form based on the comparison made in Table 1.

Figure 5. LMS SCADAS Mobile

Figure 6. Strain distribution for $h = 8$ mm at simulation time of 5.0 s.

Figure 7. Strain distribution for $h = 10$ mm at simulation time of 5.0 s.
Figure 8. Strain distribution for $h = 12$ mm at simulation time of 5.0 s.

For gap height of 8 mm, the strain values are taken at three points on the radial gate and the mean of them are found.

$$\bar{\varepsilon} = \frac{4.56213 \times 10^{-11} + 6.34291 \times 10^{-11} + 5.55232 \times 10^{-11}}{3} = 5.48579 \times 10^{-11} \quad (1)$$

Based on the Young’s modulus equation, stress is calculated when the strain is applied.

$$\bar{\sigma} = E\bar{\varepsilon} = 0.2085 \text{ Pa} \quad (2)$$

From stress, the force acting on the gate surface is obtained by dividing it with half the surface area of the gate, $A = 49940.5 \times 10^{-6}$ m$^2$.

$$F = \bar{\sigma}A = 0.0104 \text{ N} \quad (3)$$

The above calculations for force on radial gate were repeated for gap heights of $h = 10$ mm and $h = 12$ mm. The values of force that acts on the radial gate of various gap heights are then tabulated in Table 1 and later compared with experimental data in Figure 9.

| Gap Height (mm) | Strain ($\times 10^{-11}$) | Force (N) |
|-----------------|-----------------------------|-----------|
| 8               | 4.56213 6.34291 5.55232     | 5.48579   |
| 10              | 10.2563 11.1345 10.3286      | 10.5731   |
| 12              | 8.50078 4.96849 6.95983      | 6.8097    |

Based on Figure 9, it is seen that percentage deviations between the numerical simulation and experimental results is spanning from 9.59% to 14.66%, indicating the magnitude of forces exerted on radial gate. Such varying difference between the simulated data and experimental value is due to the method of experimental set up. The force is taken from the outer surface of the radial gate at two points. When the flowing water acts against the inner surface, the force is distributed throughout the surface thus the values vary at different areas. For instance, the upper area of the gate will experience less force compared to the bottom area. It does not come in direct contact with the flowing water. The
outer surface will be exerted with less force too due to energy loss in the form of sound and kinetic. Furthermore, the sensor and accelerometer may not be able to detect and transmit all the vibration motion and force as a result of lack of sensitivity. They are not calibrated and maintained consistently hence affecting the accuracy of outcomes. Whereas for the simulation part, the level of water enclosed is slightly different from the experiment. It uses a fixed and approximated value, the ability to replicate real-life fluctuating water flow scenario is limited.

**Figure 9.** Comparison of simulation and experimental forces acts on the radial gates of different gap heights.

5. **Conclusions**
This paper enclosed the numerical and experimental methodologies in determining the forces exerted on the radial gate in a dam during water flow. Both the forces data obtained through the fluid/structure interaction (FSI) numerical scheme and experimental works agree pretty with each other, with observed discrepancy less than 14.66%. This study eventually reported that at the force exerted on the radial gate is the highest at the configuration of gap height 10 mm while compared to the 8 mm and 12 mm gates. The methodology outlined in current paper would definitely benefit to future works relating to dam design and reliability analysis.

**Acknowledgements:**
The authors gratefully acknowledged the financial support provided by Institute of Postgraduate Studies (IPS), Universiti Sains Malaysia (USM), through the USM Fellowship 2017 scheme. Additional, this work was also partly supported by the Short Term Grant 60313020 from the Division of Research and Innovation, Universiti Sains Malaysia, and FRGS Grant 6071322 from the Ministry of Higher Education.

**References:**
[1] Energy Education on Dam failures. Available from: http://energyeducation.ca/encyclopedia/Dam_failures. [Extracted at 3 June 2017]
[2] Ismail, N. (2014). Issues and Problems towards the Sustainable Dam Management System in Malaysia, Faculty of Arts and Social Science, University of Malaya, Kuala Lumpur, Malaysia.
[3] Razad, A. A., & Ros, F. C. (2009). One Dimensional Dam Breach Modelling for Proposed Hydropower Development in Ulu, (October).
[4] Sidek, L. M., Ros, F. C., & Aziz, N. H. A. (2011). Numerical Modelling of Dam Failure for Hydropower Development in Cameron Highlands-Batang Padang Scheme, Pahang Malaysia, (November).

[5] Gomez-Gesteira, M. (2009). State-of-the-art of classical SPH for free-surface flows. Journal of Hydraulic Research, 48(extra), 0. Available from: <https://doi.org/10.3826/jhr.2010.0012>. [11 March 2017]

[6] Benedict, D., Crespo, A. C., Dominguez, J. M., Barreiro, A., & Go, M. (2011). GPUs, a New Tool of Acceleration in CFD: Efficiency and Reliability on Smoothed Particle Hydrodynamics Methods, Universidade de Vigo, Ourense, Spain and University of Manchester, Manchester, United Kingdom. Available from: <https://doi.org/10.1371/journal.pone.0020685>. [11 March 2017]

[7] Gomez-Gesteira, M. (2009). State-of-the-art of classical SPH for free-surface flows. Journal of Hydraulic Research, 48(extra), 0. Available from: <https://doi.org/10.3826/jhr.2010.0012>. [11 March 2017]

[8] Benedict, D., Crespo, A. C., Dominguez, J. M., Barreiro, A., & Go, M. (2011). GPUs, a New Tool of Acceleration in CFD: Efficiency and Reliability on Smoothed Particle Hydrodynamics Methods, Universidade de Vigo, Ourense, Spain and University of Manchester, Manchester, United Kingdom. Available from: <https://doi.org/10.1371/journal.pone.0020685>. [11 March 2017]

[9] Gómez-Gesteira, M and Dalrymple, RA (2004). “Using a 3D SPH Method for Wave Impact on a Tall Structure,” J Waterw Port Coast Ocean Eng, 130(2), 63-69.

[10] Mfundo Vezi (2014). Dynamic Modelling of Arch Dams in the Ambient State, University of Cape Town, Cape Town, South Africa.