Proceedings of International Conference on Structural Architectural and Civil Engineering
Held on 21-22, Nov, 2015, in Dubai, ISBN:9788193137321

COMPARISON OF EXPERIMENTAL STUDY AND CFD ANALYSIS OF THE FLOW UNDER A SLUICE GATE

Ali Yıldız
Selcuk University, Konya,
TURKEY Yurdagul Kumcu

Necmettin Erbakan University,
Koycegiz Campus,
Konya, Turkey

A. İhsan Martı
Selcuk University, Konya,
TURKEY

***Abstract*— Flow in the open channels has great importance in the hydraulic engineering. Any disturbances as contractions and expansions affect the flow characteristics like velocities, flow rates and water surface profiles. Their effects on the flow must be considered during the design process of flow structures. In the last decades numerical solutions become popular as experimental studies take time, may be not economical and have scale effect. Computational Fluid Dynamics (CFD) is a type of numerical modeling and helps us to solve problems involving fluid flow. In this study, some physical experiments were set in the laboratory to obtain data required for flow along the sluice gate and conjugate depths of hydraulic jump of flow after sluice gate. FLOW-3D, which uses VOF (Volume Of Fluid) method and solve RANS (Reynolds Averaged Navier-Stokes) equations, is used for numerical modeling in order to verify experimental results. All measurements were compared with FLOW 3D in order to verify the ability of the CFD modeling. FLOW 3D and experimental results showed 95% consistency with each other.**

***Keywords*—Flow-3D, numerical modeling, open channels, sluice gate.**

I. Introductıon

Sluice gates are widely used in hydraulic structures for controlling the discharges and water levels in hydraulic structures [1]. Additionally, the sluice gates are used to measure flow rates, and they capture floating elements that may cause accumulation on the downstream [2].

In order to make an economical and functional sluice gate design, the hydraulic variables such as flow rate, pressure, and velocity should be investigated accurately. In addition to the analytical methods, the design of sluice gates may be supported by physical models. At the design stage of hydraulic structures, the small-scaled physical models are constructed to observe the behavior of water and determine the problems that may be encountered. However, preparing model tests requires professional labor. Moreover, experimental studies may be more expensive, takes longer time and have scale effect.

Computational Fluid Dynamics (CFD) is a type of including fluid flow [3] and examines fluid-fluid and fluid-solid interactions. Although the analysis of a numerical model takes too much time on computer, the results of a numerical model provide 3-D flow data which cannot be obtained from 1-D and 2-D models.

Mostly, the results of the numerical model and the physical model are compared to determine the reliability of the results obtained by the numerical model. Besides the accuracy test of the numerical model, these comparisons are also used for the calibration of the numerical model. During the comparison process between the numerical and physical models, they must be evaluated in terms of hydraulic engineering judgment.

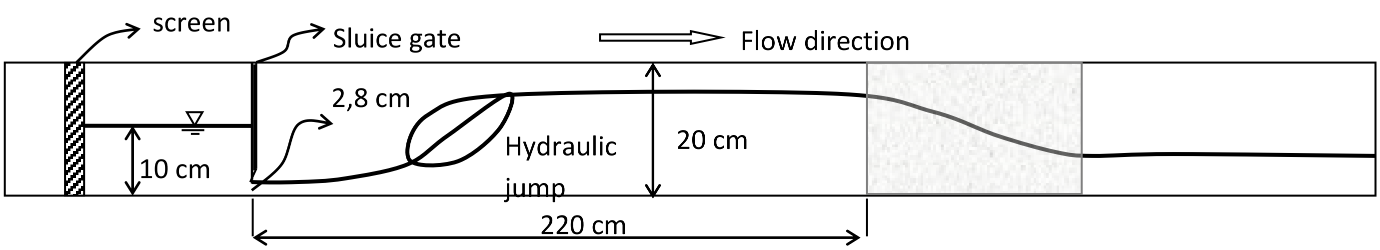
In this study, an experimental study for the flow under a sluice gate was performed in the laboratory, in a flume. Then, a numerical model of the sluice gate was built using Flow-3D computer program widely used in CFD researches. Finally, the results of the numerical and physical models were compared and discussed with each other.

