OKÜ Fen Bilimleri Enstitüsü Dergisi
5(Özel Sayı):156-169, 2022

OKU Journal of The Institute of Science and Technology, 5(Special Issue):156-169, 2022

Osmaniye Korkut Ata Üniversitesi Fen Bilimleri Enstitüsü Dergisi

Osmaniye Korkut Ata University Journal of The Institute of Science and Technology

Experimental and Numerical Analysis of Energy Dissipation with Sluice Gate and Weirs in Trapezoidal Channel

Ercan GEMİCİ[1*], Numan KOCAMAN[2], Tuğba VURAL[3], Mert ZÜNGÖR[4]

[1]BartınUniversity, Faculty of Engineering, Architecture and Design, Department of Civil Engineering, 74110, Bartın
[2]Bartın Üniversitesi, Graduate School, Department of Civil Engineering, 74110, Bartın
[3]Bartın Üniversitesi, Graduate School, Department of Civil Engineering, 74110, Bartın
[4]Bartın Üniversitesi, Graduate School, Department of Civil Engineering, 74110, Bartın

[1]https://orcid.org/0000-0001-8464-4281
[2]https://orcid.org/0000-0003-4237-7677
[3]https://orcid.org/0000-0001-8713-3524
[4]https://orcid.org/0000-0002-1105-1511
*Corresponding author: egemici@bartin.edu.tr

Research Article

Article History:
Received: 15.12.2021
Accepted: 25.01.2022
Published online: 23.02.2022

Keywords:
Energy dissipation
Open channel
Velocity
Specific hydraulic head
Sluice gate
Weir

ABSTRACT

In open channels, only one of the sluice gates or weirs is often used for regulation purposes to limit the acceleration of flow through the channel and to help reduce the forces around the water structures. When the sluice gates are used single, depending on the tail water depth and the gate opening, all cases might experience free or submerged flow conditions and hydraulic jump might occur at the downstream. In this case, it is possible to damage the submerged or semi-submerged structures in this area due to the high flow velocity and high hydraulic energy at the downstream of the gate. The combination of weirs and gates can be preferred both to preserve the stability of the river bottom and to prevent damage to structures such as submerged pipes and transmission lines stretching across cross-sectional direction. In this study, the variation of the flow characteristics in the channel and around the pipeline extending across the channel cross-section was investigated when varying cross-sections weirs with the sluice gate are used together. Experimental flow velocity and water levels are measured for different discharges. These measurements were used to determine the boundary conditions of the computational fluid dynamics (CFD) software and to verify the flow property values obtained by CFD. After the verification, the velocity, pressure and specific hydraulic head values of the points that could not be measured experimentally with CFD software were also obtained. As a result of both experimental and numerical analysis, it has been seen that hydraulic head can be reduced significantly by using the gate and weir structures together. The effect of weir geometry on energy loss is clearly seen at high discharges. By using the two structures together, both the water depth in open channels will be kept at the desired level and possible damage due to water forces acting on underwater structures will be prevented.

Trapez Kanallarda Savak Kapağı ve Savaklarla Enerji Azaltımının Deneysel ve Sayısal Analizi

Araştırma Makalesi

Makale Tarihçesi:
Geliş tarihi: 15.12.2021
Kabul tarihi:25.01.2022
Online Yayınlanma: 23.02.2022

Anahtar Kelimeler:
Enerji azaltımı
Açık kanal
Hız

ÖZET

Açık kanallarda kanal boyunca akışın hızlanmasını sınırlamak ve su yapıları etrafındaki kuvvetleri azaltmaya yardımcı olmak için genellikle kapak veya savaklardan yalnız biri regülasyon amacıyla kullanılmaktadır. Savak kapakları yalnız kullanıldığında, kuyruk suyu derinliğine ve kapak açıklığına bağlı olarak serbest veya batmış akış koşulları meydana gelebilir ve mansapta hidrolik sıçrama oluşabilir. Bu durumda, kapağın mansabındaki yüksek akış hızı ve hidrolik enerji nedeniyle bu bölgedeki batık veya yarı batık yapıların zarar görmesi olasıdır. Kapak ve savakların beraber kullanımı,

Özgül hidrolik enerji
Savak kapağı
Savak

hem nehir tabanının stabilitesinin korunması hem de en kesit boyunca uzanan batık borular ve iletim hatları gibi yapılara zarar gelmemesi için tercih edilebilir. Bu çalışmada savak kapağı ile değişen en kesitli savaklar birlikte kullanıldığında kanalın ve kanal en kesiti doğrultusunda uzanan boru hattı etrafındaki akış özelliklerinin değişimi araştırılmıştır. Deneysel akış hızı ve su seviyeleri farklı debiler için ölçülmüştür. Bu ölçümler, hesaplamalı akışkanlar dinamiği (HAD) yazılımının sınır koşullarını belirlemek ve HAD ile elde edilen akış özelliği değerlerini doğrulamak için kullanılmıştır. Doğrulamanın ardından HAD yazılımı ile deneysel olarak ölçülemeyen noktaların hız, basınç ve özgül hidrolik enerji değerleri de elde edilmiştir. Hem deneysel hem de sayısal analizler sonucunda, kapak ve savak yapılarının birlikte kullanılmasıyla hidrolik enerjinin önemli ölçüde azaltılabileceği görülmüştür. Yüksek debilerde savak geometrisinin enerji kaybı üzerindeki etkisi açıkça görülmektedir. İki yapının birlikte kullanılmasıyla hem açık kanallardaki su derinliği istenilen seviyede tutulacak hem de su altı yapılarına etki eden su kuvvetleri nedeniyle olası hasarların önüne geçilecektir.

To Cite: Gemici E., Kocaman N., Vural T., Züngör M. Experimental and Numerical Analysis of Energy Dissipation with Sluice Gate and Weirs in Trapezoidal Channel. Osmaniye Korkut Ata Üniversitesi Fen Bilimleri Enstitüsü Dergisi 2022; 5(Özel sayı): 156-169.

