

JOURNAL OF SCIENCE



SAKARYA UNIVERSITY

Sakarya University Journal of Science

ISSN 1301-4048 | e-ISSN 2147-835X | Period Bimonthly | Founded: 1997 | Publisher Sakarya University |
<http://www.saujs.sakarya.edu.tr/>

Title: Numerical and Experimental Modeling of Flow Over a Broad-Crested Weir

Authors: Serdar Korkmaz, Nasir Ahmad Ghaznawi

Received: 2018-05-15 00:00:00

Revised: 2018-08-18 12:00:00

Accepted: 2018-11-22 00:00:00

Article Type: Research Article

Volume: 23

Issue: 2

Month: April

Year: 2019

Pages: 252-258

How to cite

Serdar Korkmaz, Nasir Ahmad Ghaznawi; (2019), Numerical And Experimental Modeling of Flow Over a Broad-Crested Weir. Sakarya University Journal of Science, 23(2), 252-258, DOI: 10.16984/saufenbilder.423705

Access link

<http://www.saujs.sakarya.edu.tr/issue/39539/423705>

New submission to SAUJS

<http://dergipark.gov.tr/journal/1115/submission/start>

NUMERICAL AND EXPERIMENTAL MODELING OF FLOW OVER A BROAD-CRESTED WEIR

Serdar Korkmaz^{*1}, Nasir Ahmad Ghaznawi¹

Abstract

Broad-crested weirs are hydraulic structures installed in open-channels used to control the flow of water and measure the discharge. In this study, the flow over a broad-crested weir in open-channel was experimentally and numerically modeled. Experiments were carried out in a channel with a length of 1.5 m and a width of 0.055 m under five different flow rates. In each experiment, water depth and flow velocity in flow direction were measured using a pitot tube. Numerical model was developed using SimFlow software. The velocities and pressure fields were computed numerically. The water depths and velocities in flow direction obtained from the numerical model were compared to experimental values. There is generally a good agreement between experimental and numerical results. Relatively small errors in velocity profiles are observed in higher discharges.

Keywords: broad-crested weir, computational fluid dynamics, SimFlow

1. INTRODUCTION

Discharge measurement is a significant subject in hydraulic engineering. A weir is a device for measuring flow in both man-made open channels and natural streams. Broad-crested weirs are defined as structures where the crest of the weir is horizontal and the streamlines are parallel to each other over the weir (Fig. 1) [1, 2]. Depending on the ratio of upstream water depth to weir length, these broad-crested weirs are divided into four groups as long-crested weirs, broad-crested weirs, narrow-crested weirs and sharp-crested weirs [3].

With reference to Fig.1, H/L ratio should be between 0.08 and 0.5 in broad-crested weirs [4]. They are used as discharge measurement structures, for example in irrigation channels, wastewater systems, rainwater, hydropower structures and laboratory flumes. These structures have complex flow properties and for this reason pressure field, velocity field and flow profile should be well estimated so that they can be well designed. The analysis of the flows interacting with the broad-crested weir structure can be observed by physical model experiments. Various experimental studies have been conducted on this topic [1, 2, 5, 6, 7, 9, 10]. However, it is difficult

* Corresponding Author

¹ Department of Civil Engineering, Uludağ University, Bursa, Turkey

to obtain the analytical solutions to the equations of motion of the water due to the presence of turbulence. Thus, new developments in computer science and numerical techniques have arisen the use of Computational Fluid Dynamics as a strong tool for this aim [1, 5]. In one of the studies, the flow over four different broad-crested weirs were investigated under five different steady flow rates in a channel with a base slope of $S = 0.001$ and the experimental results were compared to those obtained from the numerical model FLOW-3D [5]. In another study, the flow through a circular weir were studied. In the experimental study, eight circular weirs and five flow conditions were determined. They concluded that the discharge measurements were affected by the upstream conditions and that the cylinder size and the weir height had no effect on the discharge coefficient [6]. Some researchers studied the flow over a broad-crested weir with trapezoidal cross-section. Experimentally, the water surface profile was determined for different flow conditions. The numerical model with the same experimental subject was calculated by the ANSYS-Fluent package program based on the finite volume method. They found that the standard $k-\epsilon$ model, although with a small margin, was more successful than the other models according to mean square error values [7]. Another study experimentally studied the flow characteristics passing through a combined broad-crested weir and box type culvert. Effects of parameters on the performance of the structure such as upstream water depth, interior geometry of the culvert, the culvert length, the culvert inner radius, the height of the weir crest, the width of the weir and the slope of the weir were investigated. The aim was to find the coefficient of the flow and to obtain equations that predict the flow in the structure [8]. In the present study, the flow over a broad-crested weir in an open-channel is both experimentally and numerically modeled. In the laboratory experiments, water depth and velocities in flow direction were measured under five different flow rates. Numerical model was developed by using SimFlow package program. In numerical calculations the realizable $k-\epsilon$ turbulence model was used and the free water surface profile was determined by the Volume of Fluid (VOF) method. The solutions were realized in three dimensions. The free surface and velocity