II. Experimental Study

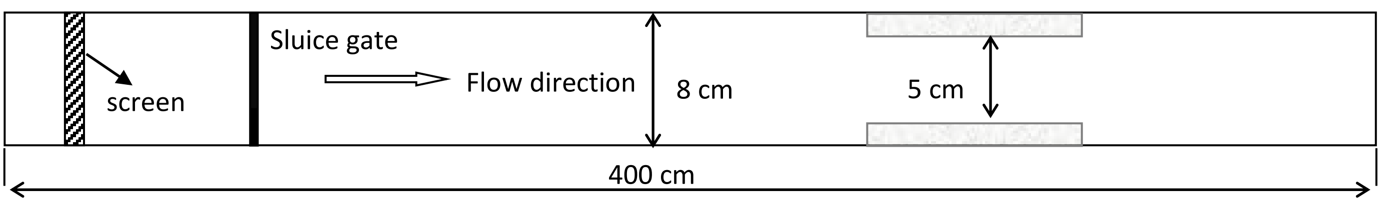
The experimental setup contains a simple open channel which was made of smooth plastic so that to follow the flow phenomena along the channel. The flow channel is 4m long, 0.08m wide and 0.20m height with a slope of 0.0001. The experimental setup used in the tests can be seen in Fig. 1.

Fig.1 Physical Model

The flume had a closed loop water system and the flow to the flume was supplied from a constant head water tank by a pump.



(a)



(b)

Fig. 5 (a) Comparison of velocities (b) Comparison of flow depths

Water coming from the pipe caused waving and three screens were observed on the channel. The screens behaving
as filters which act as a breakwater and provide a smooth profile before the sluice gate.

A gate was mounted at the tail end to adjust the flow depths. The flow velocity was measured using a flow meter and the flume discharge was calculated by multiplying the velocity of the channel with the cross-section of the flow area. The sluice gate controlled the water level in the reservoir and drained excessive water when necessary. The flow rate and the velocity of the water passing under the sluice gate were calculated depending on the opening of the sluice gate.

During the experimental studies, a sluice gate was placed at the upstream part of the flume and a contraction was set at the downstream of the channel as seen in Fig.3. Water was released under the sluice gate and a hydraulic jump occurred as the flow regime changed. In the flume, water level increased because of the sudden contraction and resulted in a hydraulic jump between the sluice gate and the contractions. If the flow regime changes from supercritical to subcritical flow, hydraulic jump occurs. Although one of the most important engineering applications of hydraulic jump formation is to dissipate the excess energy in the channels so that reducing the damages on the water structures.

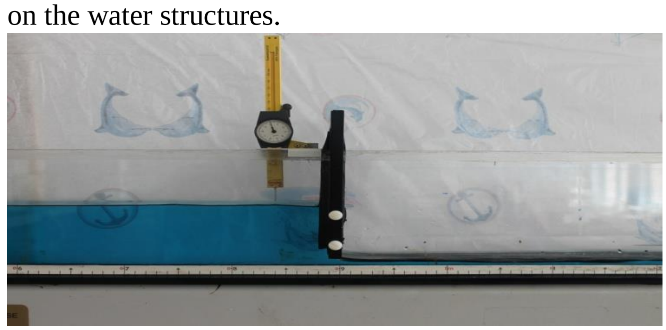


Fig. 3 Side view of the sluice gate

By the way, uncontrolled or unexpected hydraulic jump can be very dangerous for hydraulic structures.

Therefore, the location of the hydraulic jumps should be well determined in the design stage of the hydraulic structures accurately.

III. Numerical Method

Flow-3D

The flow at the channel was modeled by Flow-3D VOF-based CFD program and solved in the Reynolds Averaged Navier Stokes (RANS) equation with the k-ε turbulence model [4] and Shallow Water Equations (SWE). The program evaluates the location of the flow obstacles by implementing a cell porosity technique called as the fractional area/volume obstacle representation of FAVOR method [5]. Flow-3D is very successful especially for solving open channel flows. The computational domain is subdivided using Cartesian coordinates into a grid of variable-sized hexahedral cells. The general governing RANS (2) and the continuity equations (1) for an incompressible flow, including the FAVOR variables, are given by

$$\frac{\partial}{\partial x_i}(u_i A_i) = 0 \qquad (1)$$

$$\frac{\partial u_i}{\partial t} + \frac{1}{V_F}\left(u_j A_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{1}{\rho}\frac{\partial}{\partial x_i} + g_i + f_i \qquad (2)$$

where u_i represents the velocities in the x_i directions (x, y, z-directions); t is time; A_i is the fractional area open to the flow in the subscript directions; V_F is the volume fraction of fluid in each cell; p is the hydrostatic pressure, g_i is the gravitational acceleration in the subscript directions; and f_i represents the Reynolds stresses for which a turbulence closure model is required.

Solution region, initial and boundary conditions

The analysis time of the model was selected as 80 seconds, the distance units were set in SI system and the type of temperature is chosen as Degree Celsius. The water at 20^{o}C temperatures was selected as the fluid, and then the incompressible flow mode was activated. As the turbulence model ReNormalized-Group (RNG) [6], [7] model is suitable for the model.

