Multidisciplinary Applied Research and Innovation Vol. 4 No. 2 (2023) 1-9

MARI

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/mari
e-ISSN :2773-4773

Numerical Modelling of Two Dimensional Dam Break with Heavy Flow using Ansys Software

Nik Ahmad Irfan Nik Muhammad Azmi[1], Muhammad Afiq Hakimi Amir[1], Anis Suzana Sukor[1] , Nur'Ain Idris[1]*

[1]Department of Civil Engineering, Centre for Diploma Studies,
Universiti Tun Hussein Onn Malaysia, Pagoh Higher Education Hub,
84600, Pagoh, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/mari.2023.04.02.001
Received 01 October 2022; Accepted 30 November 2022; Available online 15 January 2023

Abstract: In tracing the current era of modernization, simulation techniques have become an alternative option for measuring dam-break flow due to the rapidity of the development of computer technology. The primary objective of this project is to develop a model and simulate dam breaks based on the Lobovský's experiment by using Ansys software and a 2D heavy flow of dam break particle model. To confirm the conclusions acquired from simulation with Lobovský's experimental. The physical experimental carried out by Lobovský, included the length of the tank in this simulation of 1610mm. The reservoir part region is 600mm and the dry part is 1010mm. 5 sensors were placed in several places on the wall of the tank to take the results when running the simulation. To carry out this project, Ansys software was used to make the particle dam break model and then the simulation. The fourth sentence presents key findings and trends that can be observed from the data. The fifth sentence summarizes the discussion regarding those findings and some suggestions for future work. The major goal of this study is to estimate how much harm will be caused when the dam breaks. The use of simulation is a great way to predict how breaking the dam would affect the surrounding area. It can save costs and be produced on both a small and large scale by employing simulation. Additionally, since they can be performed anywhere using a laptop and the Ansys software (fluent), simulations don't waste time. This simulation uses the research from the article to validate the dam break's history and the water deformation.

Keywords: 2D, Simulation, Dam Break Model, Ansys Software, Numerical Modelling

1. Introduction

The dam failure will have a major impact on the impacted community, as it will result in property losses, as well as long-term emotional trauma, along with fatalities, destruction, and massive disruption of downstream ecosystems and economies. The most common and pervasive tragic incident is flooding.

*Corresponding author: ainidris@uthm.edu.my

The majority of research has focused on flood simulation prediction and early warning under ideal conditions [1]. Dam failures are frequently caused by structural problems, such as inadequate initial design or construction, or a lack of maintenance services [2]. The model was determined by the theoretical parameters for the fluid flow dam-break wave [3]. The experimental measurements were conducted for a dam-break flow across a horizontal dry bed in order to get insight into dam-break wave dynamics [4]. By doing simulations, it is easier and faster than doing real experiments because it can save costs as well as save time. Simulation is an attempt to mimic the features, appearance, and characteristics of a real system that is typically through a computer model. Simulation is one of the most widely used quantitative approaches to decision-making. Simulation allows the researcher to draw conclusions about a new study without building it or making changes to an existing system without disrupting it [5].

Over the historical period, there have been about 10 dam failures every year on average. The water levels at the Klang Gate Dam, Subang Lake Dam, and Semenyih Dam were discharged in stages on December 19, 2021, after they had exceeded the danger thresholds due to two days of nonstop rain [6]. Historically, the failure of a dam has serious physical and financial implications. Floods are one of nature's most destructive forces, and some of the most devastating flash floods are caused by dam failures [7]. Flood simulation has been studied using a variety of models, including 1D hydraulic models, 2D hydraulic models, 1D-2D linked hydraulic models, and hydrological models. The reservoirs at Klang Gates Dam and Batu Dam in the Gombak area have reached their capacity; extra water will overflow via the spillway and be routed to nearby rivers, but the pace of flow will be controlled [8].