distributions of flow obtained numerically are compared with the values measured experimentally.

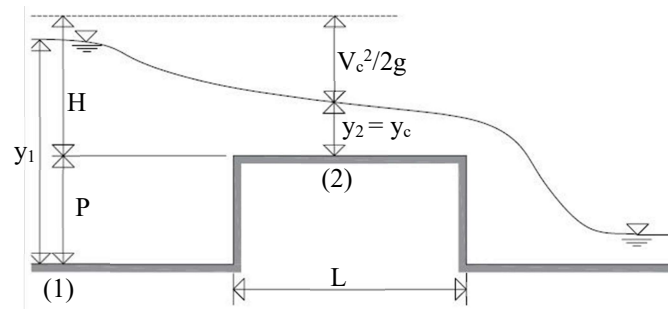


Figure 1. Geometry of a broad-crested weir

2. METHODOLOGY

2.1. Experimental Study

Experimental study was conducted in the hydraulics laboratory of Civil Engineering Department of Uludağ University. The flume has a length of 1.5 m and a width of 0.055 m. The bottom of the flume is of coated steel and side walls are of transparent acrylic (Fig. 2). The dimensions of the broad-crested weir are 0.055 m in width, 0.028 m in height and 0.070 m in length. Five different flow rates were used in those experiments. Velocity measurements were carried out using a pitot tube (Fig. 3) within different sections of the flow. As it provides direct measurement of the velocity head, the error induced is only limited to the capillary effects which is estimated to be ± 0.1 m/s. The OAG 100 B hydraulic bench was used for the control of flow rate. The water was pumped from the bottom water reservoir (sump) in order to convey the water into the open channel, and it outflows from the channel into an open reservoir just above the sump for flow rate measurements and the cycle of water was maintained in this way. The pump capacity is 0.80 L/s which is the highest discharge that can be used in experiments. A stopwatch and the open reservoir of hydraulic bench were used to measure the flow rate. Flow rate is also shown on a digital display. The weir was located at 0.485 m from the beginning of the channel. Velocity measurements were taken at specified sections along the channel and at different depths. In order to correctly obtain

the velocity variation, the measurements were taken at more closely spaced intervals above the weir than upstream and downstream sections of the channel.



Figure 2. Experimental channel



Figure 3. Pitot tube made from a ruler and a medical infusion tube.

2.2. Numerical Modeling with SimFlow

SimFlow software was used as the numerical model in this study. SimFlow is a powerful general purpose CFD (Computational Fluid Dynamics) software and it is an open source interface of OpenFOAM. It can solve all kinds of fluid mechanics problems based on the general laws of continuity, momentum and energy. For example; compressible and incompressible fluid flows, turbulent flows, heat transfer, multiphase flows, cavitation and chemical reactions. In Simflow, as turbulence modeling options Laminar, RANS ($k-\varepsilon$, RNG $k-\varepsilon$, Realizable $k-\varepsilon$, Spalart-Allmaras, $k-\omega$, $k-\omega$ SST and $k-k1-\omega$) and LES models exist. In this study, Realizable $k-\varepsilon$ turbulence closure model was employed because it shows better performance in flows involving rotation, and hboundary layers with strong adverse pressure gradients. In this model, k represents the kinetic

