3rd International Conference on Advances in Civil and Environmental Engineering (ICACEE-2024)
University of Engineering & Technology Taxila, Pakistan
Conference dates: 21st and 22nd February 2024; ISBN: 978-969-23675-2-3

Numerical Modelling of Flow Behaviour Over a Circular Crested Trapezoidal Weir

Saad Alam Shah[1, *], Usman Ghani[1], Muhammad Hasnain Tariq[1], Kashif Iqbal[1], Waleed Khalid[2]
[1]University of Engineering and Technology Taxila, Pakistan
[2]Moscow Institute of Physics and Technology (MIPT), Department of Informatics and Computational Mathematics Dolgoprundy, Russia
*Corresponding author: saadalam196@gmail.com

ABSTRACT

Circular crested weirs have been used for a long time due to their simplicity in design, low cost and accurate measurement. The Computational Fluid Dynamics (CFD) based software ANSYS FLUENT has been used in this investigation. The CFD model has been applied to explore various flow features over a circular crested trapezoidal weir. The volume of fluid (VOF) technique was used in this work. Two weir conditions with and without roughness elements and three discharges values for each weir case were used. The flow characteristics investigated include coefficient of discharge, water surface profiles and velocity profiles. The roughness element remained submerged in all the simulation cases. The k-ε turbulence model was used for the CFD simulation. After validation, the flow characteristics like the free surface profiles, coefficient of discharge, vertical velocity profiles at the upstream and downstream of the weir and the velocity contours were investigated for varying discharges and roughness. A comparison was drawn for these properties for a channel with and without roughness element.

KEYWORDS: Circular crested weir, CFD, Numerical Modelling, k- ε model

1 INTRODUCTION

A weir is a device (or overflow structure) that is placed normal to the flow direction. The weirs are used for essentially backing up the water in such a way that the water flowing over the weir reaches a critical depth. Weirs have been used for measurement of water for many years in open channels [1].

Weirs are one of the most important hydraulic structures that are needed when building a dam and are commonly used for discharging floods more than the capacity of the dam. Precision and simplicity for discharge measurement varies with the weir shape. Weirs can be built flat or curved depending upon area or local condition [2]. Weirs are classified into many types like broad crested weirs and sharp crested weirs. Out of the many weir crest shapes circular crested weirs are preferred. The reason for their preference is their simplicity in design, low cost and their stable overflow considered [3].

Circular crested weirs have many applications in water and sewerage systems, flow regulation systems and irrigation systems [4]. The circular crested weir type is designed with a circular crest of radius R, an upstream weir face that is perpendicular to the flow direction and a downstream face angle of 45° [5]. Rounding the weir crest has the advantage of enhancing the capacity of the weirs as well as easily overpassing the floating debris as well as characterizing the stable flow profiles [6].
In the present study, a circular crested trapezoidal weir was simulated using ANSYS Fluent. The numerical model was developed using VOF technique. VOF is a multiphase model that is used with air as the primary phase and water as the secondary phase. It helps in sketching the free surface profiles. Two weir cases with and without roughness elements against three discharge values were investigated during the simulation work. A comparison was made between the results obtained.

2 MATERIALS AND METHODS

2.1 Model Validation

The experimental data used for validation purposes was that of Schmoker et al. [7]. This experimental work was performed using a channel having a length of 7m, a width of 0.5m and the height of 0.7m as shown in Fig.1 [7]. The channel had a discharge capacity of 150L/sec. The channel had a circular crested trapezoidal weir that was placed at the centre of the channel. A 250mm conduit was used for discharging the flow into the channel.