Due to the problems faced before, and since no other researcher is studying this approach utilizing simulation, As a prediction tool that simulated the deformation of water from the dam-break problem. This will help in predicting the impact & propose the mitigation required.

2. Materials and Methods

The material and methods section explains the method and flows how numerical modeling of two-dimensional dam break with heavy flow done by validating the results of deflection from simulation with the experimental done by Lobovský [4], the making of dam break model, simulate dam breaks the model and validate the simulations. This simulation process had be conducted by using a high-performance computer at Campus Universiti Tun Hussein Onn Malaysia Pagoh. The purpose of this study is to elaborate on the methodological technique that was used to carry out the research based on the specific objectives of the study. Referring to the research from Lobovský [4], The length of the tank in this simulation is 1610mm. The reservoir part region is 600mm and the dry part is 1010mm. 5 sensors were placed in several places on the wall of the tank to take the results when running the simulation. The dimension of the dam was a bit different because of the air.

The simulation is conducted by following the schematic diagram before starting the process. This is because to make a plan before the sketching process in Ansys software. The schematic diagram was sketched using Microsoft PowerPoint as **Figure 1.**

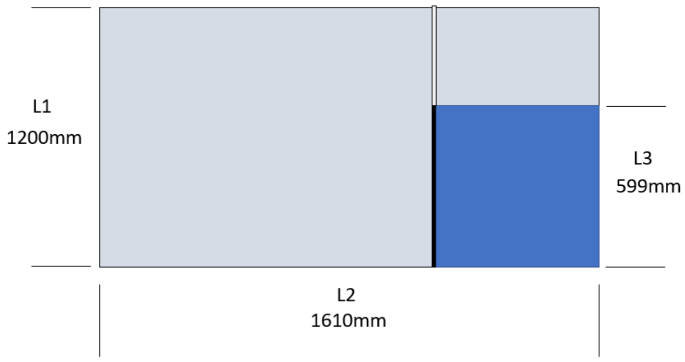

Figure 1: Schematic diagram from Microsoft PowerPoint

2.1 Validation

This section explained the validation of Ansys software simulation and experimental [4]. There is two verification that must emphasize it is verification with the Hydrostatic Test and Verification of 2D Dam Break to make sure the simulation is valid.

2.1.1 Verification with Hydrostatic Test

The first test is the most basic hydrostatic issue, which is a rectangular tank with a suitable surface and water within. The pressure distribution was compared between several boundary treatments that used the prior boundary treatment, the pseudo-Neumann boundary condition, and the suggested updated boundary treatment, MVMRG, which completely satisfied the Neumann boundary condition [9].

Obviously, the result produced by utilizing the modified boundary treatment of the Neumann condition, MVMRG, is nearly identical to the theoretical result. However, the Pseudo Neumann treatment yields a lower value, particularly towards the bottom surface, since the treatment applied at the boundary surface is not completely treated as non-homogeneous of the Neumann condition.

Before anything started, first of all, compared the pressure of the water tank with dimension 1500mm x 1500mm x 1500m hydrostatic test and theoretical as shown in **Figure 2.** The simulation result will be compared to experimental data from [4] earlier research. When the simulation goes well Ansys software will generate the data. This experiment will compare the deformation of fluid from the [4] experiment.

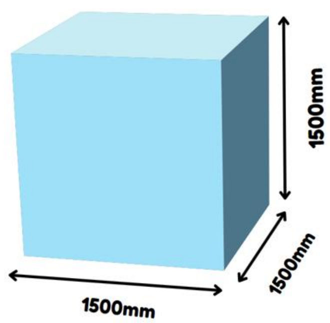

Figure 2 : The full tank with a cube-shaped Hydrostatic test

2.1.2 Verification of 2D Dam Break

The software used in this simulation is Ansys Software. Ansys software Engineers who need to make better, faster judgments can benefit from Ansys computational fluid dynamics (CFD) products. Ansys CFD simulation products have been proven and are well-known for their excellent computational power and precision. Reduce development time and effort while enhancing the performance and safety of the product.