energy of the turbulence and ε represents the dissipation rate. The realizable $k-\varepsilon$ model has been developed for high Reynolds number and suitable for turbulent flows [8, 9]. Furthermore, it uses a wall function in near-wall region and therefore, dimensionless wall distance (y^+) should be carefully chosen such that $30 < y^+ < \sim 100$. For this reason, wall-adjacent cell centroid should be within the log-law layer. On the other hand, the position of free-surface was tracked using VOF method [10, 11]. As stability condition, $\max Co = 1$ and $\max \alpha Co = 1$ were specified. $\max Co$ represents maximum allowed courant number and $\max \alpha Co$ represents maximum allowed courant number of the phase interface in mutliphase flows. The solvers used are GAMG (Geometric agglomerated algebraic multigrid) for pressure and PbiCG (Preconditioned bi-conjugate gradient) Stabilized for velocity solution. Discretization of time was made using Implicit Euler scheme. Upwind discretization scheme is used for convection.

2.3 Boundary Conditions

Figure 4 shows the flow domain and boundary conditions used for the numerical model of the open channel flow over the broad-crested weir. The modelling has been realized in three dimensions. The channel bottom and side walls were specified as wall. In SimFlow, boundary conditions are presented in Table 1.

Table 1. Boundary Conditions*

	Pressure	Velocity
Top	0	ZG
Bottom	FFP	NS
Inlet	FFP	VHI
Outlet	FFP	IO
Left Wall	FFP	NS
Right Wall	FFP	NS

* FFP (Fixed Flux Pressure); ZG (Zero Gradient); NS (No-Slip); VHI (Variable Height Inlet); IO (Inlet-Outlet).

Fixed Flux Pressure boundary condition sets the pressure gradient to the provided value such that the flux on the boundary is that specified by the velocity boundary condition [11]. For wall boundaries, the standard wall function option is used in which roughness height is specified 0 m and roughness constant is specified 0.5. The blue

region in Fig. 4 shows the initial condition. Water is specified upstream at a height equal to that of the weir.

Residuals in velocity components and pressure less than 0.001 were taken as termination criteria.

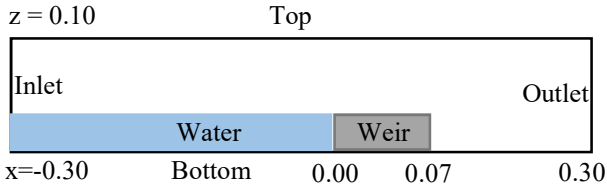


Figure 4. Numerical flow domain and boundary conditions

2.4 Mesh Convergence Study

Firstly, a mesh convergence study was performed by running the model several times with different number of cells. After each run, the change in maximum pressure was observed. In the mesh with 83054 cells this value reduced below 0.1% (Fig. 5) and hence the mesh convergence was realized. The base mesh consists of 160, 15 and 27 rectangular cells in x, y, z directions respectively. A regular cell has a size of 0.375 cm. In addition, 2 levels of refinement was performed around the weir area.

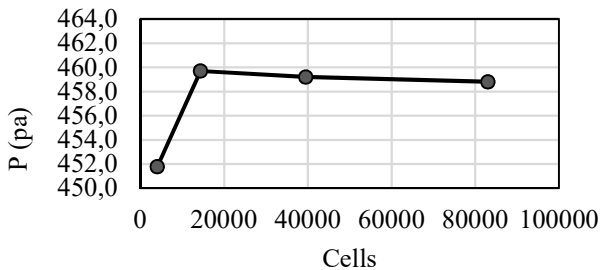


Figure 5. Maximum pressure against varying number of cells.

3. RESULTS

The discharges measured, dimensionless wall distance y^+ and H/L ratios are given in Table 2. As can be seen from Table 2, all H/L ratios are smaller than 0.5.

Table 2. Measured discharge, upstream Froude number (Fr_{up}), dimensionless wall distance (y^+) and H/L ratios

Exp. no	Discharge (L/s)	Fr_{up}	y^+	H(cm)	H/L
1	0.277	0.13218	71.21	2.4	0.34
2	0.330	0.14852	75.23	2.7	0.38
3	0.391	0.17128	79.25	2.8	0.40
4	0.456	0.18471	80.24	3.1	0.44
5	0.514	0.19805	81.22	3.3	0.47

3.1. Water Surface Profiles

An example photo of water surface from experiment and the corresponding numerical solution are presented in Fig. 6. It is clear that the trapped air under the nappe is calculated by SimFlow. On the other hand, the comparison of water surface profiles between numerical and experimental results are presented in Figs.7-11.