Introduction

Sluice gates that allow water to swell or accumulate are important structures that keep the water level under control and prevent additional energy consumption in the water conveyance. They are frequently used in irrigation of agricultural lands, water intakes, river transport or as a sub-element of various water structures. Sluice gates raise the water level the upstream, which also causes the energy of the water to increase. High hydraulic head can cause various problems during the conveyance of water downstream. Hydraulic jump occurs in this region, the velocity and pressure of the water reach high values. High velocity and pressure forces damage both the river bed and the submerged or semi-submerged water structures within the river. In order to dissipate the energy, stilling pools in the river bed, arrangements in the river cross-section or additional water structures are constructed.

The flow passing under the gate is called free or submerged flow according to the conditions occurring downstream. The flow where only the water height at the upstream of the gate is effective and free jumping occurs at the downstream is free flow. Flows where the downstream water level affects the upstream water level and the downstream water level is higher than the gate opening are expressed as submerged flow. In the literature, there are many studies examining the flow characteristics for free flow (Silva and Rijo, 2017; Ferro, 2018; Petrila, 2002; Hoseini and Vatankhah, 2021; Dou et al., 2020) and submerged flow (Vaheddoost et al., 2021; Bijankhan et al., 2017; Sauida, 2014; Shayan and Farhoudi, 2013; Shaddehi and Bijankhan, 2020) after the gate and investigating the flow around the weir in various geometries (Haghiabi et al., 2021; Hu et al., 2018; Zounemat-Kermani and Mahdavi-Meymand, 2019; Zhang et al., 2018; Tullis, 2011). However, the number of studies is limited for situations where the gate and weir structure are used sequentially in channels that are not rectangular.

Abdelmonem et al. (2018) carried out an experimental study to increase the energy loss downstream of the sluice gate and to shorten the hydraulic jump length. Different gate openings, different discharges and different pendulum sill positions were used in the experiments using a 3 cm diameter pendulum sill behind the gate. It was stated that the pendulum should be placed in the first half of the hydraulic jump length to increase the energy loss.

Tan et al. (2008) examined the water level and discharge changes in unstable flows for the double-row movable sluice gate case in the form of upstream and downstream gate, and reported that the flow would be more stable if the water level in front of the gate was kept constant instead of the water level behind the gate. Liu et al. (2015) stated that the mechanical energy loss continues continuously by decreasing over time due to continuous scour in the river bed due to the under gate flow, but the discharge change passing under the gate is very small. Habibzadeh et al. (2011) who were proposed a theoretical model for the discharge coefficient in free and submerged rectangular gates have been stated that the formation of a circulating zone in the upstream pool causes turbulence and energy loss. Also they have been particularized that the energy loss is higher in submerged conditions, because the flow boundary is subject to very high shear layer drag in submerged gates. Cassan and Belaud (2012) investigated the turbulent flow near the gate structure on a smooth and rigid bed experimentally and numerically, and especially focused on submergence and large opening of gate. Erdbrink et al. (2014) investigated the mean pressure near the gate and turbulent kinetic energy with numerical simulations.

Weir characteristics have an important place in the flow pattern (Karimi et al., 2018). There are many studies investigating broad-crested weir flow with rectangular, triangular, trapezoidal, ogee, labyrinth, piano key, etc. cross-sections. Denys and Basson (2020) investigated the hydrodynamic behavior of rectangular piano key weirs. Al-Khatib and Gogus (2014) created a discharge estimation model with rectangular and compound cross-section broad-crested weirs. Imanian et al. (2021) investigated the free surface flow over a broad-crested weir under different hydraulic loads. Experimental part of their study has been consisted of the 3D velocity profile with Acoustic Doppler Velocimeter (ADV) around a rectangular weir placed in a 5 m long channel. In addition, the area on the weir was solved numerically with the OpenFOAM, which is open source CFD toolbox, using standard k-ε, RNG k-ε, realizable k-ε, k-ω SST and LRR turbulence models, and the numerical results have been compared with the experimental data.

In this study, it is aimed to reduce the high energy of the water transferred from upstream of the sluice gate to the downstream with different weir sections. Velocity measurements around the pipe were conducted with the ADV device for different discharges in the trapezoidal channel. Experimentally obtained data were also used to validate the numerical analyzes. Flow-3D commercial software was used as a numerical analysis model. The variation of velocity, pressure, specific hydraulic head, turbulent kinetic energy, shear velocity and shear stresses around the weir and pipe under different conditions were investigated.

Material and Methods

Experimental Setup

Laboratory experiments were carried out in a trapezoidal open channel setup in the Bartın University Civil Engineering Hydromechanics Laboratory. There is a thin, vertically movable sluice gate at the beginning of the channel. At the upstream of this sluice gate, the channel length is 5m, the bottom width is 0.5m, the height is 0.5m and the slope of the channel is 1:1. longitudinal bottom slope of the channel is 0.003. The experimental setup is shown in Figure 1.

(a) (b) (c)

Figure 1. Experimental Setup (a) no weir (b) V-weir (c) E-weir

Thickness of the rigid vertical sluice gate is 2.5 mm and the gate opening can be adjusted manually. Water circulation in the channel is achieved by the centrifugal pump that can supply water up to 108 L/s. Discharge can be adjusted with the valve on the pipe that conveys water to the high reservoir. The desired discharge in the channel can be controlled with ultrasonic and electromagnetic flow meters on the same pipe. In the experiments, the water surface profile was determined and velocities were measured with ADV.The ADV measures the velocity at a rate of approximately 10 Hz, averages the data, and records 1 s velocity-vector data. The acoustic Doppler velocimetry system provided instantaneous values of the two velocity components.It was oriented with the xy-plane being horizontal, the x direction aligned with the flow direction and positive downstream upwards.The point velocities were found by dividing the cross-section into pieces with 5 cm distances and making vertical velocity measurements at these points.Initially, the experiments were carried out under the conditions of a constant 3 cm sluice gate opening, at 30 and 66 m^3/h discharges, submerged flow where there are no weirs.Then, two weirs with different cross-sections for the same conditions were placed 2m downstream of the sluice gate.