2.2 Model & Simulate 2D Dam Break

The software that used in this simulation is Ansys Software. Ansys software Engineers who need to make better, faster judgments can benefit from Ansys computational fluid dynamics (CFD) products. Ansys CFD simulation products have been proven and are well-known for their excellent computational power and precision. Reduce development time and effort while enhancing the performance and safety of the simulation product.

2.2.1 Geometry

First of all open Ansys Workbench because Ansys Workbench is the main of Ansys. All Ansys such as Ansys Fluent, Polyflow, and CFX is in Ansys Workbench so next step the simulation can proceed by choose which Ansys want to use. In this simulation use Ansys Fluent.

In this simulation, Ansys 2021 R2 was employed. To begin, double-check that the units are in millimeters. Go to the dam's sketch function. Use the Rectangular feature and create a sketch based on the Lobovský dam's L1 599mm and L2 1610mm lengths. The gate measures 1010mm tall when measured from the dam's wall. There will be a space between the dam and the gate that we assume is a rubber 1mm from that area. **Figure 3** shown the interphase and function of Geometry part.

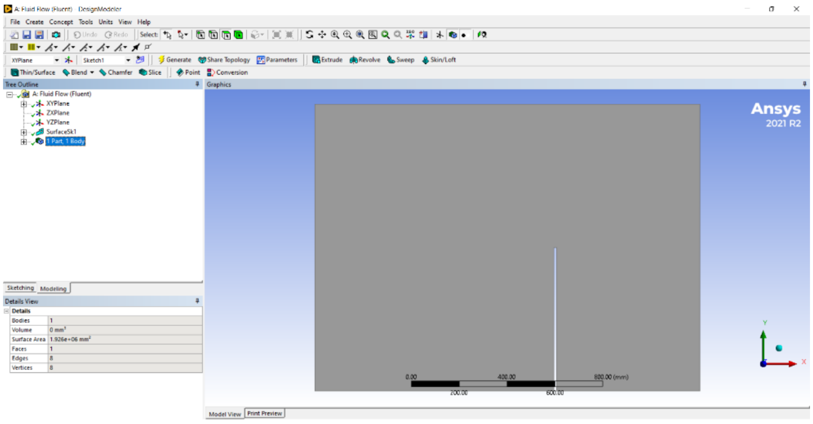

Figure 3: Geometry Part

2.2.2 Meshing

After the Geometry part, the simulation can proceed by selecting the Mesh. The mesh part is important because this part will determine the precision of the simulation. To begin, double-check that the units are set to millimeters. The smoothing function will be changed from medium to high. The number of elements is 201, while the number of nodes is 234. To alter the mesh from plain to square and triangle, use the method function. The element size will be 5 pixels (a smaller element size mesh will be more precise but take more time to generate). Named the dam is the outer, while the gate is the outlet. **Figure 4** shows the interphase of Ansys Fluent while doing meshing. After Meshing was done, the geometry of the simulation turn to green colour that’s means the meshing was done. So the simulation will continue with the setup.