The physical model was drawn with the original size in AutoCAD and imported to Flow-3D with STL format. The computational domain involved uniform rectangular grids of grid spacing Δx=Δy=Δz=0.003 m, resulting in the total number of mesh blocks of 5,823,120.

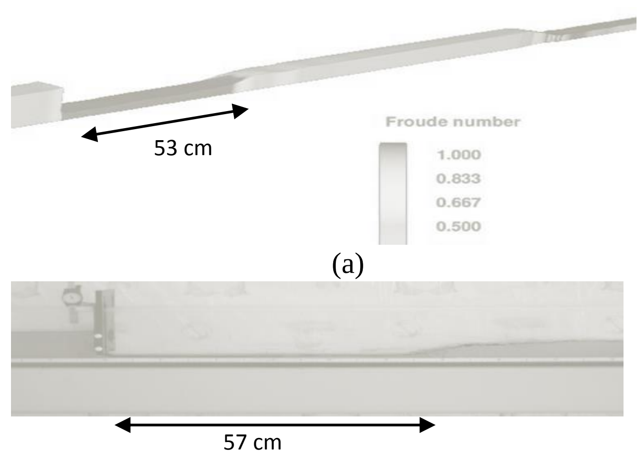
To represent the physical model accurately, the boundary conditions have to be defined carefully. There are 6 boundary conditions representing the boundary of Cartesian system (+x, +y, +z, -x, -y, -z). The boundary conditions were designed to be compatible with the psychical experiments in real conditions.

The upstream boundary conditions was specified as 10 cm hydrostatic pressure; the side walls and bottom of channel were defined as wall; no slip boundaries of zero tangential and normal velocities. On the top, atmospheric boundary condition was assigned to describe the free surface flow condition.

IV. Results and dıscussıon

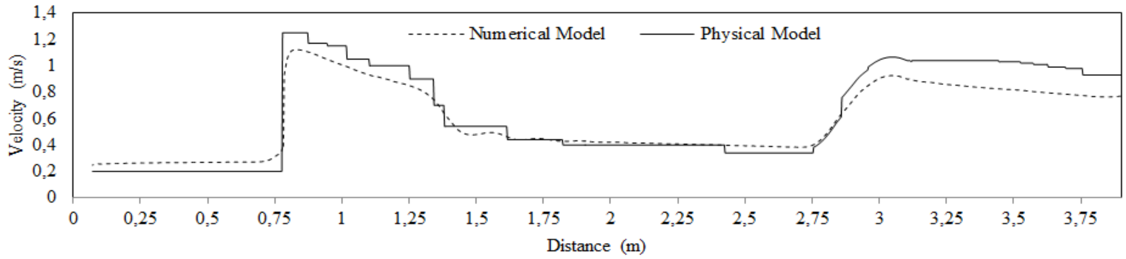
The physical model carried out in the laboratory and the numerical model analyzed by Flow-D numerical model were compared and discussed in terms of flow depths, velocities and free surface elevations along the channel.

The main objective of the comparison of the physical model with the numerical model was to determine the consistency of the models with each other.

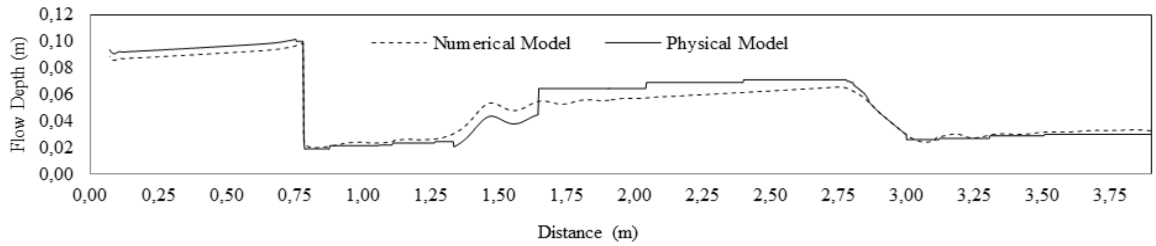


(b)

Fig.4 Hydraulic jump in (a) numerical model (b) physical model



(a)



(b)

