

PAPER • OPEN ACCESS

## Flow Characteristics Investigation On Trapezoidal Weir Using FLOW 3D

To cite this article: O. Setyandito *et al* 2022 *IOP Conf. Ser.: Earth Environ. Sci.* **998** 012013

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

You may also like

- [Mass-Transfer Enhancement in Large-Diameter Pipeline under Water/Gas Stationary Slug Flow](#)  
Hongwei Wang, Howard D. Dewald and William P. Jepsen
- [Constant Froude number in a circular hydraulic jump and its implication on the jump radius selection](#)  
A. Duchesne, L. Lebon and L. Limat
- [Vertical drafts and mixing in stratified turbulence: Sharp transition with Froude number](#)  
F. Feraco, R. Marino, A. Pumir et al.

### ECS Toyota Young Investigator Fellowship



For young professionals and scholars pursuing research in batteries, fuel cells and hydrogen, and future sustainable technologies.

At least one \$50,000 fellowship is available annually.  
More than \$1.4 million awarded since 2015!



Application deadline: January 31, 2023

**Learn more. Apply today!**

# Flow Characteristics Investigation On Trapezoidal Weir Using FLOW 3D

O. Setyandito<sup>1</sup>, Novandy<sup>1</sup>, Juliastuti<sup>1</sup>, S. Christian<sup>1</sup>, R. T. Lopa<sup>2</sup>

<sup>1</sup>Civil Engineering Department, Faculty of Engineering, Bina Nusantara University, Jakarta, Indonesia 11480

<sup>2</sup>Civil Engineering Department, Faculty of Engineering, Hasanuddin University, Gowa, Indonesia 92171

Corresponding Author: osetyandito@binus.edu, novandy@binus.edu

**Abstract.** Weir is a building that is formed to change the nature of river flow to accommodate irrigation needs. The main purpose of a weir is to raise the river water level, drain the channel through the intake channel, allow water to be tapped, control all flow, etc. If a flow experiences a large shock, it will cause a hydraulic jump resulting in a transition of flow from supercritical to subcritical. The analysis can be performed, it was done using the Computational Fluids Dynamics (CFD) software, FLOW-3D. To model the 3-D weir used in this FLOW-3d, AutoCAD is used and finally, Flow Sight will be used to display visualization results such as graphics and videos. This paper investigates the Froude number and depth-averaged velocity on the trapezoidal weir. Moreover, the numerical modelling of trapezoidal weirs has been compared with the experimental result and numerical modelling of other types of weirs. The results show that the Froude number is proportional to the average velocity of this depth because the Froude number increases with the increase in the average velocity. There is a difference with the flow that flows after the weir, in a trapezoidal weir the Froude number decrease slightly but after that, it increases again downstream.

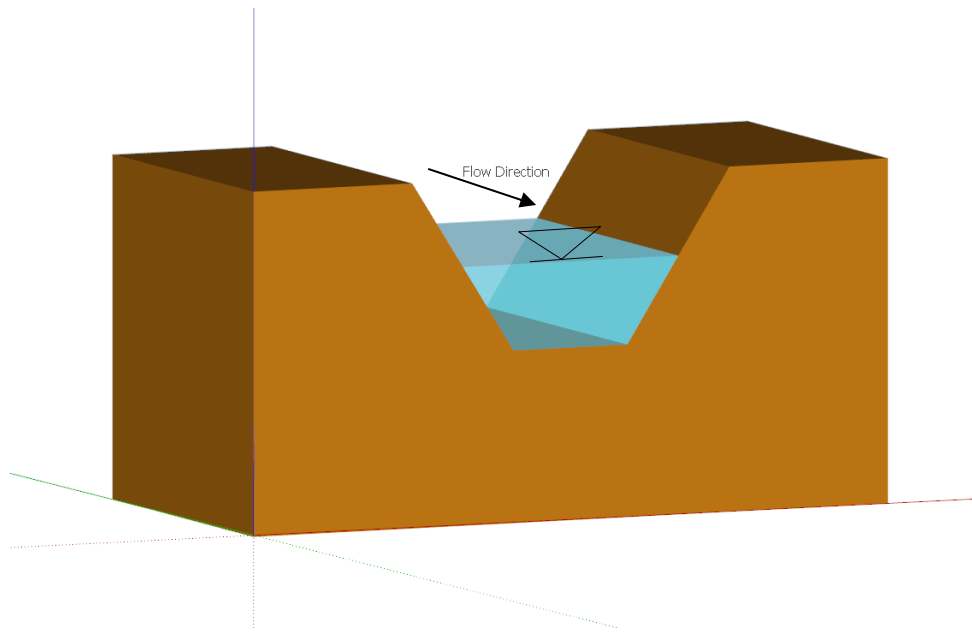
**Keywords:** Trapezoidal Weir, Froude Number, Flow-3D, open-channel flow