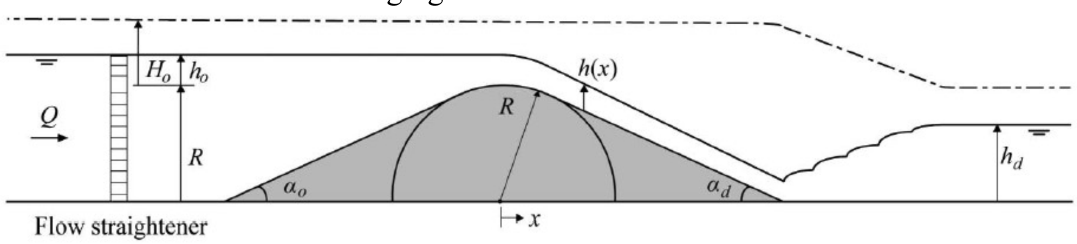


Figure 1: Channel used by Lukas Schmocker and Willi H. Hager [7]

Where, Q is the overflow discharge, R is the radius of circular crest, H_o is the approach flow energy head, h_o and h_d are the overflow depth at the upstream and flow depth at the downstream sides, α_o and α_o are upstream and the downstream weir angles
The numerical model was created in the ANSYS Fluent. The model was created with the same dimensions as used in the experimental work and the meshing was done on this model. To reduce the size of the mesh and to minimize the computational time the mesh was refined only in zones close to the weir.
The resulting mesh had number of cells equal to 32,000. The k- ε turbulent model [8] was used along with the VOF model in the CFD application. The following boundary conditions were used. The pressure inlet was assigned at the inlet. The top and outlet were taken as pressure outlet. The side walls and the bed were taken as wall. The channel with boundary conditions has been shown in Figure 2. After applying the suitable boundary conditions, the coupled solution method was used with the best suited under relaxation factors. The convergence criteria were set to be 10^{-3}[9].

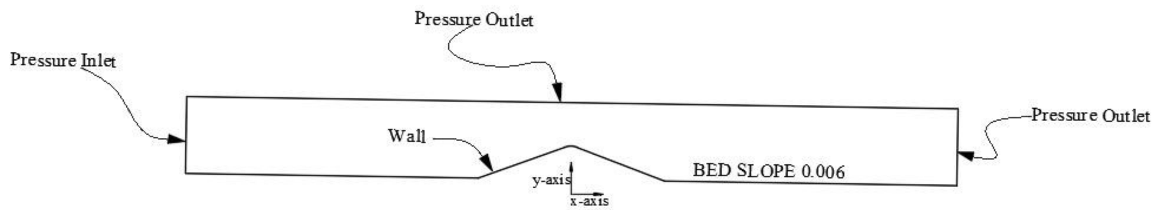


Figure 2: Boundary Conditions of Channel

Mesh independence was tested against three mesh resolutions i.e. 21,028, 32,000 and 55,860 (coarse mesh, medium mesh and fine mesh). Mesh sensitivity was tested against the grid convergence index (GCI) method, which is widely accepted and recommended method for estimating discretization error and has been applied to several CFD cases in previous studies [10]. The GCI is a measure of how far the simulation result is from the asymptotic numerical result. A small GCI means the simulation is within the asymptotic range [11]. It was found that the maximum uncertainty between the coarse and the fine mesh was 1.19% and 0.3% respectively. Hence the medium mesh of 32,000 was used for the simulation. After creation of the numerical model, the simulated results for coefficient of discharge were validated against the experimental results of Schmoker et al. [7] as shown in Figure 3. The measuring location for the coefficient of discharge was taken to be 3.5 m from the inlet.

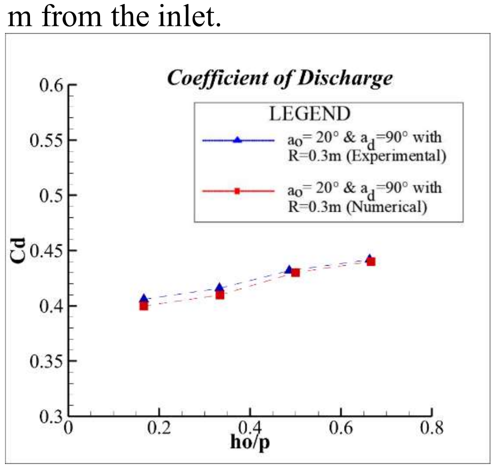


Figure 3: Validation of Co-efficient of Discharge

The results obtained from the numerical simulation were very close to the experimental results (the relative error was found to be within 1.5%) hence the simulation process for validation was considered satisfactory.