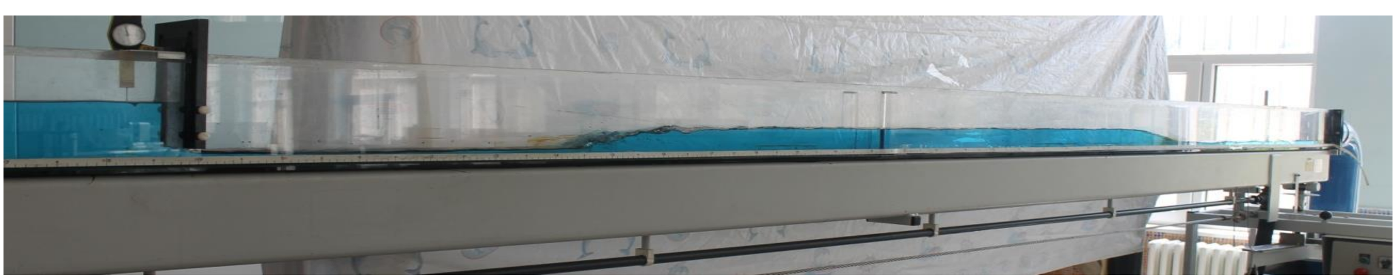
Fig. 5 (a) Comparison of velocities (b) Comparison of flow depths

So, the free surface profile, the place and the shape of the hydraulic jump and the change of the water surface profile due to the contraction were investigated experimentally in the laboratory and the results were compared with the numerical model results.

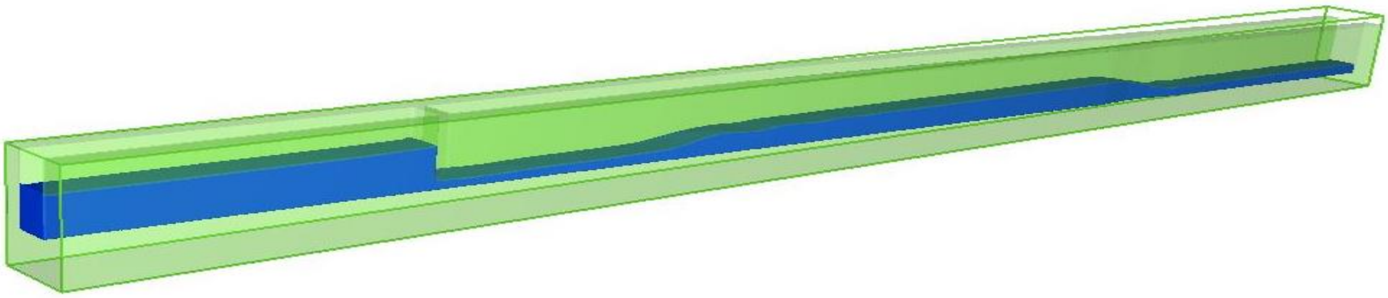
The hydraulic jump was observed at both physical and numerical models. The shapes and the places of the hydraulic jumps in physical and numerical models were respectively determined as 53 cm and 57 cm away from the sluice gate. The numerical model provided 92.5% success in terms of the position of the hydraulic jump (Fig. 4).

In order to draw a free surface profile and make comparison, the flow depth of the water in the channel was measured by a point-gage with the distance of 5 cm and 10 cm and the flow depth of the numerical model was taken as a distance of 5mm. There were no major differences between the models in terms of free surface profiles, and they presented 87% consistency as seen from Fig. 5 (a).
This study showed that the numerical tools using RANS equations are sufficiently advanced to simulate a flow passing through a sluice gate. As seen from Fig. 6, the physical model and the numerical model are similar to each other.

The flow depth measurements taken from the physical model were affected from the waves, since especially the water falling from the pipe was causing some fluctuations. As a conclusion, some small differences obtained between the flows depths obtained by the experimental tests and numerical model studies; especially after the hydraulic jump. If the measurements can be made more accurately, the consistency of the models will increase and the differences can be ignorable.



` (a)



(b)

Fig. 6 (a) General view of physical experiment in the laboratory (b) Numerical model view done by Flow-3D

V. References

[1] V. T. Chow, *Open - Channel Hydraulics*, McGraw Hill Publication, Japan, 1959.

[2] W.H. Hager and M. Schwalt, "Broad Crested Weir" *Journal of Irrigation and Drainage Engineering*, vol. 120, no.1, pp: 13-26, 1994.

[3] C. W. Hirt, "Volume of fluid Method for the Dynamics of Free Flow Modeling," *Flow Science Report* FSI-92-00-02, Flow Science, Inc, Santa Fe NM, 1981.

[4] B.E. Launder and D.B. Spalding "The numerical computational of turbulent flows" *Computer Methods in Applied Mechanics and Engineering*, vol. 3, no.2, pp.269-289, 1974.

[5] Flow Science Inc, *Flow-3D User's Manuals.* Santa Fe NM, 2007.

[6] V. Yakhot and S.A. Orszag, "Renormalization group analysis of turbulence. I. Basic theory." *J. scientific Computing*, vol. 1, no. 1, pp. 1-51, 1986.

[7] Yakhot, and L.M. Smith, "The renormalization group, the ε-expansion and derivation of turbulence model," J. Scientific Computing, vol. 7, no.1,pp. 35-61, 1992