## 1. Introduction

Weirs are constructions designed to alter the properties of river flows to provide irrigation demands. The weir's primary role is to raise the water level of the dammed river, allowing water to be tapped and routed into the channel through the intake structure. Controlling flow, sediment movement, and river geometry is another role, allowing water to be used safely, effectively, and ideally. When the water passes over the weir, the difference in water level elevation between upstream and downstream of the weir will be relatively considerable, resulting in a plunge and a huge energy shift. As a result, the flow will undergo a normal shock or hydraulic jump, which occurs as the flow transitions from supercritical to subcritical. The occurrence of a hydraulic jump will result in scouring downstream of the weir, reducing the weir's stability [1]. In terms of engineering, a weir must be able to meet three basic requirements: hydraulic performance, structural stability, and environmental effect and safety considerations. Weirs are commonly used to perform one or more of the following basic functions: channel stabilization, water level management, flow measurement, environmental enhancement [2]. The quantity of discharge may be determined by measuring the head of water above the weir and utilizing a well-established head-discharge relationship. Weirs are classed as rectangular, triangular, trapezoidal,



parabolic, and other forms based on their forms. The most prevalent types of weirs are sharp-crested rectangular weirs, V- notch weirs, and broad crested weirs [3]. Trapezoidal weirs are a mix of rectangular and triangular weirs. These weirs combine the advantages of both weirs and eliminate individual constraints. The discharge via the trapezoidal weir will be a mixture of the two rectangular and triangular weirs, resulting in a relative increase in both weirs for a given head. An example of a trapezoidal weir is depicted in Figure 1. Sharp crested weirs are the most often used flow monitoring equipment in open waterways because they are generally precise. They are rarely utilized in field channels because of their significant disadvantage in the formation of afflux and considerable head loss in the form of coefficient of discharge [4].



**Figure 1** Cipolletti (Trapezoidal Weir)

The use of numerical models has improved the capacity to estimate weir flows during the last few decades. In such instances, a comprehensive three-dimensional (3D) flow model that solves for the three velocity components under non-hydrostatic circumstances may be required. However, the use of 3D flow models to model weir flows appears to be relatively restricted at the moment. Modelling structures like a weir, stilling basins, water intakes, fish ladders, and other instream structures has been done extensively with Flow-3D [5]. Studies of flow characteristics aided by Flow-3D have been done by [6]–[9] wherein their studies, the hydraulic structures such as dam, bridge abutment, as well as piers are done to study the influence of the structures' shape with the flow characteristics. In another research [10], the Flow-3D is applied to understand the impact of tsunami waves on the wall bearing layout. These studies show that theoretical and experimental studies of free surface flow can be done. The findings of these studies might be used to model the flow that runs through the trapezoidal weir so that it can produce experimental results in the form of Froude Numbers and the average depth of distribution velocity of the trapezoidal weir. Then after getting the results of the two variables can be compared with previous discoverers that are in line with this topic.

As a dimensionless number, the Froude number can be used to compare with other studies. Froude number is typically used when friction loss is small or neglected in open channel flows or free-surface flows in the hydraulic structure such as spillways, weirs or bridge piers. To calculate the Froude Number of the flow, the formula (1) can be used where it defines the ratio of inertia forces and gravitational forces:

$$Fr = \frac{v}{\sqrt{gh}} \quad (1)$$

Where:

Fr = Froude Number;  
 V = Velocity of Fluid (m/s);  
 g = Gravitational Acceleration ( $\text{m/s}^2$ );  
 h = Channel Height (m)

