10.46299/ISG.2022.MONO.TECH.4.1.5

1.5 Modeling of weirs by using CFD

Weirs are commonly used in irrigation channels and environmental projects for flow measurement and flow control in open-channel flows. Weirs increase water storage or irrigation capacity by raising the upstream water level [37]. Usually, weirs also functioning as devices for measuring the discharge of the flow in open channels [38]. Taking into consideration the relative width of the weir crest H_0/L, weirs can be classified into the following groupes:

- $0 < H_0/L \leq 0.1$ for long-crested weirs;
- $0.1 < H_0/L \leq 0.4$ for broad-crested weirs;
- $0.4 < H_0/L \leq 1.5$ for short-crested weirs; and
- $1.5 < H_0/L$ for sharp-crested weirs,

where H_0 is the upstream total head, and L is the width of the weir crest in the direction of flow [38]. The broad-crested weir can be considered as the separate type of the weir. Comparing with the other types of the weirs, the broad-crested weirs are noted with many advantages, such as structural stability, very low cost and extremely low sensitivity to tailwater submergence [38]. Tracy [39], Isaacs [40], and Hager and Schwalt [38] have performed a lot of different series of the experiments in order to perform a deep study of the hydraulic characteristics of broad weirs with various geometries and designs. All their previous studies focused on determining the discharge coefficient and revealing the influence of the geomeries on the flow pattern. Usually, the physical phenomena of the flow over a broad-crested weir under free-flow conditions can be described by the following equation:

$$Q = C_d b(2gH_0^3)^{1/2} \tag{1}$$

where Q – flow discharge rate, C_d – discharge coefficient, b – width of weir (in the direction perpendicular to the flow direction), g – gravitational acceleration, and H_0 –

upstream total head. $H_0 = h_0 + Q^2/[2gb^2 (h_0 + P)^2]$, where h_0 – upstream overflow piezometric head and P – weir height.

Numerical modeling of weirs is very difficult and important topic in the engineering work and it can be used instead of physical modeling to save cost, effort and time due to the advances in computer technology. CFD is a numerical tool that was used for simulating flow by applying COMSOL Multiphysics simulation software for solving problems in open channels or some other modelling software platforms. The assumptions of flow over weirs which used to derive and solve using numerical techniques are fully turbulent, unsteady, incompressible, and non-viscous. The differential equations used in Fluent software are continuity and RANS (Reynolds Averaged Navier–Stokes equations). These equations are based on the principles of conservation of mass and Newton's Second Law.

Thus, the process of the numerical modelling of the weirs involves the solution of the Navier-Stokes equations, which are based on the assumptions of conservation of mass and momentum within a moving fluid. Taking into account, the absence of sources of mass and momentum, the conservation of mass is described by the differential equation:

$$\frac{\partial \rho}{\partial \iota} + \nabla \cdot (\rho v) = 0 \qquad (2)$$

where ρ – is the density and v – is the velocity of the fluid. The conservation of momentum is similarly described by the equation:

$$\frac{\partial}{\partial \iota}(\rho v) + \nabla \cdot (\rho v v) = -\nabla p + \nabla \cdot \tau \qquad (3)$$

Equations (2) and (3) are presented for completeness only; readers are referred to Ferziger and Peric (1997)(3) for a detailed description of their derivation and solution using numerical techniques. In order to represent the real impact of the turbulence effect on the flow, additional transport equations are solved for various turbulence quantities.

All previously performed studies were focused on experimental aspect and well organized and performed studies of this phenomena. The results of these studies showed that the upstream angle significantly affects a broad-crested weir's hydraulic performance in terms of free-surface profile, discharge coefficient, velocity profile, and flow separation zone. Thus, despite computational fluid dynamics (CFD) becoming an extremely important method which can be used for investigations of the hydraulic problems. Also, less attention has been given to study the effect of the upstream angle on the hydraulic performance of a broad-crested weir based on numerical simulations. Sarker and Rhodes [48] conducted a numerical simulation of the flow over a broad-crested weir based on the standard *k-e* turbulence closure model. Obtained by the computing free-surface profiles which were based on the volume-of-fluid (VOF) method that were found to agree well with the measured results. Numerous research performed experimental and numerical simulations of two-dimensional free-surface flows interacting with rectangular and triangular broad-crested weirs. In their study, the standard k-e and standard *k-w* turbulence models were used. The numerical computing results showed that the predicted values and parameters for the velocity field and free-surface profile from the standard k-w turbulence model were in better alignment with values obtained during the experimental research. Haun et al. [49] performed a numerical simulation of flow over a trapezoidal broad-crested weir.