Numerical Model

Numerical simulation of flow pattern was carried out using Flow-3D commercial software, which makes 3D flow modeling. There are different turbulence closure models in the software that applies the volume of fluid (VOF) method in solving problems with free surfaces. Flow-3D uses Reynolds equations derived from Navier-Stokes equations. These models are solved for the time-averaged turbulent flow field. Time-averaged turbulence quantities are mean velocities, turbulent kinetic energy, and Reynolds stress terms. According to the Boussinesq hypothesis, Reynolds stresses can be expressed in terms of mean velocity gradients$-\rho\overline{u_i'}\overline{u_j'}$in Reynolds-averagedNavier-Stokes (RANS) equations,

$$-\rho\overline{u_i'}\overline{u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_i}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij} \tag{1}$$

Among Reynolds stress models, the two-equation eddy-viscosity equations usually solve the two additional transport equations and calculate the turbulent viscosity μ_t as a function of k (turbulent kinetic energy) and ε (turbulent dissipation rate) or k and ω (specific dissipation rate). In the standard k-ϵ model, the turbulent viscosity μ_t is:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2}$$

where ρ is the density of the fluid and C_μ is constant. The standard k-ε model does not take into account effects of streamline curvature.

RNG k-ε model brings improvements over the k-ε model with statistical techniques. This model also considers eddy and rotational effects by changing the turbulent viscosity. This model also considers eddy and rotational effects by changing the turbulent viscosity. In this study, the RNG k-ε model, which generally performs better in similar studies, was used (Ran et al., 2018).

The dimensions and flow conditions of the numerical model were constituted the same as the laboratory model. The three-dimensional geometries of the channel, gate and weirs were created using Salome Platform. Mesh of rectangular cells was used in the Flow-3D software. Meshes were checked for flow domain by the Flow-3D FAVOR application method that generates grids. Two mesh blocks were created to include the all channel volume and pipe volume. The size of the channel volume grid was first set to 0.05 – 0.015 m long (dense in the gate area), 0.01 m wide and 0.01 m high, and the number of all the grids was 6050000. The mesh surrounding the pipe consists of 46080 cells, each with a side of 0.0025 m. As the initial condition, 0.4 m height of water before the valve and 30 m^3/h and 66 m^3/h discharges were defined. Volume flow rate was taken at the upstream end and the outflow was selected at the downstream end as the boundary condition. Boundary condition at the channel side walls and the bottom was defined asawall. No-slip boundary condition was applied on solid boundary surfaces. At the upper boundary of the solution region and at the pipe boundary region, symmetry was taken as the boundary condition.

Results and Discussion

Model Validation

Verification of both experimental measurements and established model is necessary for a realistic analysis. For this reason, firstly, the accuracy of velocity measurements was calibrated by using the discharges obtained from two flow meters and the velocity values measured by ADV. The discharges obtained from the flow meter were entered into the model as boundary conditions. Mesh structure, cell size and number of cells should be determined carefully to obtain accurate results. Therefore, considering the analysis time and experimental data, the optimum mesh conditions for which the model is acceptable were found by gradually improving it. For the 49 points shown in Figure 2-4, water depth differences which measured and obtained from the model were found in the range of 3,00% - 8,95% under 6 different conditions. As shown in Table 1 and Table 2, when the x-direction velocities obtained from the experimental and model are compared for a discharge of 30 and 66 m^3/h, the difference is less than 7.4%.

Table 1. Comparison of horizontal velocities obtained by the model and experiment (y=0.25m and Q=30 m^3/h)

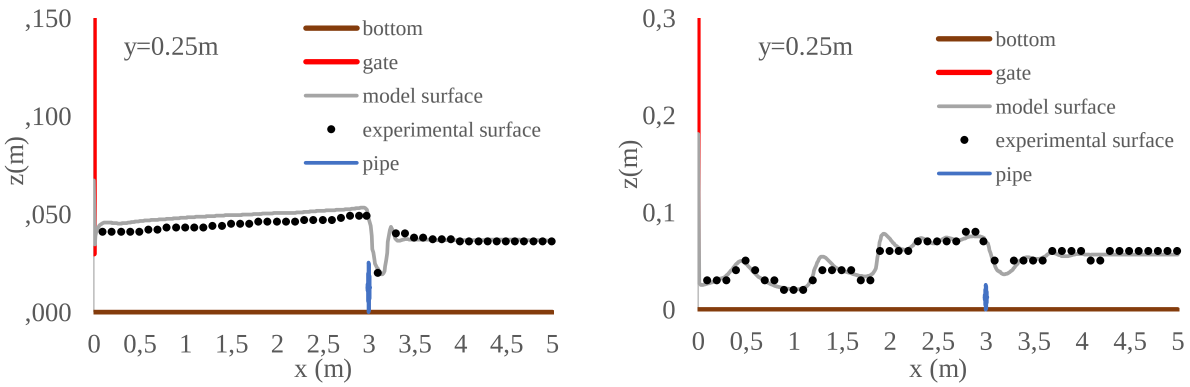
	No weir			**E-weir**			**V-weir**		
x (m)	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**
-0.20	0.113	0.120	6.18	0.047	0.050	7.09	0.030	0.033	8.44
0.20	0.406	0.427	4.98	0.247	0.259	4.52	0.229	0.244	5.97
0.60	0.378	0.394	4.01	0.336	0.353	4.82	0.294	0.315	6.53
1.00	0.353	0.369	4.35	0.256	0.268	4.51	0.247	0.258	4.08
1.40	0.345	0.351	1.69	0.179	0.184	2.59	0.170	0.174	2.24
1.80	0.338	0.348	2.76	0.151	0.156	3.70	0.136	0.142	3.99
2.30	0.343	0.366	6.32	0.279	0.292	4.55	1.479	1.524	2.94
2.80	0.301	0.313	3.76	0.589	0.613	3.90	0.203	0.192	5.70
		Average:	**4.26**		**Average:**	**4.46**		**Average:**	**4.99**