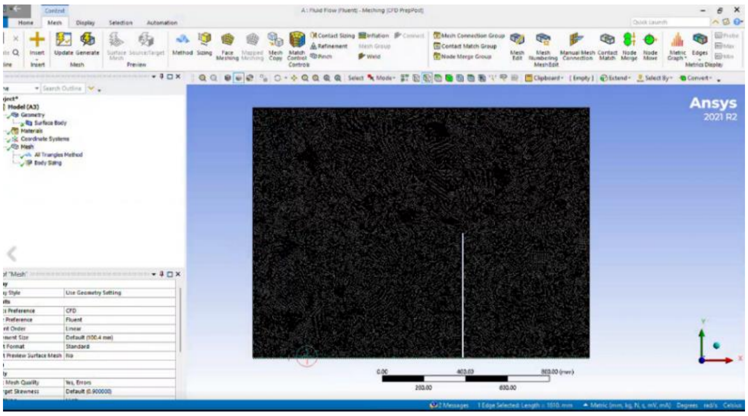

Figure 4: Interphase of Ansys Fluent while doing Meshing

2.2.3 Solution

After Meshing & Setup part the simulation can proceed by selected the Solution. Solution part is the last part then the experiment will get the video of the simulation. The transient solver will solve each time step, so the result will have reached a finite time. Insert gravity value in y-axis -9.81 m/s^{-2}. Then go to fluid database and click on water-liquid h20. Next set the multiphase model and activate the phases. Change surface tension coefficient to constant and insert 0.072 N/m. The dynamic mesh was used to smooth and meshing then insert the information. Then setting the time control to test whether the gate working or not. Then change the solution method from SIMPLE to PISO. PISO algorithm was proposed without iterations and with large time steps and a lesser computing effort, it was developed originally for non-interactive computation of unsteady compressible flow. Next change solution initialization from Standards Initialization to Hybrid Initialization. Hybrid Initialization solves a number of iterations of a simplified equation system and thereby gets usually a better guess for the flow variables, in particular for the pressure field. The number of time step was 10000 and Time step is 0.001. More number of time step will make the simulation more precise but it will take the longer time and the CPU of computer must the high performance. **Figure 5** shows the interphase of Ansys Fluent while doing calculation.

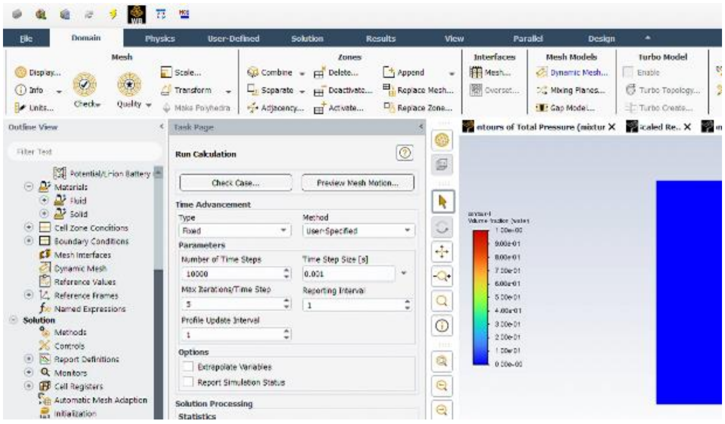

Figure 5: Interphase of Ansys Fluent while doing calculation

3. Results and Discussion

The results and discussion section presents data and analysis of the data from numerical modeling of two-dimensional dam break with heavy flow done by Ansys software. This section also gathers all data from Ansys software to test the water deformation. The result from Ansys software will discuss the comparison data from Ansys software and the experiment done by Lobovský et. al [4].

3.1 Hydrostatic Test

The first test is the simplest hydrostatic issue, which is a cube tank with dimensions of 1500x1500x1500mm and a suitable surface. The comparison of the pressure of water was made by comparing simulation water pressure and theoretical water pressure. Theoretical hydrostatic pressure in a liquid can be calculated as

$$p = \rho g h \qquad \textbf{Eq. 1}$$

where

p = pressure in a liquid (N/m^2),
ρ = density of the liquid (kg/m^3)
h = height of point or depth in the fluid where pressure is measured (m).

The equation shows that the greater the hydrostatic pressure produced by the fluid, the deeper an object is in the fluid. The layer of water acting on the edges of the tank exerts pressure. The pressure imposed by the top layer on the bottom increases as we walk down from the top of the tank to the bottom. This phenomenon is responsible for increased pressure at the tank's bottom. As indicated in **Figure 6**, the hydrostatic simulation pressure was utilized to compare with the theory of hydrostatic pressure value. Apparently, the results collected by performing the hydrostatic test are nearly identical to the theoretical values but the hydrostatic test has a slightly larger value than the theoretical value. A similar setup was then applied, but with the inclusion of various geometrical objects fully submerged with particle sizes decreased to 0.1 m.