They have proved the applicability of not only two-phase (COMSOL Multiphysics), but single-phase CFD modeling (SSIIM) as well for such flow fields. Akoz et al. [50] performed a two-dimensional simulation of flow over a semi-cylindrical weir using different turbulence closure models. Comparing the experimental and CFD results showed that the numerical simulation provided reasonable forecast results in terms of the velocity field and the free-surface profile. In this study, a series of numerical simulations was conducted to study broad-crested weirs with sloped upstream faces and vertical downtown faces, considering that the downstream slope has less of an impact on a broad-crested weir under free-flow conditions [44,45]. Numerical simulations were performed using the COMSOL Multiphysics simulation software. Its two-phase solver, which is based on the volume of fluid (VOF) method, has been proven to successfully model complex flows [51–53]. The main goals of the present study were as follows: (1) to validate the InterFoam solver with a free-surface profile, (2) to discuss the effects of the upstream face angle on discharge coefficient, and (3) to discuss the effects of the upstream face angle on the behavior of a broad-crested weir.

In order to perform comparisons of the experimental results and results obtained with CFD modeling was created a numerical model in the Multiphysics simulation software platform (figure 1.).

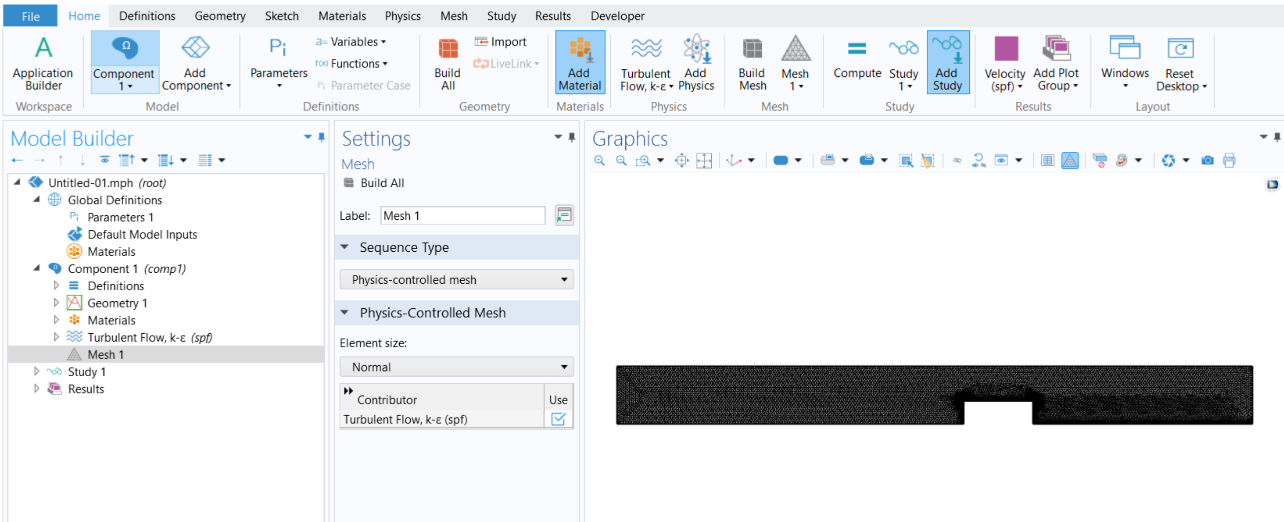

Fig. 1. Model setup with COMSOL Multiphysics simulation software.

The boundary conditions applied to the numerical model was following (figure 2.):

- Inlet: $u=u_0$; $\nabla p = 0$.
- Outlet: $\nabla u = 0$; $\nabla p = 0$.
- Wall: $u=0$; $\nabla p = 0$.
- Atmosphere: $\nabla u = 0$; $p = 0$.

In order to match the experimental conditions, the inlet boundary was applied to *U* ($u=u_0$;) which adjusted the flow depth and velocity at the inlet to obtain a constant flow rate (*Q*). The outlet boundary was equalized to a zero gradient velocity condition in the model, which was achieved by Neuman BC condition ($\nabla u = 0$; $\nabla p = 0$).