Table 2. Comparison of horizontal velocities obtained by the model and experiment (y=0.25m and Q=66 m^3/h)

	No weir			**E-weir**			**V-weir**		
x (m)	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**	**u_{model} (m/s)**	**u_{exp} (m/s)**	**ε (%)**
-0.20	0.102	0.108	5.21	0.039	0.045	13.39	0.034	0.039	12.60
0.20	1.297	1.313	1.24	0.477	0.504	5.28	0.414	0.447	7.38
0.60	1.206	1.235	2.31	0.585	0.614	4.73	0.517	0.533	2.91
1.00	1.208	1.251	3.42	0.505	0.548	7.80	0.498	0.507	1.77
1.40	1.029	1.108	7.15	0.324	0.346	6.30	0.315	0.341	7.51
1.80	1.104	1.135	2.72	0.239	0.251	4.74	0.230	0.256	10.34
2.30	0.558	0.619	9.86	0.217	0.247	12.05	1.310	1.372	4.52
2.80	0.514	0.572	10.17	1.299	1.364	4.73	1.425	1.463	2.60
		Average:	**5.26**		**Average:**	**7.38**		**Average:**	**6.20**

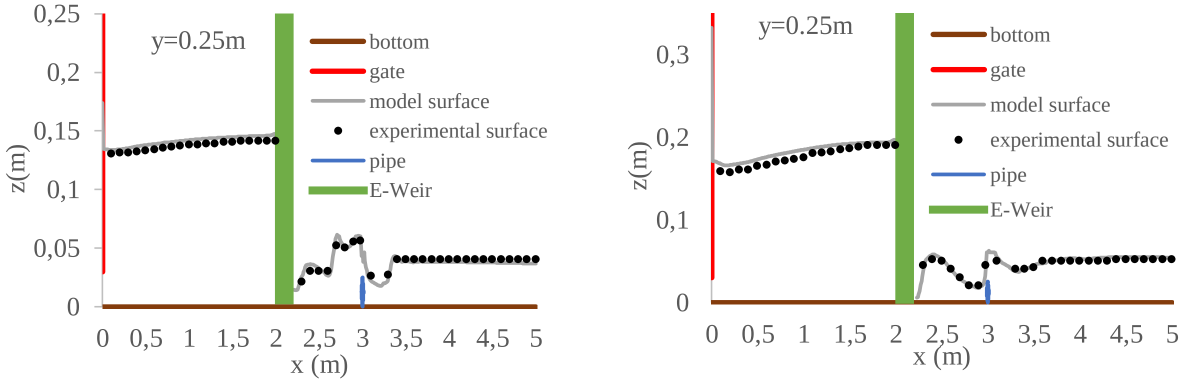
Water Surface Profiles

Flow pattern is described using water surface profiles measured along the centerline of the open channel. The water surface profile is used to determine the flow type and hydraulic jump characteristic. It is seen that there is a hydraulic jump after the sluice gate in Figure 2b and after the weirs in Figure 3-4. In both discharges conditions, both weirs are free flowing. M1 profile is formed between the sluice gate and the weirs for all discharges.



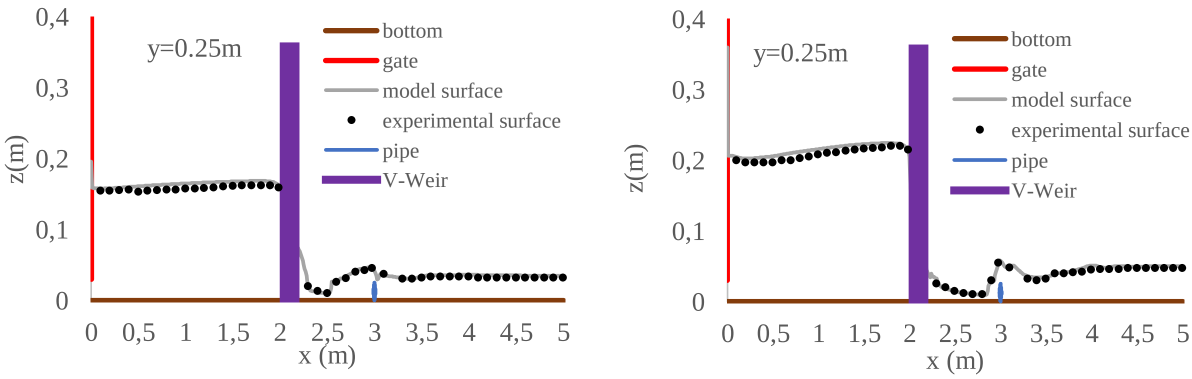
(a) (b)

Figure 2. Comparison of the water surface profile for no weir (a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

Figure 3. Comparison of the water surface profile for E-weir (a) Q=30 m^3/h (b) Q=66 m^3/h