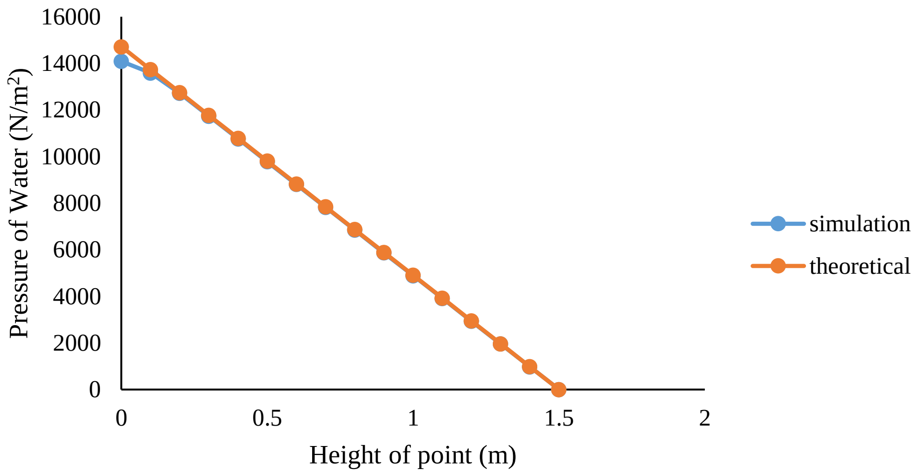

Figure 6:The comparison of pressure distribution with depth by using hydrostatic pressure between simulation and theoretical value

3.2 Ansys Software Simulation

Results data from Ansys software was compared with the experimental done by Lobovský [4]. There were 4 locations spotted named Deformation Stages A, Deformation Stages B, Deformation Stages C, and Deformation Stages D.

3.2.1 Deformation Stages A

Situation A illustrates that both the gate from the experimental and the result from Ansys are not opened and the position of water deformation is the same. It can be concluded that the size of the tank and the position of the gate from the wall of the tank is in the correct condition as.

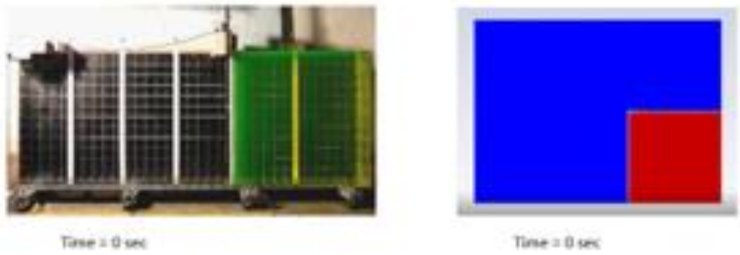

Figure 7: Deformation Stages A

3.2.2 Deformation Stages B

Situation B illustrates that the position of the water deformation touches the wall of the tank. The results from Ansys show a different time. Experimental by Lobovský [4] is 4 sec but the time from Ansys is 44 sec as **Figure 8**.

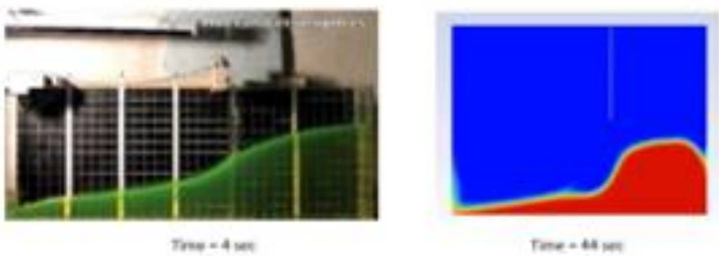

Figure 8: Deformation Stages B

3.2.3 Deformation Stages C

Situation C illustrates that the position of water deformation exceeds the position of the gate at 600mm from the right. The results from Ansys show a different time. Experimental Lobovský [4] is 7 sec but the time from Ansys is 50 sec as **Figure 9**.