2.2 Simulation Setup

In this simulation work, a channel having a bed slope of 0.006 was used. Two weir cases with and without roughness elements and three different discharges for each case were used. Different geometric and hydraulic conditions of these cases have been shown in Table 1.

Table 1: Various Cases for Numerical Modelling

Case	Bed Slope	Flow depth (m)	Discharge (l/sec)	Roughness Element		
				Length (cm)	Width (cm)	Depth (cm)
C-1	0.006	0.50	25	nil	nil	nil
C-2			50			
C-3			75			
C-4			25	2.5	2.5	5
C-5			50			
C-6			75			

In case of weir with roughness elements, the elements having dimensions 2.5cmx 2.5cm x 5cm were placed at the top of the weir on with a centre-to-centre distance of 6.75cm as shown in Figure 4. The roughness elements remained submerged throughout the experimentation. Three types of discharges were considered for this analysis i.e. 25l/sec, 50 l/sec and 75l/sec.

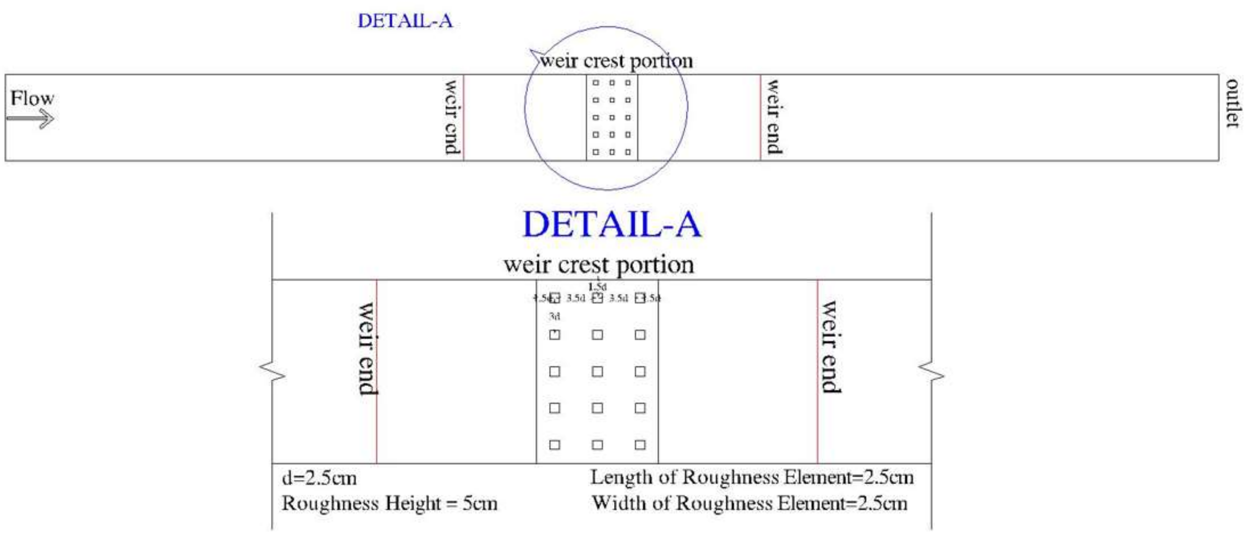
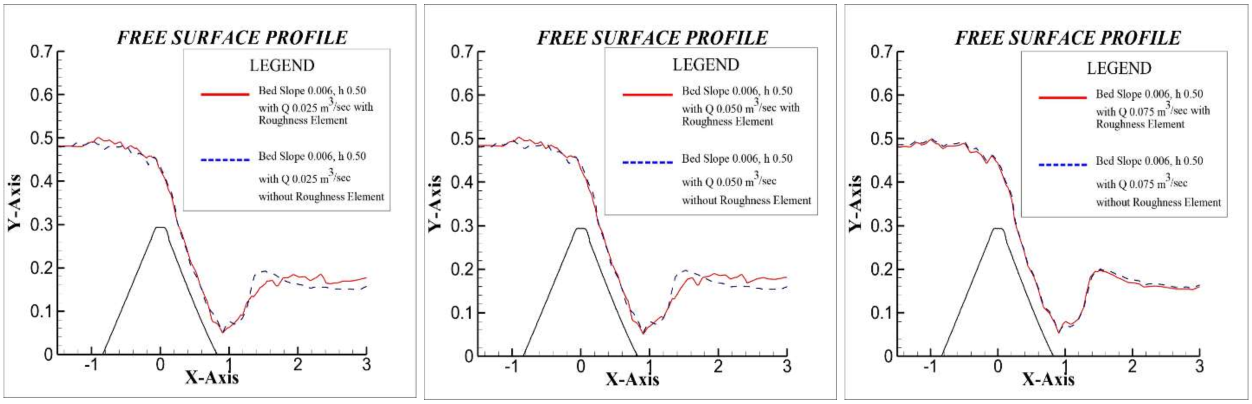