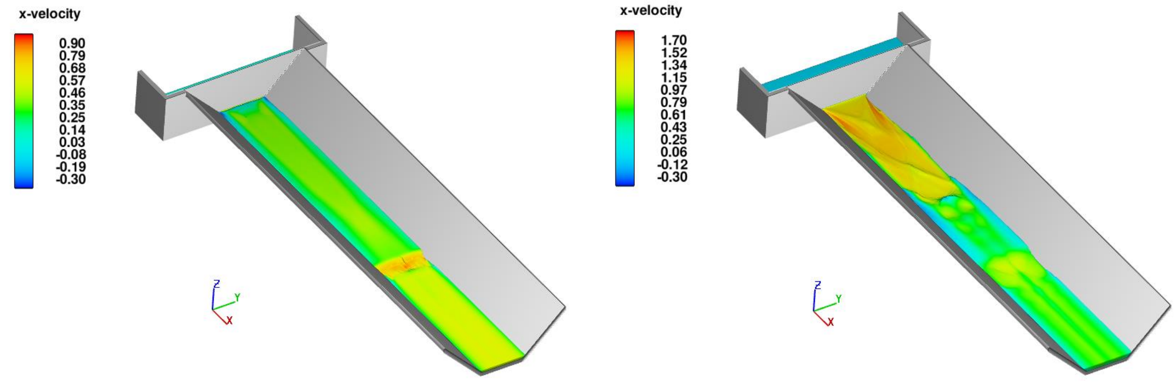


(a) (b)

Figure 4. Comparison of the water surface profile for V-weir (a) Q=30 m^3/h (b) Q=66 m^3/h

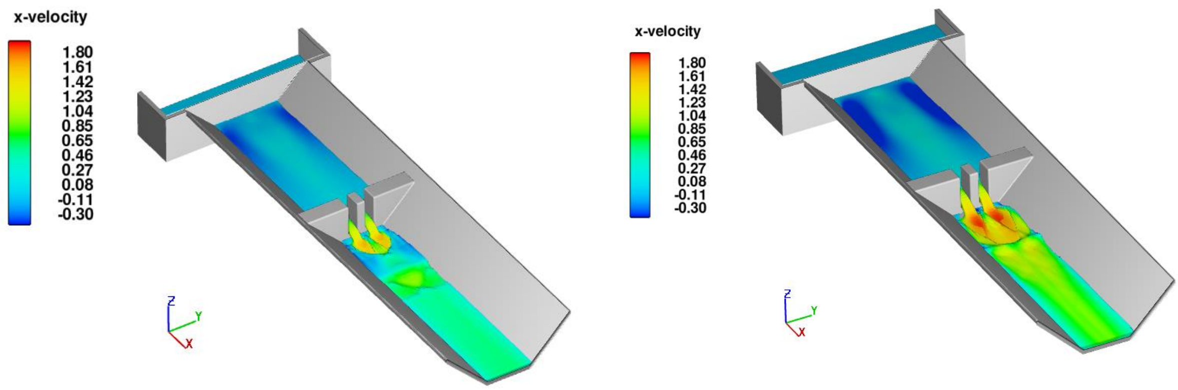
Velocity Distribution

Determination of velocity distribution is essential to modeling of hydraulic processes in open channel. The velocity distribution in open channel is affected by channel geometry, bed characteristics and in-river water structures. Water forces erode the channel bottom and can directly affect the water structure and damage it. It is seen in Figure 5b that the flow velocity after the gate is considerable amount high at major discharge and this high velocity continues until the middle of the channel. After the weir, it is seen in Figure 6-7 that the high velocity is faded out earlier.



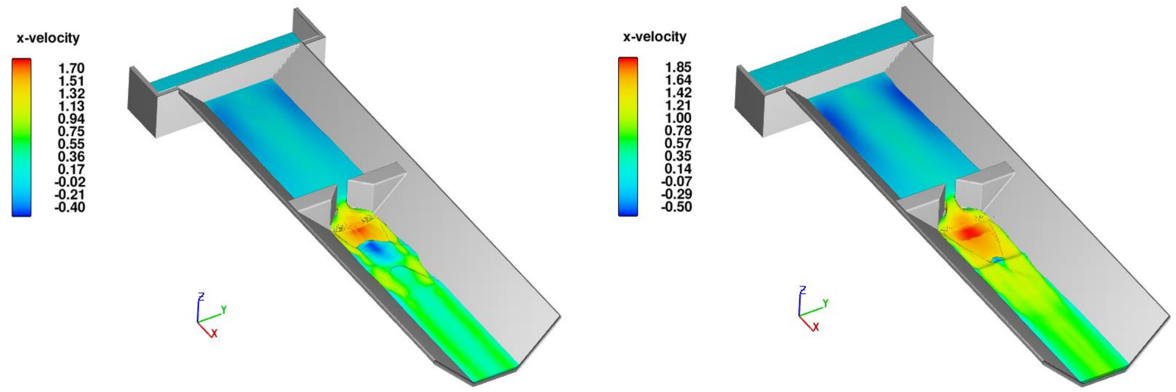
(a) (b)

Figure 5. 3D distribution of x velocity for no weir(a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

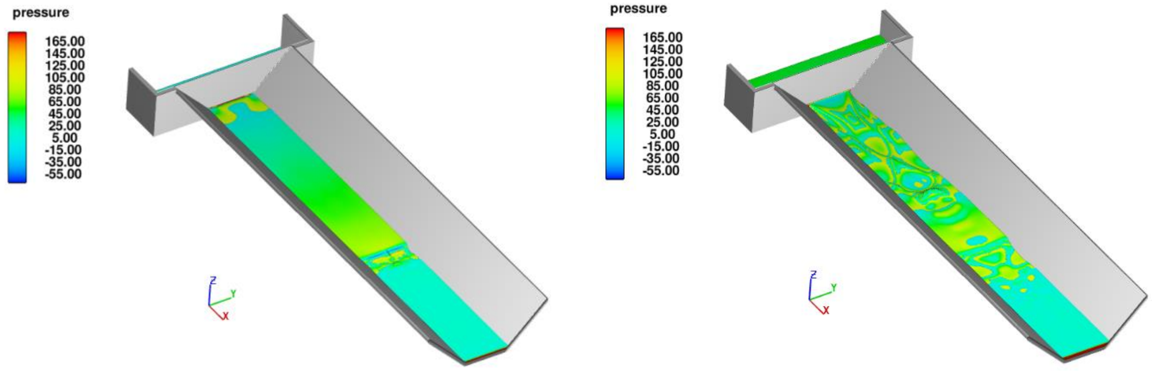
Figure 6. 3D distribution of x velocity for E-weir(a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