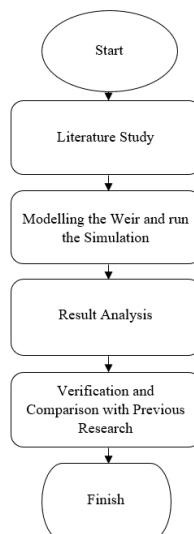
Rahimpour et al. [11] set both computational and experimental methods to study flow over a trapezoidal sharp-edged crest. The discharge coefficient in subcritical flow is connected to the Froude number, weir sidewall slope, crest length over upstream depth, and weir height over upstream depth of the side weir, according to the findings. The experimental study was done by Qasim, Abdulhussein, Hameed, and Matoonq [12] study the effect of the weir-gate combination of some composite shapes, like rectangular, triangular, and also trapezoidal to the flow's characteristic, such as the discharge coefficient and Froude Number. The research conducted by Al Shaikhli and Kadhim [13] develop an equation for flow over rectangular and V-Notch weir by using multi non-linear regression (MNL) and computational fluid dynamics (CFD) simulation approaches. Other than that, this experiment compares the result between experimental result, theoretical result, and of course the regression and numerical result. Al-Qadami et al. [14] show the numerical modelling for triangular hump weir whereas Daneshfaraz et al. [15] were studying the numerical modelling for the sloped broad crested weir. A comparison between the result of the flow characteristics of this study and the aforementioned studies will be done.

Based on its shape, the trapezoidal or Cipolletti weir is very similar to the V-Notch weir as both are categorized as a constricted weir. The difference is Cipolletti weir flow characteristics are highly affected by the dimension of the base of the weir [16], while V-Notch weir flow is affected by its opening constriction [17]. V-Notch weir itself is a sharp-crested weir that has a V-shaped opening. Both weirs are usually used for a channel with a relatively small flow discharge. Cipolletti weir is used in a channel with flow discharge of 200 – 2000 litres/second, while V-Notch is for flow discharge of < 200 litres/second.

In general, this study aims to review the fluid flow with a trapezoidal weir that is analyzed by using numerical modelling analysis, which was done using one of the computational fluid dynamics (CFD) software, FLOW-3D. AutoCAD is used to model the 3- dimensional weir that will be used in the FLOW-3D. Lastly, FlowSight is used as an advanced post-processing software of FLOW-3D that is useful for showing the visualization results, such as graphs and simulation video, that were simulated in FLOW-3D.

## 2. Research Methods

To achieve the objective, there are several steps on this study. Figure 2 shows the research flowchart for this study.

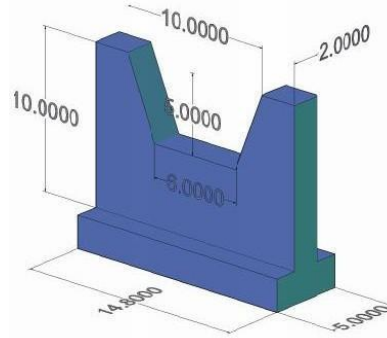


**Figure 2.** Research Flowchart

### 3. Model and Simulation

#### 3.1. Physical Model.

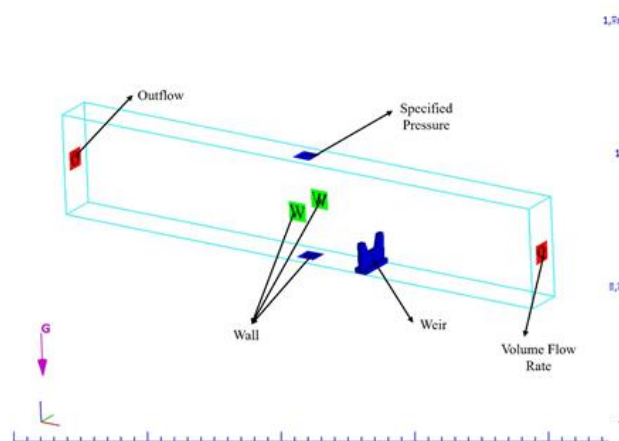
Geometric model is modelled by using AutoCAD. The geometric model as shown in Figure 3 has length of 14.8 cm, height of 12 cm. The weir crest has length of 6 cm of bottom side, length of 10 cm of upper side, height of 5 cm and side wall slope is 2H:5V



**Figure 3.** Dimension of Trapezoidal Weir

#### 3.2. Boundary Setup

In Flow 3D modelling there are several boundaries that must be define. The geometric model is imported in STL format to FLOW-3D. Using the mesh boundary to make rectangular flume with length of 210 cm, width of 14.8 cm, and the height of 40 cm as shown in Figure 4. The dimension of the flume is adjusted based on the flume in laboratory of civil engineering at Bina Nusantara University.