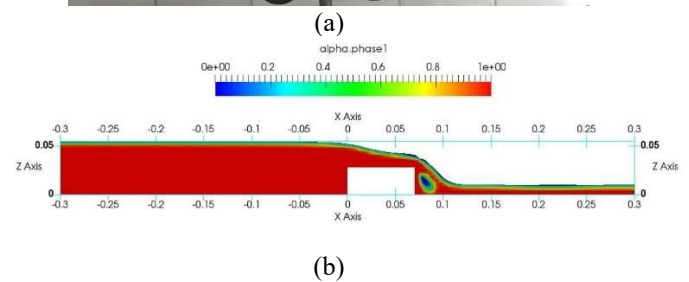
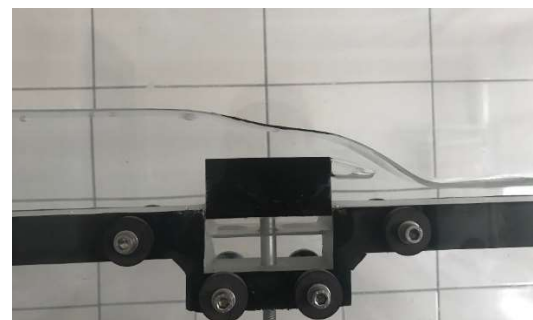


Figure 6. Water surface for $Q_3=0.391$ L/s (a) from experiment, (b) from numerical solution, colors indicate volume fraction of water.

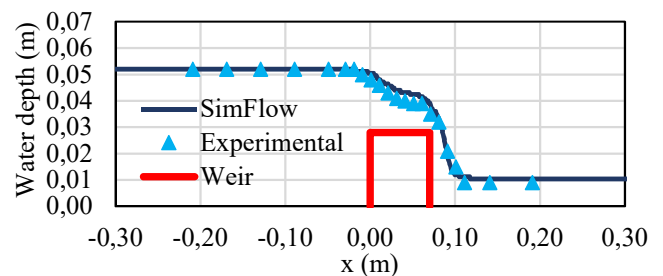


Figure 7. Comparison of experimental and numerical water surface profiles for $Q_1=0.277$ L/s

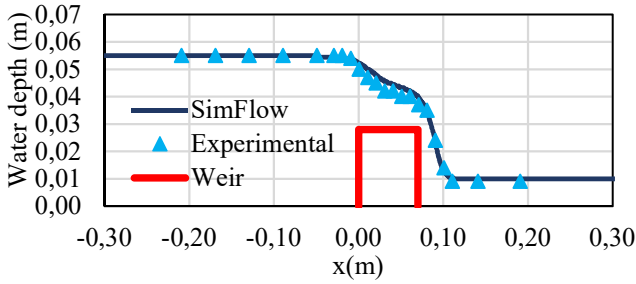


Figure 8. Comparison of experimental and numerical water surface profiles for $Q_2= 0.330$ L/s

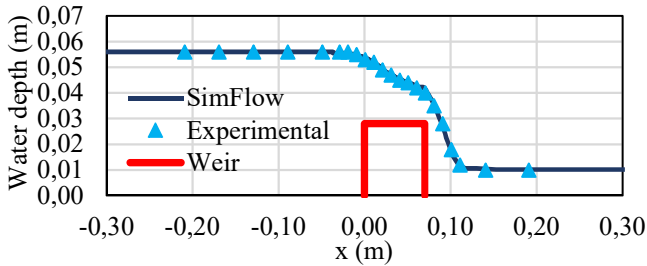


Figure 9. Comparison of experimental and numerical water surface profiles for $Q_3= 0.391$ L/s