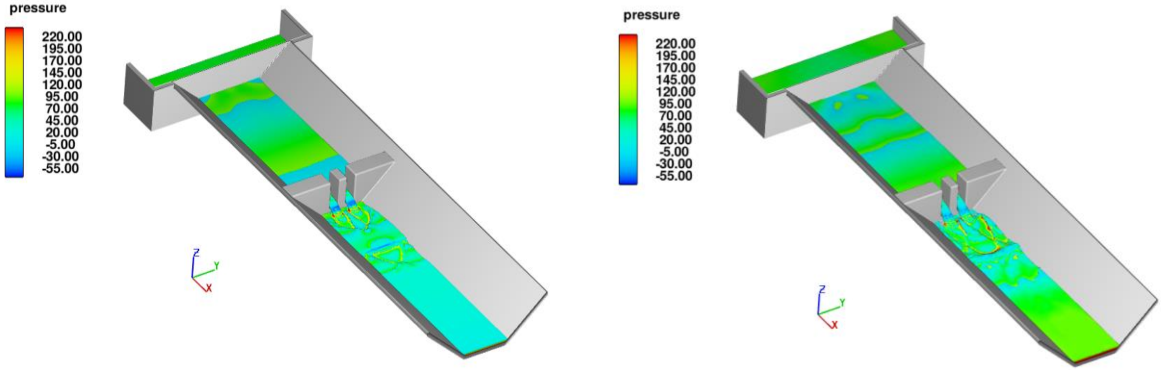
Figure 7.3D distribution of x velocity for V-weir (a) Q=30 m^3/h (b) Q=66 m^3/h

Pressure force on the water structure is the most important parameter in the design of the structure. In this study, the focus is on the pressure distribution in front of the pipe, which placed 3 meters after the sluice gate, at the downstream of gate. In Figure 8b and Figure 9b are compared, it can be seen that the E-weir has decreased the pressure at the end regions of the pipe for high discharge. On the other hand, comparing Figure 8 and Figure 10, it is seen in that the V-weir has reduced the pressure in the middle regions of the pipe for both discharges.



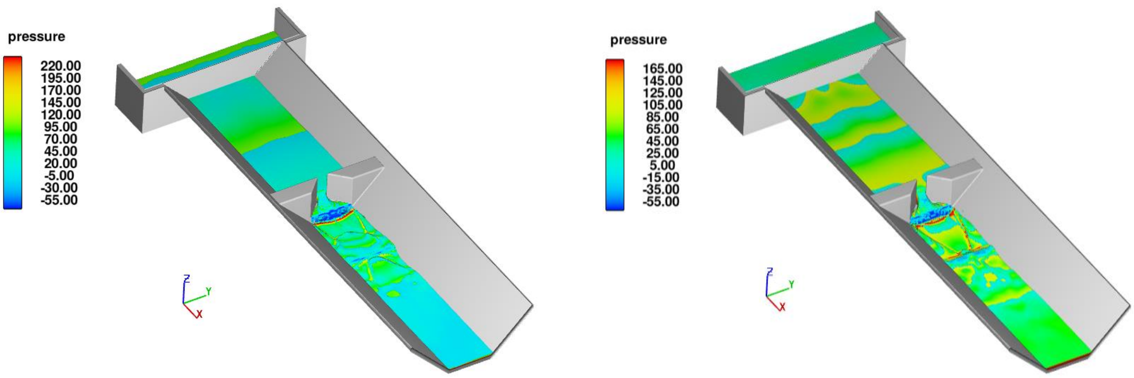
(a) (b)

Figure 8.3D distribution of pressure for no weir(a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

Figure 9. 3D distribution of pressure for E-weir(a) Q=30 m^3/h (b) Q=66 m^3/h



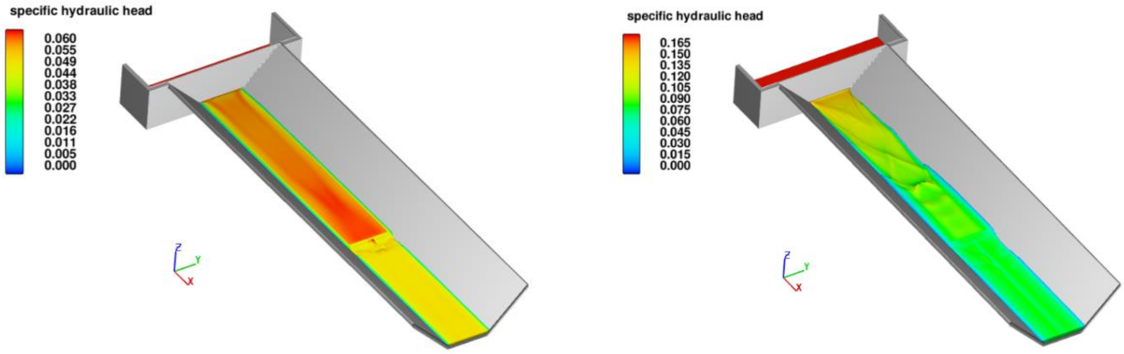
(a) (b)