**Figure 4** Boundary Setup

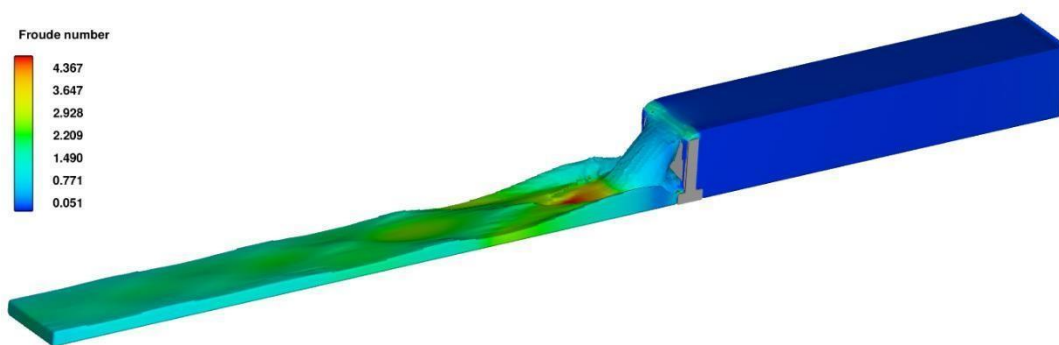
An outflow (O) boundary condition was used for outlet flow. The outflow boundary condition was located at X – min boundary. The specific discharge (Q) boundary condition was used for inlet flow. The specific/inlet flow discharge used is  $0.00336 \text{ m}^3/\text{s}$ . The specific discharge is obtained from laboratory experiment. The wall (W) boundary conditions were acted as a flume and flume bed. Assume the wall boundaries condition do not have friction and no-slip condition. The top of the mesh was acted as open – channel condition which is 0 Pa gage and friction is 0. Figure 4 represent the annotations of boundary conditions of sharp crested trapezoidal weir in Flow – 3D application. The size of mesh used is 0.005 m. It takes 8 hours 41 minutes 3 seconds to render 40 seconds of simulation of trapezoidal weir

with specification laptop; Processor: Intel Core i7-8565U @ 1.8 Hz with 8 cores; RAM: 8GB; GPU: Intel UHD Graphics 620.

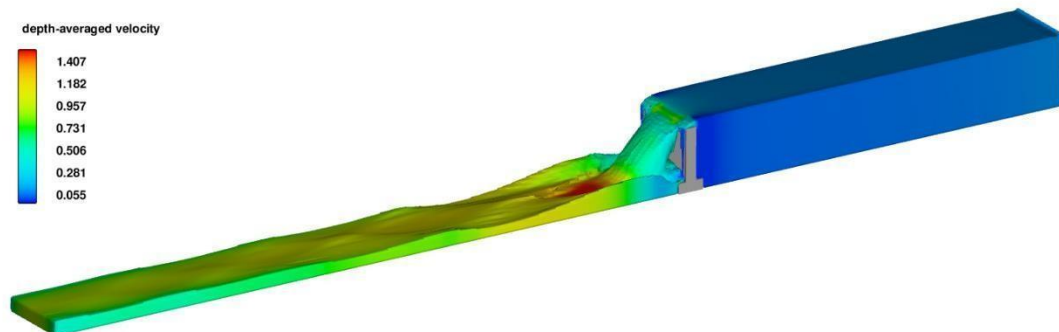
## 4. Result and Discussion

### 4.1. Flow-3D Result for Trapezoidal Weir

By using the computational flow dynamics (CFD) software, which in this paper is FLOW-3D. The flow over trapezoidal weir can be easily modelled. In Figure 5 and Figure 6, it can be seen the result of the experiment of the trapezoidal weir by using FLOW-3D. The figure will show the Froude Number and depth-averaged velocity distribution of trapezoidal weir at  $t = 40$  second