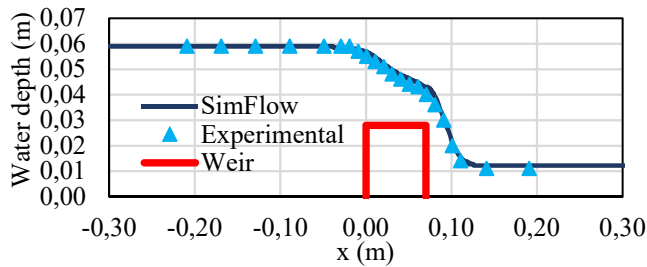


Figure 10. Comparison of experimental and numerical water surface profiles for $Q_4= 0.456$ L/s

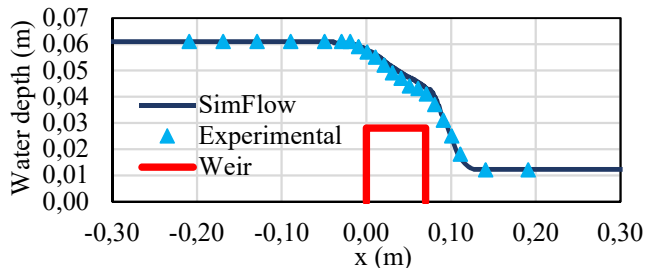


Figure 11. Comparison of experimental and numerical water surface profiles for $Q_5= 0.514$ L/s

From Figs. 7-11, it can be seen that there is generally a good agreement between the experimental and numerical results. NRMSE (Normalized Root-Mean-Square Error) values for water surface profiles in all experiments are given in Table 3. The normalization was done with respect to water depth at upstream.

Table 3. NRMSE in water surface profiles for all flow rates.

Exp. 1	Exp. 2	Exp. 3	Exp. 4	Exp. 5
0.0326	0.0284	0.0011	0.0121	0.0117

As seen from Table 3, errors are relatively smaller in experiments 3, 4 and 5. The smallest error is observed in experiment 3.

3.2 Velocity Profiles

The comparison of velocity profiles between numerical and experimental results for $Q_5=0.514$ L/s are presented in Figs. 12-15.

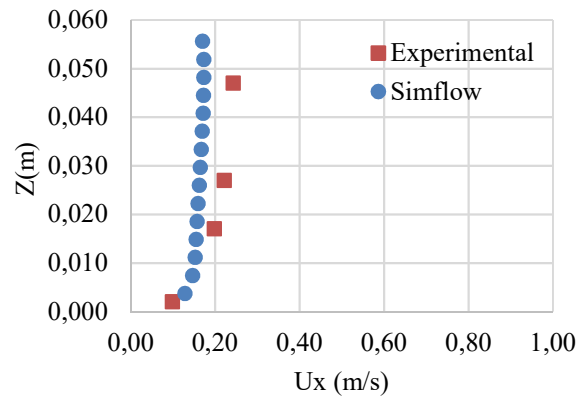


Figure 12. Comparison of velocity profiles between numerical and experimental results at $x= -0.049$ m.

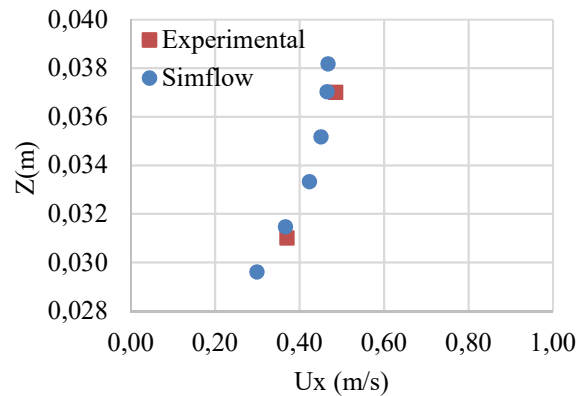


Figure 13. Comparison of velocity profiles between numerical and experimental results at $x= 0.031$ m.