Figure 10. 3D distribution of pressurefor V-weir(a) Q=30 m^3/h (b) Q=66 m^3/h

Specific Hydraulic Head Distribution

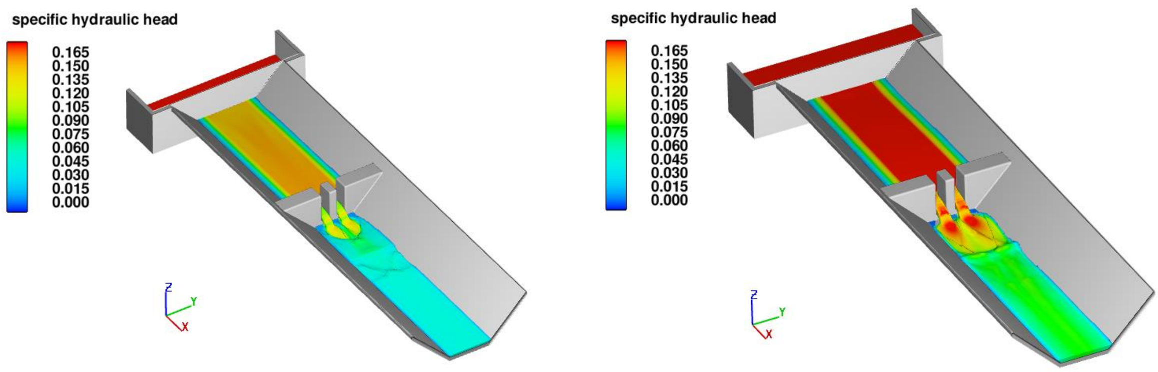
Forces on the structure are proportional to the magnitude of hydraulic head of the water. In open channels, specific hydraulic head is usually used instead of hydraulic head. Specific head was calculated by using Equation 3. D is water level, v is velocity and $g$is gravitational acceleration in Equation 3.

$$H_{specific} = D + \frac{v^2}{2g} \tag{3}$$



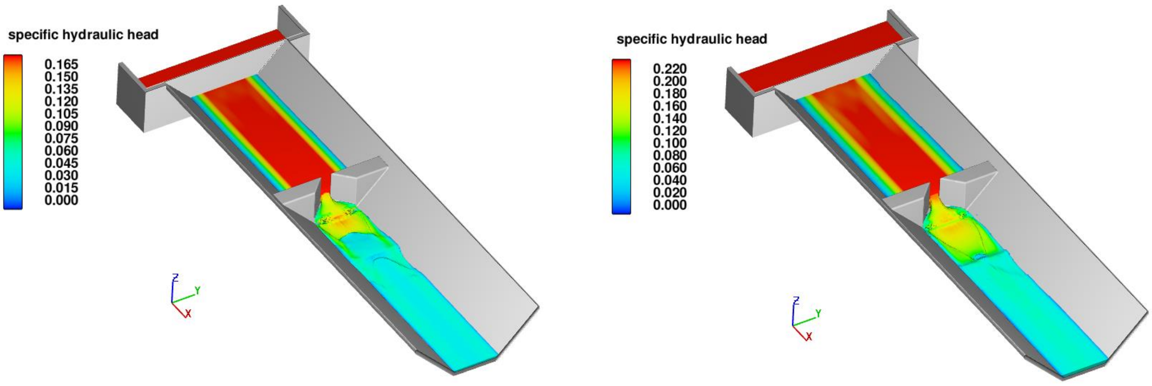
(a) (b)

Figure 11. 3D distribution of specific hydraulic head for no weir (a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

Figure 12.3D distribution of specific hydraulic head for E-weir (a) Q=30 m^3/h (b) Q=66 m^3/h



(a) (b)

Figure 13. 3D distribution of specific hydraulic headfor V-weir(a) Q=30 m^3/h (b) Q=66 m^3/h

Table 3. Variation of specific hydraulic head in front of the pipe (x=2.98m)

y (m)	**Q=30 m^3/s**					**Q=66 m^3/s**				
	Specific Hydraulic Head			**Energy Dissipation**		**Specific Hydraulic Head**			**Energy Dissipation**	
	No weir	**V-weir**	**E-weir**	**V-weir ε (%)**	**E-weir ε (%)**	**No weir**	**V-weir**	**E-weir**	**V-weir ε (%)**	**E-weir ε (%)**
0.00	0.0517	0.0620	0.0502	19.73	-2.90	0.0745	0.0680	0.0710	-8.73	-4.60
0.05	0.0551	0.0721	0.0554	30.90	0.54	0.0809	0.0783	0.0862	-3.28	6.46
0.10	0.0559	0.0616	0.0609	10.13	8.86	0.0852	0.0802	0.0904	-5.89	6.09
0.15	0.0562	0.0534	0.0659	-4.97	17.32	0.0868	0.0845	0.0836	-2.59	-3.70
0.20	0.0562	0.0491	0.0670	-12.70	19.13	0.0866	0.0747	0.0655	-13.78	-24.34
0.25	0.0570	0.0472	0.0678	-17.20	19.07	0.0860	0.0570	0.0583	-33.76	-32.25
0.30	0.0564	0.0492	0.0698	-12.77	23.73	0.0863	0.0752	0.0613	-12.85	-29.01
0.35	0.0561	0.0533	0.0657	-5.07	16.99	0.0865	0.0863	0.0895	-0.24	3.42
0.40	0.0558	0.0623	0.0603	11.62	8.05	0.0851	0.0833	0.0905	-2.11	6.32
0.45	0.0548	0.0716	0.0552	30.58	0.69	0.0809	0.0803	0.0828	-0.69	2.36
0.50	0.0514	0.0612	0.0499	19.12	-2.88	0.0745	0.0684	0.0663	-8.29	-11.07
			Average:	**6.31**	**9.87**			**Average:**	**-8.38**	**-7.30**

As can be seen from Table 3 and Figure 11-13 for high discharge values, specific hydraulic head decreases due to the effect of weirs, especially in the parts close to the middle of the pipe.