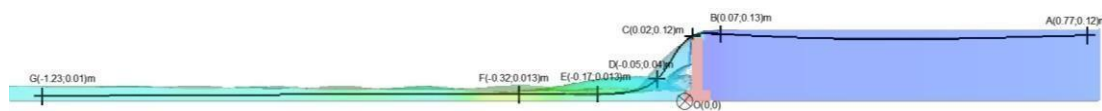


**Figure 5.** Froude Number Distributions of Fluid at  $t = 40$  Second in 3D simulation

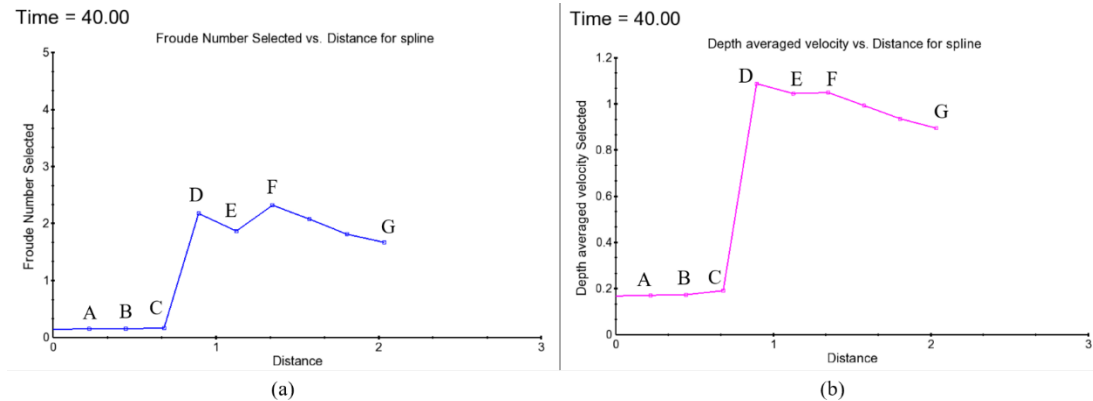


**Figure 6.** Depth-averaged Velocity Distribution of Fluid at  $t = 40$  Second in 3D simulation

Based on Figure 5, it can be determined that the Froude Number in this modelling has a range of 0.051-4.367, While the depth-averaged velocity has a range of 0.055 m/s – 1.407 m/s. Figure 6 depicts the result shown in the FLOW-3D is matched with the typical flow over a sharp-crested weir. The nappe shows when the fluid or water flows downstream. It also shows that the condition of the flow is free to flow since the weir is not in submerged or drowned condition. In the process of plotting the graph, the point that will be analyzed has to be determined first. In this case, the spline probe is used to specify the point of the water level. The spline probe was chosen because the Froude Number and depth-averaged velocity can be analyzed along with the distance of the fluid and also because the spline probe can fit the model better. The  $x$  vs.  $z$  coordinates of every point can be seen in Figure 7, the  $y$  coordinates can be neglected since it's quite the same for every point. Which is for  $y$  coordinates is about at the centre of the  $y$  axis ( $\sim 0.07$  m).



**Figure 7** x vs. z Coordinates for Every Point of Water Level



**Figure 8.** Graphs of Numerical Result of Trapezoidal Weir. (a) Graph Between Froude Number and Distance (b) Graph Between Depth-averaged Velocity and Distance

From Figure 8, the Froude Number slightly increases from point A – C, it is because the water level is about the same at that point and which means the fluid velocity is also similar to each other. Then when the water flows downstream or to point D, the potential energy will change into kinetic energy that causes the fluid velocity to increase significantly. That is why, the Froude Number also significantly increases when the water flows from point C to D. Then the water will slow down a little bit when to water flows to point E, it can happen because when the fluid flows just to reach the downstream, some energies will be dissipated. It causes the flows to slow down a bit. But the fluid velocity will increase again at point F, since the water area will decrease significantly at the end of the nappe so that the Froude Number increases. Lastly, when the water flows to the outflow it will become more stable, hence the fluid velocity and Froude Number will also decrease, although it is not that significant. From the two figures before, the characteristic of the flow over trapezoidal or Cipolletti weir can be analyzed qualitatively. Table 1 shows that to see the more exact value of the Froude Number and depth-averaged velocity from Figure 7.