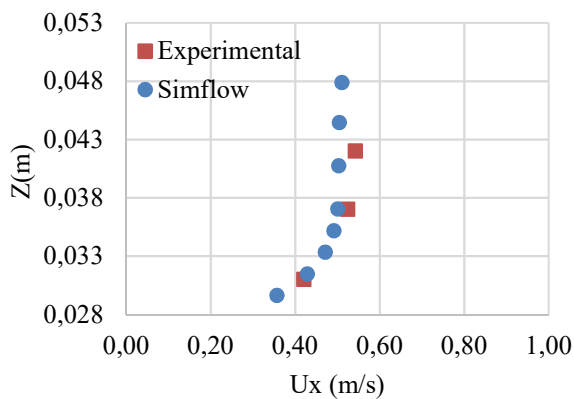


Figure 14. Comparison of velocity profiles between numerical and experimental results at $x= 0.041$ m.

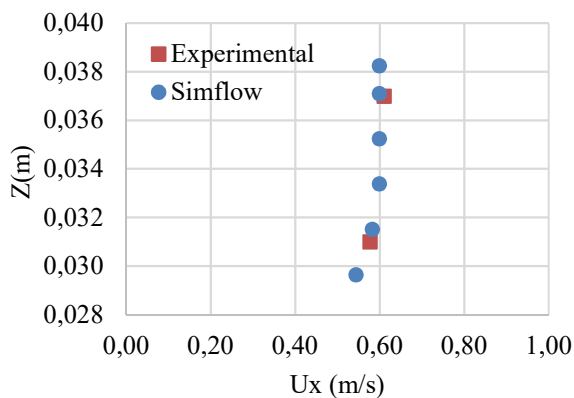


Figure 15. Comparison of velocity profiles between numerical and experimental results at $x= 0.061$ m

From Figs. 12-15, the examination of the velocity profiles shows generally good agreement between experimental and numerical results. NRMSE values for all velocity profiles in all experiments are given in Table 4. The normalization was done with respect to mean velocities.

Table 4. NRMSE in velocity profiles for all flow rates.

X(m)	Exp. 1	Exp. 2	Exp. 3	Exp. 4	Exp. 5
-0.049	0.732	0.350	0.246	0.437	0.338
0.031	0.861	0.516	0.242	0.141	0.095
0.041	0.784	0.308	0.235	0.110	0.174
0.061	0.275	0.093	0.128	0.086	0.055

According to Table 4, errors generally decrease with increasing discharges. The smallest overall error is observed in experiment no. 5.

3.3 Pressure distributions

The pressure obtained from numerical models results are presented for each flow rate in Figs. 16-20.

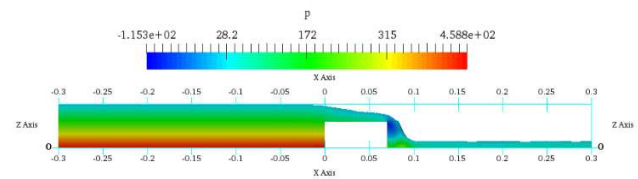


Figure 16. Pressure distribution for $Q_1= 0.277$ L/s.

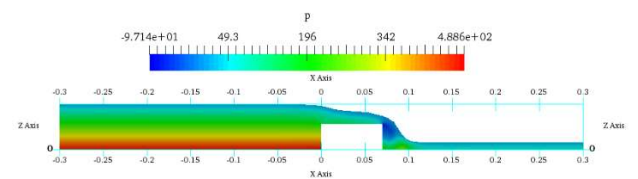


Figure 17. Pressure distribution for $Q_2= 0.330$ L/s.

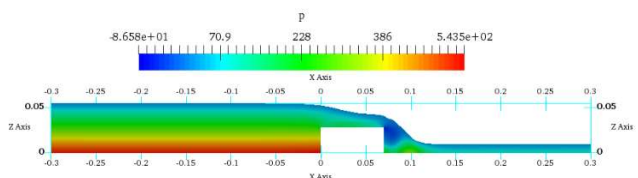


Figure 18. Pressure distribution for $Q_3= 0.391$ L/s.

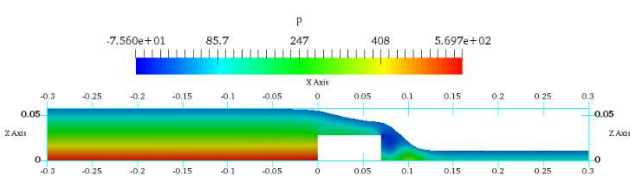


Figure 19. Pressure distribution for $Q_4= 0.456$ L/s.