Conclusion

In this study, it is aimed to reduce hydraulic head around the downstream pipe by using different weirs with the effect of under sluice gate flow. The model errors slightly increase with the effect of secondary flow in the upstream of sluice gate, especially at high discharges. The average difference between experimental and numerical horizontal velocity values for all six situations at the velocity measurement points is below 7.4%. This shows that numerical models can be used for such problems. V-type weirs are more effective than E-type weirs in reducing velocity and pressure. Based on numerical results, for discharge of 66 m^3/h, the specific hydraulic head in front of the pipe was reduced with the effect of V-weir and E-weir by 8.38% and 7.3% on average, respectively. Weirs with discharge of 30 m^3/h were not effective enough. The reason for this is that after the weirs at low discharge, the energy has increased because the water accumulates in front of the pipe. However, for cases where energy reduction is required only in the middle region, V-type weirs, which reduce the energy up to 17%, can be chosen. E-weirs and V-weirs can be preferred to shorten the hydraulic jump distance at high discharges.

Statement of Conflict of Interest

Authors have declared no conflict of interest.

Author's Contributions

The contribution of the corresponding author is 40% and contribution of other authors is equal and 20%.

*This study was presented as a summary paper at the International Conference on Engineering, Natural and Applied Sciences (ICENAS'21) held online on 24-26 November 2021.

References

Abdelmonem YK., Shabayek S., Khairy AO. Energy dissipation downstream sluice gate using a pendulum sill. Alexandria Engineering Journal 2018; 57(4): 3977-3983.

Al-Khatib, IA., Gogus M. Prediction models for discharge estimation in rectangular compound broad-crested weirs. Flow Measurement and Instrumentation 2014; 36, 1-8.

Bijankhan M., Kouchakzadeh S., Belaud G. Application of the submerged experimental velocity profiles for the sluice gate's stage-discharge relationship. Flow Measurement and Instrumentation 2017; 54, 97-108.

Cassan L., Belaud G. Experimental and numerical investigation of flow under sluice gates. Journal of Hydraulic Engineering 2012; 138(4): 367-373.

Denys FJ., Basson GR. Unsteady hydrodynamic behavior at piano key weirs. Journal of Hydraulic Engineering 2020; 146(5): 04020028.

Dou M., Qin C., Li G., Wang C. Research on calculation method of free flow discharge based on artificial neural network and regression analysis. Flow Measurement and Instrumentation 2020; 72, 101707.

Erdbrink CD., Krzhizhanovskaya VV., Sloot PM. Free-surface flow simulations for discharge-based operation of hydraulic structure gates. Journal of Hydroinformatics 2014; 16(1): 189-206.

Ferro V. Testing the stage-discharge relationship of a sharp crested sluice gate deduced by the momentum equation for a free-flow condition. Flow Measurement and Instrumentation 2018; 63, 14-17.

Habibzadeh A., Vatankhah AR., Rajaratnam N. Role of energy loss on discharge characteristics of sluice gates. Journal of Hydraulic Engineering 2011; 137(9): 1079-1084.

Haghiabi AH., Nou MRG., Parsaie A. The energy dissipation of flow over the labyrinth weirs. Alexandria Engineering Journal 2021; In Press.

Hoseini P., Vatankhah AR. Stage-discharge relationship for slide gates installed in partially full pipes. Flow Measurement and Instrumentation 2021; 77, 101838.

Hu H., Qian Z., Yang W., Hou D., Du L. Numerical study of characteristics and discharge capacity of piano key weirs. Flow Measurement and Instrumentation 2018; 62, 27-32.

Imanian H., Mohammadian A., Hoshyar P. Experimental and numerical study of flow over a broad-crested weir under different hydraulic head ratios. Flow Measurement and Instrumentation 2021; 80, 102004.

Karimi M., Attari J., Saneie M., Jalili Ghazizadeh MR. Side weir flow characteristics: comparison of piano key, labyrinth, and linear types. Journal of Hydraulic Engineering 2018; 144(12): 04018075.

Liu SH., Liao TT., Luo QS. Numerical simulation of turbulent flow behind sluice gate under submerged discharge conditions. Journal of Hydrodynamics 2015; 27(2): 257-263.

Petrila T. Mathematical model for the free surface flow under a sluice gate. Applied Mathematics and Computation 2002; 125(1): 49-58.

Ran D., Wang W., Hu X. Three-dimensional numerical simulation of flow in trapezoidal cutthroat flumes based on FLOW-3D. Frontiers of Agricultural Science and Engineering 2018; 5(2): 168-176.

Sauida MF. Calibration of submerged multi-sluice gates. Alexandria Engineering Journal 2014; 53(3): 663-668.

Shaddehi FR., Bijankhan M. Experimental study on free and submerged multi-jets. Flow Measurement and Instrumentation 2020; 75, 101805.

Shayan HK., Farhoudi J. Effective parameters for calculating discharge coefficient of sluice gates. Flow Measurement and Instrumentation 2013; 33, 96-105.

Silva CO., Rijo M. Flow rate measurements under sluice gates. Journal of Irrigation and Drainage Engineering 2017; 143(6): 06017001.

Tan GM., Ding ZL., Wang CD., Yao X. Gate regulation speed and transition process of unsteady flow in channel. Journal of Hydrodynamics 2008; 20(2): 231-238.

Tullis BP. Behavior of submerged ogee crest weir discharge coefficients. Journal of Irrigation and Drainage Engineering 2011; 137(10): 677-681.

Vaheddoost B., Safari MJS., Zeynali RI. Discharge coefficient for vertical sluice gate under submerged condition using contraction and energy loss coefficients. Flow Measurement and Instrumentation 2021; 80, 102007.

Zhang J., Chang Q., Zhang QH., Li SN. Experimental study on discharge coefficient of a gear-shaped weir. Water Science and Engineering 2018; 11(3): 258-264.

Zounemat-Kermani M., Mahdavi-Meymand A. Hybrid meta-heuristics artificial intelligence models in simulating discharge passing the piano key weirs. Journal of Hydrology 2019; 569, 12-21.