**Table 1.** Value of Froude Number and Depth-averaged Velocity in Every Point at 40<sup>th</sup> Second Condition

Point	Depth-averaged Velocity (m/s)	Froude Number	Flow Type
A	0.171	0.148	Subcritical
B	0.173	0.151	Subcritical
C	0.191	0.167	Subcritical
D	1.085	2.167	Supercritical
E	1.045	1.868	Supercritical
F	1.049	2.318	Supercritical
G	0.895	1.672	Supercritical

Based on the value of Froude Number and depth-averaged velocity from Table 1, it can be concluded that the depth-averaged velocity and Froude Number are quite proportional to each other. When the depth-averaged velocity is increased, the Froude Number is also increased too, and vice versa. This happens because Froude Number is the ratio between the inertial force (influenced by fluid velocity)

with the gravitational forces (influenced by wave velocity). Then the type of flow also can be determined by looking at the Froude Number value in every point. From point A to C, the flows are categorized as subcritical flow ( $Fr < 1$ ) meaning that the flow at point A through point C is quite deep and low velocity. While the flow from point D to point G are categorized as supercritical flow ( $Fr > 1$ ) where the flows are categorized in shallow flow (less than the critical depth) with high fluid velocity.

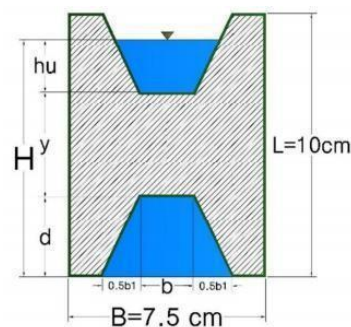
#### 4.2. Result Comparison

There has been a lot of papers and literature [12] that investigate the flow characteristics over trapezoidal notch weir. The experiments conditions themselves are very diverse, such as the effect of inclination, flow over trapezoidal side weir, the effect of the weir-gate combination, and many more. Based on [12], for the trapezoidal weir-gate itself, there are three models based on its dimension, which give some results shown in Table 2.

**Table 2.** Trapezoidal Model Dimensions and The Selected Results from the Experiments [12]

Model	Model No	$h_u$ (cm)	$y$ (cm)	$d$ (cm)	$b$ (cm)	$H$ (cm)	$Fr_{up}$	$C_d$
Trapezoidal	3--3--1	1	5	2	2	8	0.143	0.916
Weir-Gate	3--6--2	2	4	3	2	9	0.162	0.628
Combination	3--9--3	3	2	4	2	9	0.165	0.403

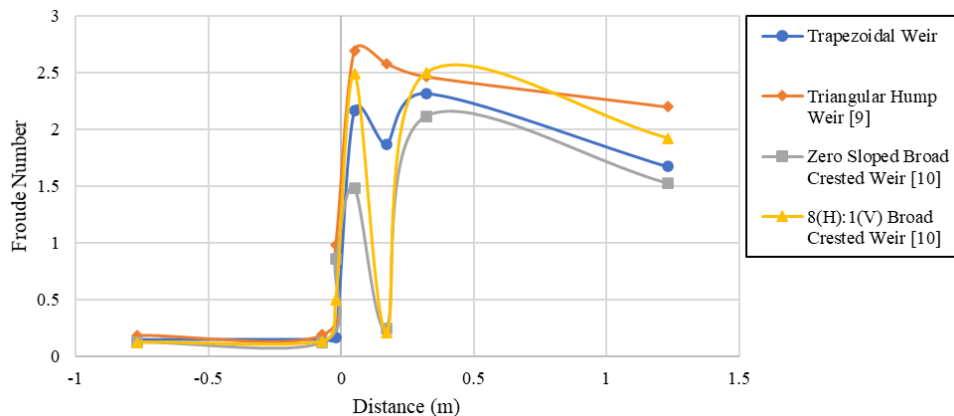
The depiction of the dimension of the weir-gate also can be seen in the Figure 9.



**Figure 9.** Dimension of Trapezoidal Weir-Gate Combination [12]

The slope of every models respectively is 3V:1H, 3V:1H, and 4V:1H. The crest head is also varied from 1-3 cm. Then, the Froude Number increases when the crest head ( $h_u$ ) increases. But the coefficient discharge ( $C_d$ ) decreases when the Froude Number increases. From Figure 5, it shows that the Froude Number above the crest is about 0.9894 showing that the result matched the other experiment done by other researchers. Other study done by Prakash, Kovoov, Principal, & Udupi [18] analyzed inclined trapezoidal weir. The result when the inclination is  $0^\circ$  (normal condition) also shows that the head over the crest proportional with the actual discharge. Then, discharge is proportional with the flow velocity, and flow velocity proportional with the Froude Number. Which also means the head over crest proportional to the Froude Number, just like shown in the results above.