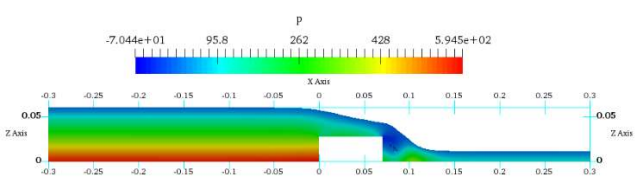


Figure 20. Pressure distribution for $Q_5= 0.514$ L/s.

4. CONCLUSION

In this study, the flow field over a broad-crested weir was modeled experimentally and numerically under five different flow rates. Experiments were carried out in the Hydraulic Laboratory of Civil Engineering Department at Uludağ University in an open channel with dimensions of 1.44 m x 0.055 m. SimFlow software was used to develop the numerical model. In numerical solutions the realizable $k-\epsilon$ turbulence model was used and the free water surface profile was determined by the

VOF method. The examination of the water surface profiles revealed that there is generally a good agreement between the experimental and numerical results. According to Table 4, fitting on the velocity profiles improve over the weir and errors decrease with increasing flow rates. The smallest overall error is observed in experiment no. 5. As an interface to the open source CFD program OpenFOAM, SimFlow performed well in modeling the flow over broad-crested weir. In SimFlow, drawing geometries or importing them and generating meshes with refinement is possible. In addition, most of the OpenFOAM solvers and boundary conditions are supported which makes SimFlow comparable to other CFD software.

REFERENCES

- [1] A. Duru. "Numerical modelling of Contracted sharp crested weirs", . *Yüksek Lisans Tezi*, ODTÜ The Graduate School of Natural and Applied Sciences, Ankara, 2014.
- [2] S. Haun, N. Reidar, B. Olsen and R. Feurich, "Numerical modeling of Flow over trapezoidal Broad Crested Weir", *Engineering Applications of Computational Fluid mechanics*, Vol. 5, No. 3, 397-405, 2011.
- [3] K. Subramanya, *Flow in open channel*. Tata McGraw-Hill, 7 West Patel Nagar, New Delhi, 547pp, 2009.
- [4] R.H. French, *Open-channel hydraulics*. McGraw-Hill, Singapore, 693 p, 1986.
- [5] M. İlkentapar, "Açık kanallarda geniş başlıklı savaklar üzerinden akımın deneysel ve sayısal modellenmesi", *Yüksek Lisans Tezi*, EÜ Fen Bilimleri Enstitüsü, İnşaat Mühendisliği Anabilim Dalı, Kayseri, 2015.
- [6] H. Chanson, J.S. Montes, "Overflow Characteristics of Circular Weirs: Effect of Inflow Conditions", *Jl of Irrigation and Drainage Engrg*, ASCE, Vol. 124, No. 3, 152-162 (ISSN 0733-9437), 1998.
- [7] N.G. Soydan, M. S. Aköz, O. Şimşek ve V. Gümüş, "Trapez Kesitli Geniş Başlıklı Savak Akımının k-ε Tabanlı Türbülans Modelleri ile Sayısal Modellenmesi", *Çukurova University Journal of the Faculty of Engineering and Architecture*, 27(2), 47-58, 2012.
- [8] T.T. Shih, W.W. Liou, A. Sahbbir, Z. Yang, J. J. Zhu, "A new k-ε eddy viscosity model for high reynolds number turbulent flows-model development and validation", Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition Lewis Research Center, Cleveland, Ohio ,24 (3), 227-238, 1995.
- [9] H. Bal, "Geniş başlıklı savak içeren açık kanal akımın sayısal modellenmesi", *Yüksek Lisans Tezi*, ÇÜ Fen Bilimleri Enstitüsü, İnşaat Mühendisliği Anabilim Dalı, Adana, 2011.
- [10] M. Hamad. "Investigation of flow characteristics through box shape culvet combined with broad crested weir", *MSc thesis*, University of Gaziantep, The Graduate School of Natural and Applied sciences, Gaziantep, 2013.
- [11] Anonim, The open source CFD toolbox. <https://www.openfoam.com>, last accessed: 10.10.2018.