Figure 4: Roughness Elements in Weir

3 RESULTS AND DISCUSSION

3.1 Free Surface Profiles

The free water surface profiles have been calculated for the channel with the induced bed slope of 0.006 and the flow depth is considered for the case in which the roughness elements remained submerged. The graphs for free surface profiles have been plotted for the channel with and without the roughness as shown in Figure 5.



(a) (b) (c)

Figure 5: Free Surface Profiles (a) 0.025 m^3/sec, (b) 0.050 m^3/sec, (c) 0.075 m^3/sec

The free surface profiles on the downstream side of the hydraulic jump were slightly lower for weir without roughness elements whereas in the case of upstream side of the weir there was no significant change observed.

3.2 Coefficient of Discharge

The coefficient of discharge has been calculated for the channel with submerged roughness elements and without roughness elements. The results have been shown in Figure 6.

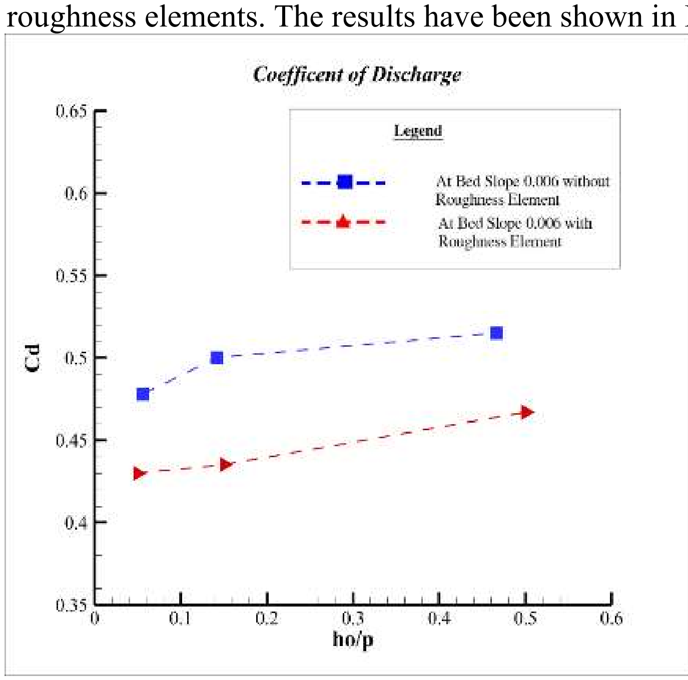


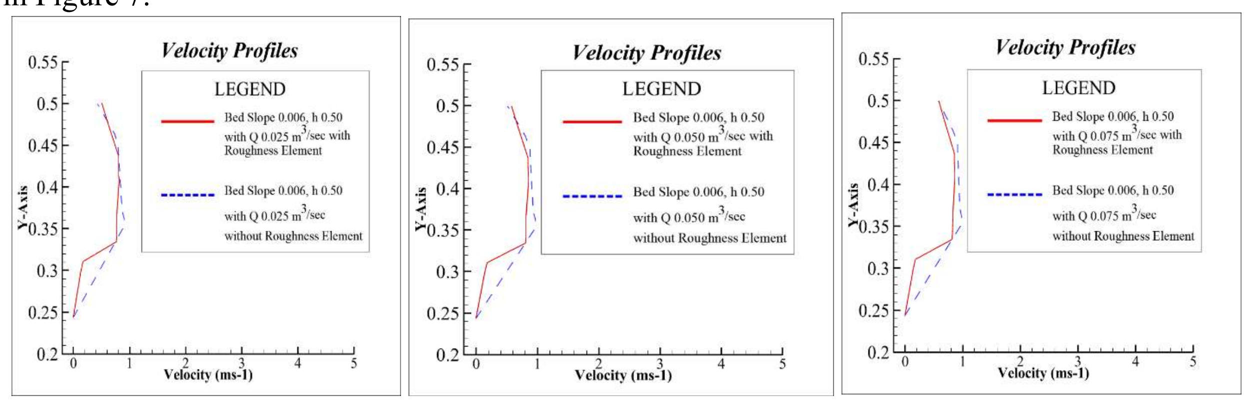
Figure 6: Coefficient of Discharge

Since the Coefficient of discharge is directly proportional to the discharge thus increasing the discharge would result in increased co-efficient of discharge. Whereas, the roughness element act as an obstruction