Beside comparison between the same type of weir this study also comparing the result with other type of weirs

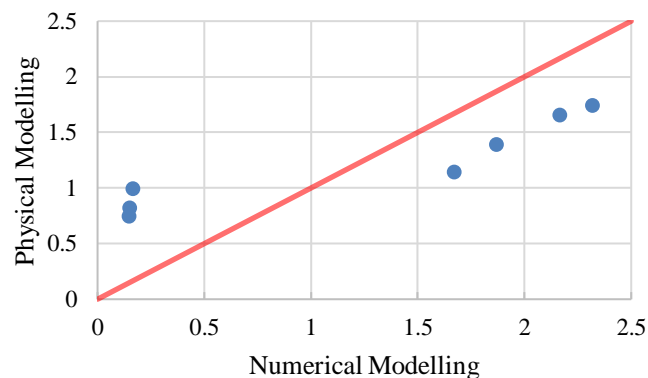


**Figure 10.** Comparison of Numerical Result Between Several Types of Weirs

From Figure 10 it can be seen that in every type of weir, the Froude Number is increasing significantly when the fluid flows down the weir (at distance  $-0.02\text{ m} - 0.05\text{ m}$ ). But the pattern is a little bit different from one another when the fluid flows after the weir. At the trapezoidal weir, the Froude Number shows a little decrement before it increases again downstream. This can be explained due to the energy dissipation at bottom of the nappe. The velocity will then increase as the cross-section area become smaller after the nappe.

At triangular hump weir, the Froude Number will gradually decrease, since the fluid becomes more stable after passing the weir, however it is still categorized as supercritical flow since the Froude Number is still more than 1. For the sloped broad crested weir, either its zero sloped or 8(H):1(V) sloped, the Froude Number will decrease significantly just after it passed the weir. The type of flow even changes from supercritical flow to subcritical flow caused by hydraulic jump. The Froude Number increased again since the cross-section area of the flow decreases. This pattern is a little bit different from the other two weirs. This can be explained due to the modelling was using turbulent flow, which can also affect the result of the Froude Number that happen at the weir. It is also important to remember that the condition of every result above is different. That is why the comparison is only good to get the flow characteristic differences. To get a more accurate comparison, the water profiles, volume rate, boundary condition, and other parameters has to be set in a comparable value. Hence, it comes highly recommended to do further research.

A comparison with a physical modelling of trapezoidal weir in a cutthroat flume [19] is also conducted. The Froude number were compared to understand whether the physical and numerical modelling will meet in a linear line. The result shows that for the numerical modelling of cipoletti weir to meet in the linear region between the physical modelling, a coefficient needs to be multiplied. This coefficient shows the different configuration of the structure, where the coefficient used is 2.5. The comparison can be seen in the Figure 11.



**Figure 11.** Comparison of Numerical Modelling and Physical Modelling

## 5. Conclusion

The result shows that the model corresponds with the usual flow over the sharp-crested weir with the Froude Number ranging from 0.051 to 4.367, while the depth-averaged velocity reaches the range of 0.055m/s – 1.407m/s. The flow experiences changes in its velocity and Froude Number along the channel until it finally reaches its stable form and final velocity value. This conforms with the physical modelling as well as other types of weirs that are compared in this study. To get accurate results, further research is needed to get a good comparison of the differences in characteristics.