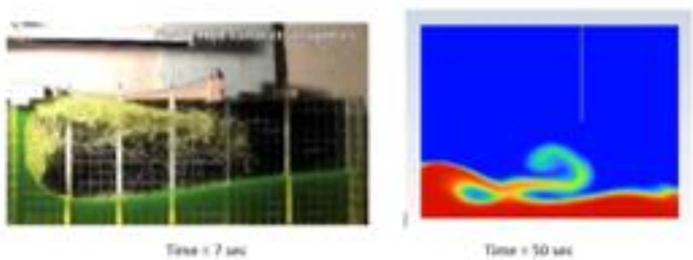

Figure 9: Deformation Stages C

3.2.4 Deformation Stages D

Situation D illustrates the position of water deformation at the end. the results from Ansys show a different time. Experimental Lobovský [4] 107 sec but the time from Ansys is 222 sec as **Figure 10**.

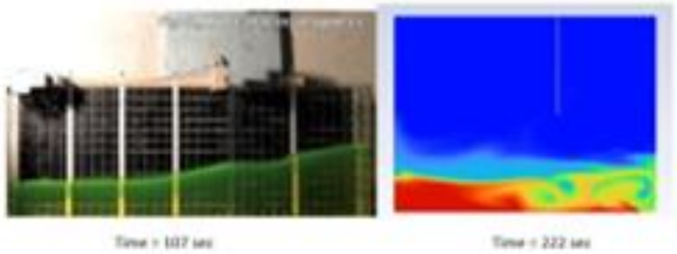

Figure 10: Deformation Stages D

The data from Ansys is not valid with the experiment done by Lobovský [4] because of a few factors. The factor that make Ansys software doesn't valid experimental [4] does not state the size of the rubber at the gate to make sure the water is from the dry reservoir and wet reservoir so when the simulation was created assume the rubber for the gate is 1mm so that the effect height of the gate. Other than that, the water density in the experiment done by Lobovský(2014) is 997 kg m^3 but in the simulation, the density of water is 998.2 kg m^3. The last factor for this simulation does not exist it is because kinematic viscosity in the experimental is $8.9 \times 10^{-7}\ m^2\ s^{-1}$ but in the simulation for the kinematic is auto-generated so the simulation doesn't show the value of kinematic viscosity.

Figure 11 shows the comparison between Ansys software and experimental [4]. The orange line is representative of an experiment [4]. Then the blue line representative for Ansys software simulation. The water deformation shown only in Situation A is the same data as the [4].

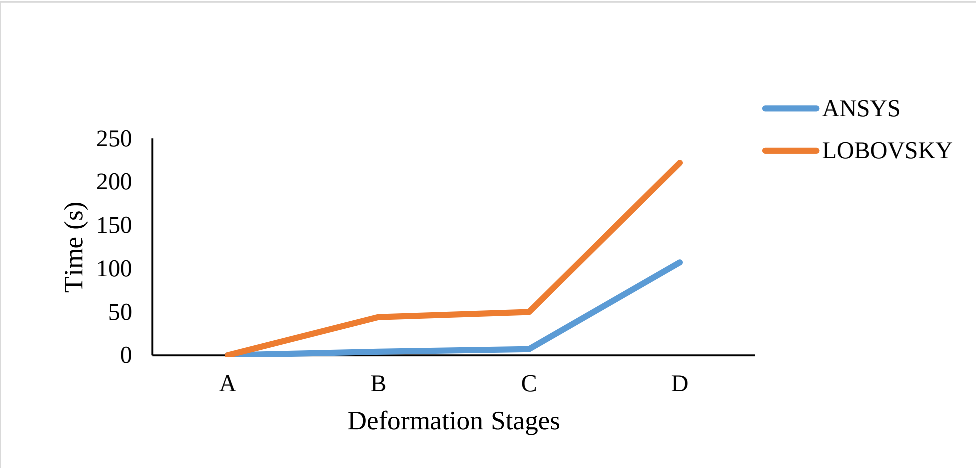

Figure 11: Comparison between Ansys software and experimental

Table 1: Comparison of deformation water between Ansys software & Experimental [4]