resulting in resistance to flow [12]. Thus, the coefficient of discharge when measured at the centre for channels with roughness elements induced would result in low values in comparison to channels without any roughness elements. A reduction in the coefficient of discharge (i.e. 10-12%) was observed for the channel with roughness elements in comparison to channels without roughness elements.

3.3 Vertical Velocity Profiles

3.3.1 Upstream Side

The vertical velocity profiles were calculated at a distance of 3.35m from the channel inlet upstream of the weir for the channel with submerged roughness elements and without roughness elements. The comparison was drawn between these two cases for three discharge values as shown in Figure 7.



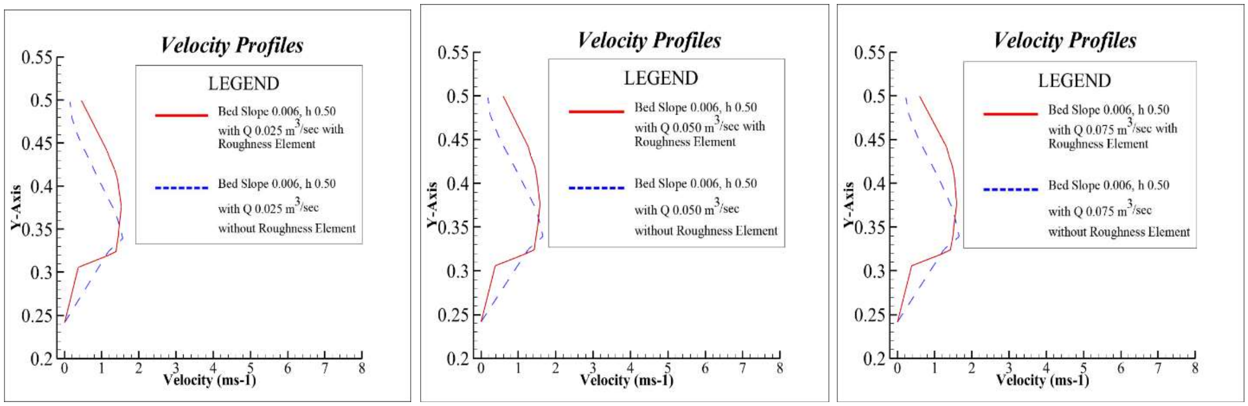
(a) (b) (c)

Figure 7: Vertical Velocity Profiles at Upstream Side (a) 0.025, (b) 0.050, (c) 0.075

The results indicated that when the discharge in the channel was increased the velocity values at the upstream side of the weir also increased. For the same discharge values the channel with roughness elements showed reduced peak values of velocity. There were negligible effects observed in the velocities close to the free surface in both the roughened and non-roughened channels.