## References

- [1] A. D. Pangestu *et al.*, “Studi Gerusan di Hilir Bendung Kolam Olak Tipe Vlughter Dengan Perlindungan Groundsill,” *Tekniksia*, vol. 23, no. Civil Engineering, pp. 463–473, 2018.
- [2] A. D. Ghare, V. A. Mhaisalkar, and P. D. Porey, “An Approach to Optimal Design of Trapezoidal Labyrinth Weirs,” *World Appl. Sci. J.*, vol. 3, no. 6, pp. 934–938, 2008.
- [3] B. Ramakrishna, S. Vaheed, and P. S. V Adilakshmi, “Flow Characteristics of Sharp Crested Weirs,” *Int. J. Adv. Res. Sci. Eng. Technol.*, vol. 5, no. 12, pp. 7650–7654, 2018.
- [4] M. N. S. Prakash and N. N. Shenoy, “FLOW MODELLING OF INCLINED TRAPEZOIDAL WEIR WITH A NEW APPROACH,” *Int. J. Sci. Environ. Technol.*, vol. 7, no. 3, pp. 823–834, 2018.
- [5] J. A. Vasquez and J. J. Roncal, “TESTING RIVER2D AND FLOW-3D FOR SUDDEN DAM-BREAK FLOW,” in *Canadian Dam Association 2009 Annual Conference*, 2009, no. October.
- [6] C. A. Pahlevi, O. Setyandito, Istiarto, K. A. Sudiyono, A. J. Pierre, and Juliastuti, “Oblong Bridge Pier Shape Influence toward Flow Velocity Characteristic and Scour Depth,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 794, no. 1, 2021, doi: 10.1088/1755-1315/794/1/012058.
- [7] C. A. Pahlevi, O. Setyandito, A. J. Pierre, Juliastuti, and M. Anda, “Flow Characteristics and Scour Depth in Lenticular Shaped Pier: Case Study at Ciujung Bridge, Banten,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 794, no. 1, 2021, doi: 10.1088/1755-1315/794/1/012059.
- [8] O. Setyandito, R. D. Alexander Michael, Juliastuti, J. P. Andrew, and Y. Wijayanti, “The effect of bridge abutment shape variation toward flow velocity characteristic,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 426, no. 1, 2020, doi: 10.1088/1755-1315/426/1/012035.
- [9] D. R. Lestari, O. Setyandito, M. Loren, Y. Wijayanti, and Juliastuti, “Flow Characteristics of Stilling Basin, Case Study: Karian Dam, Central Java Province,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 794, no. 1, 2021, doi: 10.1088/1755-1315/794/1/012066.
- [10] O. Setyandito, N. Yuwono, Nizam, F. U. Sjarifudin, Mitchell, and J. F. B. Saragih, “Hydrodynamic flow characteristics of tsunami waves impact on bearing wall layout,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 195, no. 1, 2018, doi: 10.1088/1755-1315/195/1/012016.
- [11] M. Rahimpour, Z. Keshavarz, and M. mehdi Ahmadi, “Flow over trapezoidal side weir,” *Flow Meas. Instrum.*, vol. 22, no. 6, pp. 507–510, 2011, doi: 10.1016/j.flowmeasinst.2011.08.004.
- [12] R. M. Qasim, I. A. Abdulhussein, M. A. Hameed, and Q. A. Matooq, “Experimental Study of Hydraulic Response for Combined Weir-Gate Flow of Composite Shape,” *Civ. Environ. Res.*, vol. 10, no. 11, pp. 6–14, 2018.
- [13] H. I. Al Shaikhli and K. N. Kadhim, “Development an equations for flow over weirs using MNLR And CFD simulation approaches,” *Int. J. Civ. Eng. Technol.*, vol. 9, no. 3, pp. 70–79, 2018.
- [14] E. H. H. Al-Qadami, A. S. id Abdurrasheed, Z. Mustaffa, K. W. Yusof, M. A. Malek, and A. A. Ghani, “Numerical modelling of flow characteristics over sharp crested triangular hump,” *Results Eng.*, vol. 4, no. October, p. 100052, 2019, doi: 10.1016/j.rineng.2019.100052.
- [15] R. Daneshfaraz, M. Dasineh, A. Ghaderi, and S. Sadeghfam, “Numerical Modeling of Hydraulic Properties of Sloped Broad Crested Weir,” 2019, doi: 10.22060/AJCE.2019.16184.5574.
- [16] M. S. L. M. Sandoval-Mendoza, J. A. Miranda-Escobar, C. O. Brown-manrique, A. Mujica-Cervantes, J. D. Bonilla-Rocha, and Y. Gallardo-Ballat, “Mathematical Models for Flow Estimation in Suro Weirs Used in Irrigation Systems Modelos matemáticos para la estimación

- del caudal en vertederos Sutro utilizados en sistemas de riego,” vol. 26, no. 1, pp. 57–65, 2017.
- [17] I. Amal, “Perencanaan Ulang Saluran Terbuka Disebabkan Oleh Penyempitan Bersudut pada Saluran yang Mengakibatkan Kehilangan Energi,” Muhammadiyah Malang University, 2020.
- [18] M. N. S. Prakash, G. M. Kovoov, V. Principal, and M. Udupi, “Inclined Trapezoidal Weir : Flow Modelling,” 2010.
- [19] D. Ran, W. Wang, and X. Hu, “Three-dimensional numerical simulation of flow in trapezoidal cutthroat flumes based on FLOW-3D,” *Front. Agric. Sci. Eng.*, vol. 5, no. 2, pp. 168–176, 2018.