Situation	Ansys software(Time/s)	Experimental done by Lobobsky (Time/s)
A	0	0
B	44	4
C	50	7
D	222	107

Table 1 shows the data from Ansys software & Experimental done [4] The water deformation for Ansys software is slower than the experimental done [4], The water deformation is slower because the factor which is the hydrostatic test assumes rubber for gate, density of water and kinematic viscosity.

4. Conclusion

In conclusion, results data from Ansys software was compared with the experimental done by Lobovský and there is so many difference in time (second) for each situation between Lobovsky's experimental dam break data and the data from numerical modeling of two-dimensional dam break with heavy flow done by Ansys software. Ansys software's water movement is slower than the experiment. Water deformation is slower since the hydrostatic test assumes rubber for the gate, water density, and kinematic viscosity. The water deformation displayed in Situation A is the same as the results from Lobovský's experiment, with a slight delay from Situation A to Situation B. As a result, it is recommended to upgrade to better software that can simulate a better dam break and display viscosity or know the exact size of the rubber of the gate in order to achieve excellent data.

Acknowledgment

This research was supported by Ministry of Higher Education (MOHE) through Fundamental Research Grant Scheme (FRGS/1/2020/TK0/UTHM/03/16) or Vot No. K315. The authors would also like to thanks the Spatial Technology For Civil Engineering (STFORCE), Centre for Diploma Studies (CeDS), Research Management Centre, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Li et al., "A rapid 3D reproduction system of dam-break floods constrained by post-disaster information. Environmental Modelling & Software," 139, 104994, 2021, https://doi.org/10.1016/J.ENVSOFT.2021.104994.

[2] LaRocque et al., " Experimental and Numerical Investigations of Two-Dimensional Dam-Break Flows. Journal of Hydraulic Engineering," 139(6), 569–579, 2013, https://doi.org/10.1061/(ASCE)HY.1943-7900.0000705

[3] Xu et al., "SPH simulations of 3D dam-break flow against various forms of the obstacle: Toward an optimal design. Ocean Engineering," 229, 2021, https://doi.org/10.1016/J.OCEANENG.2021.108978

[4] Lobovský et al., "Experimental investigation of dynamic pressure loads during dam break. Journal of Fluids and Structures," 48, 407–434, 2014, https://doi.org/10.1016/J.JFLUIDSTRUCTS.2014.03.009

[5] S. Wilson, " Understanding Bottlenecks: An Operations Management Experiential Learning Exercise. Decision Sciences Journal of Innovative Education," 16(3), 166–184, 2018, https://doi.org/10.1111/DSJI.12162.

[6] S. A. Razali and N. Parzi," Police Standby Following Emergency Water Discharge Klang Gates Dam", https://www.nst.com.my/news/nation/2021/12/755739/police-standby-following-emergency-water-discharge-klang-gates-dam. [Accessed 23 June, 2022]

[7] E. Psomiadis et al., "Potential Dam Breach Analysis and Flood Wave Risk Assessment Using HEC-RAS and Remote Sensing Data: A Multicriteria Approach. Water," 13(3), 364, 2021, DOI: 10.3390/w13030364.

[8] Bernama, "The public advised to stay away from river areas near dams in Selangor," https://asianewstoday.com/the-public-advised-to-stay-away-from-river-areas-near-dams-in-selangor/. [Accessed 23 December, 2021]

[9] N. Idris et al., "The Modification of Boundary Treatment in the Incompressible SPH for Pressure Calculation Accuracy on the Solid Boundary", MATEC Web Of Conferences, 47, 02018, 2016, doi: 10.1051/matecconf/20164702018