3.3.2 Downstream Side

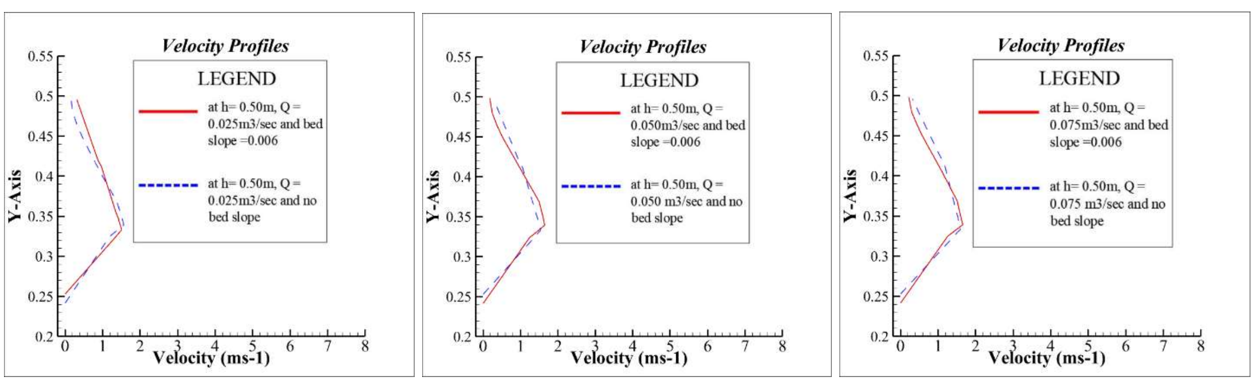
The vertical velocity profiles were also calculated at a distance of 3.64m from the channel inlet on the downstream side of the weir.



(a) (b) (c)

Figure 8: Vertical Velocity Profile at Downstream Side (a) 0.025, (b) 0.050, (c) 0.075

The results as shown in Figure 8 indicated that when the discharge in the channel was increased the velocity values at the downstream side of the weir also increased. It was observed that for the same discharge values the velocity values on the downstream side of the weir were more in comparison to the upstream side of the weir. For the same discharge values the channel with the roughness elements showed reduced peak values of velocity [13]. However, in the upper zones of flow depths the flow velocity was found to be higher in case of the roughened channels as compared to that of the channel without any roughness induced.



(a) (b) (c)

Figure 9: Vertical Velocity Profile at Downstream Side for Different Bed Slopes(a) 0.025, (b) 0.050, (c) 0.075

A comparison of vertical velocity profiles has also been made for bed slopes 0 and 0.006 on the downstream side of the weir as shown in Figure 9. It was observed that for the same discharge value, the peak velocity was approximately 3% for a bed slope 0.006 as compared to the horizontal channel.

3.4 Velocity Magnitude Contours

The velocity magnitude contours for the channel with bed slope 0.006 and induced roughness with the inlet discharge of 25L/sec was measured at 3.5m from the inlet of channel as shown in the Figure 10.

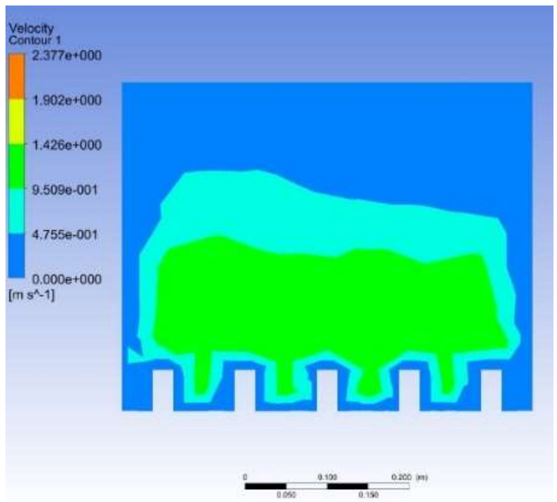


Figure 10: Velocity Contours at the Centre of Weir

It can be seen from the figure that the minimum velocity was observed close to the walls. The peak velocity was observed above the roughness element as there was no obstruction to the flow in this zone.

4 CONCLUSIONS

The study reveals that adding roughness elements reduces the water surface profiles on the downstream side of the weir indicating altered flow dynamics. While upstream water surface profiles remain almost unchanged, there is a significant decrease (approximately 10-12%) in coefficient of discharge for channel with roughness elements. It was also observed that the peak velocities in case of the roughened cases reduces by approximately 11% in comparison to non-roughened channels. The peak velocities are higher in non-roughened channels. Overall, roughness elements play a crucial role in altering flow characteristics, impacting water levels and velocities in channels with weirs.

REFERENCES

1. Mays, L. W. (2019). Water Resources Engineering. Hoboken, NJ: John Wiley & Sons, Inc.
2. Arvanaghi, H., & Azimi, V. (2016). Determination of semicircular-trapezoidal weir discharge coefficient. Agriculture Science Developments, 5(2), 22–27. doi:10.21828/asd-05-02-003 *gallery.* Neural Computing and Applications, 2020. 32(8): p. 3349-3366.

3. Chanson, H., & Montes, J. S. (1998). Overflow characteristics of circular weirs: Effects of inflow conditions. Journal of Irrigation and Drainage Engineering, 124(3), 152–162. doi:10.1061/(asce)0733-9437(1998)124:3(152).
4. Jaiswal, A., & Goel, A. (2019). Aeration through weirs—a critical review. Lecture Notes in Civil Engineering, 187–200. doi:10.1007/978-981-13-6717-5_19.
5. Schmocker, L., Halldórsdóttir, B. R., & Hager, W. H. (2011). Effect of weir face angles on circular-crested weir flow. Journal of Hydraulic Engineering, 137(6), 637–643. doi:10.1061/(asce)hy.1943-7900.0000346.
6. A. H. Chilmeran, T., & M. A. Noori, B. (2005). Characteristics of flow over normal and oblique weirs with semicircular crests. AL-Rafdain Engineering Journal (AREJ), 13(1), 49–61. doi:10.33899/rengj.2005.45568.
7. Schmocker, L., Halldórsdóttir, B. R., & Hager, W. H. (2011a). Effect of weir face angles on circular-crested weir flow. Journal of Hydraulic Engineering, 137(6), 637–643. doi:10.1061/(asce)hy.1943-7900.0000346.
8. Spano, M., & Stara, V. (2008). Application of a 2D turbulence model for estimation of the discharge coefficent on round-shaped weirs. AIP Conference Proceedings. doi: 10.1063/1.2991053
9. Aydin, M. C. (2012). CFD simulation of free-surface flow over triangular labyrinth side weir. Advances in Engineering Software, 45(1), 159–166. doi:10.1016/j.advengsoft.2011.09.006
10. Jak, E.; Hayes, P.C. Procedure of Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. J. Fluids Eng. 2008, 130, 078001.
11. Elsayed, K.; Lacor, C. CFD modeling and multi-objective optimization of cyclone geometry using desirability function, artificial neural networks and genetic algorithms. Appl. Math. Model. 2013, 37, 5680–5704.
12. Bai, R., Zhou, M., Yan, Z., Zhang, J., & Wang, H. (2023). Flow measurement by vegetated weirs: An investigation on discharge coefficient reduction by crest vegetation. Journal of Hydrology, 620, 129552. doi:10.1016/j.jhydrol.2023.129552
13. Shahid Hussain Ali, Usman Ghani, Abid Latif, Naeem Ijaz, Ghufran Ahmed Pasha. An Experimental Study of Turbulent Flow Over a Weir-Like Structure With and Without Vegetation. Life Sci J 2013;10(9s):315-323] (ISSN:1097-8135)