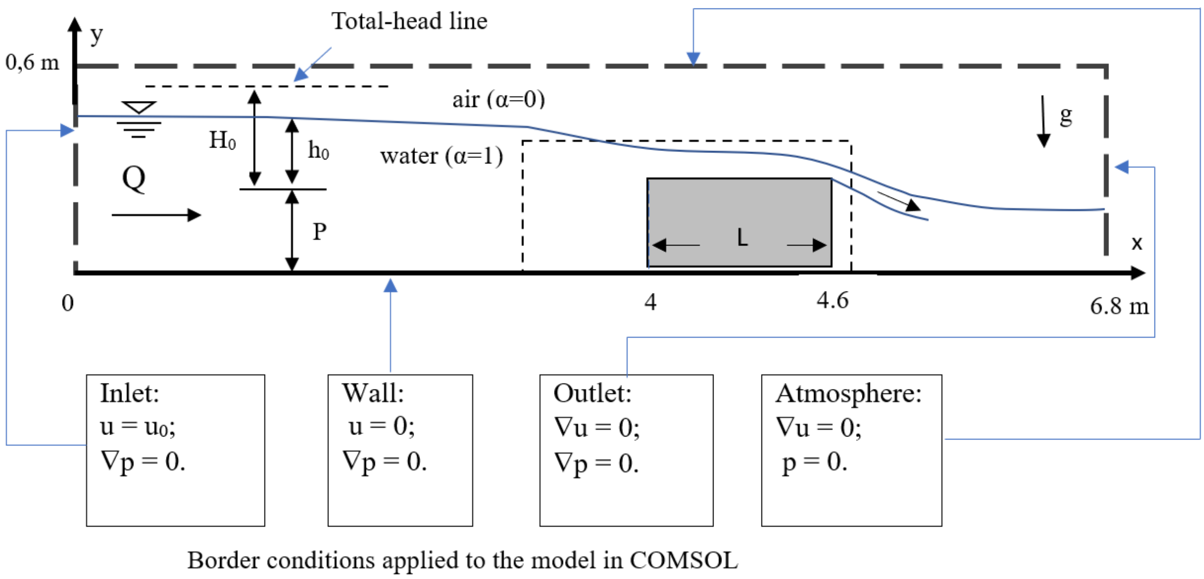

Fig. 2. Boundary conditions using COMSOL Multiphysics simulation software.

The no-slip boundary condition was applied to the surface of the channel by choosing wall condition (u=0; ∇p = 0). The top surface of the model was considered as a free surface (atmosphere), so, for *U* was imposed to allow the flow to enter and leave the simulation domain.

The mesh of the model (figure 3,4) was generated automatically by the COMSOL Multiphysics simulation software.

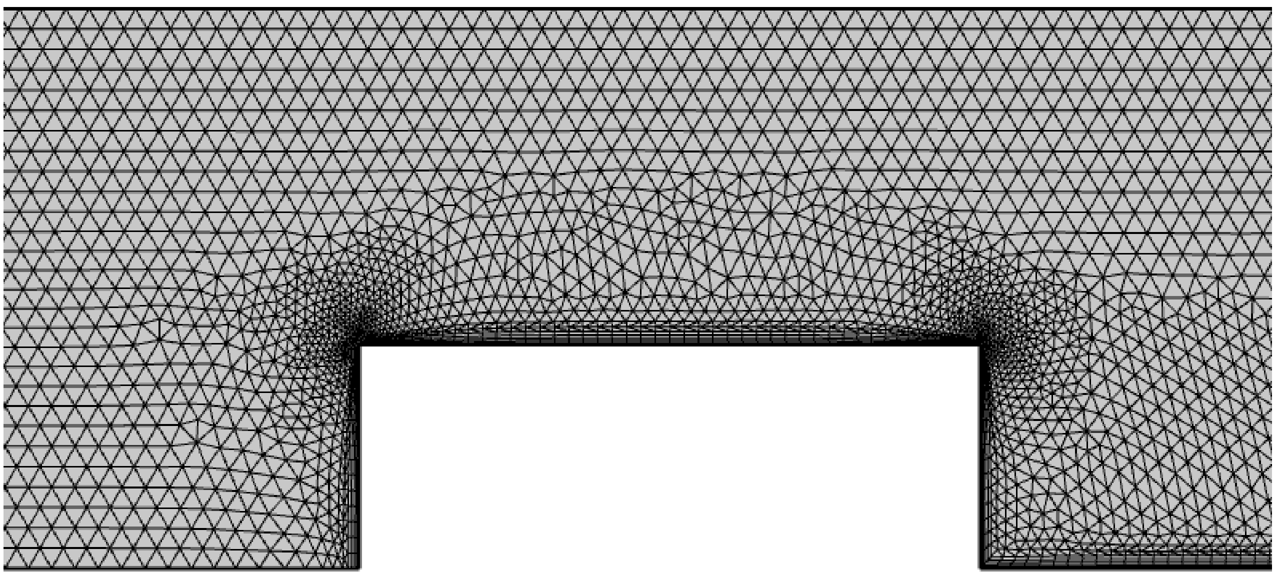

Fig. 3. Mesh generated near weir model.

The Mesh features enable the discretization of the geometry model into small units of simple shapes, referred to as mesh elements. For 1D geometries, the mesh generator discretizes the domains (intervals) into smaller intervals (or mesh elements). The endpoints of the mesh elements are called mesh vertices. The boundaries (or vertices) defined in the geometry are represented in the mesh by boundary elements (or vertex elements).

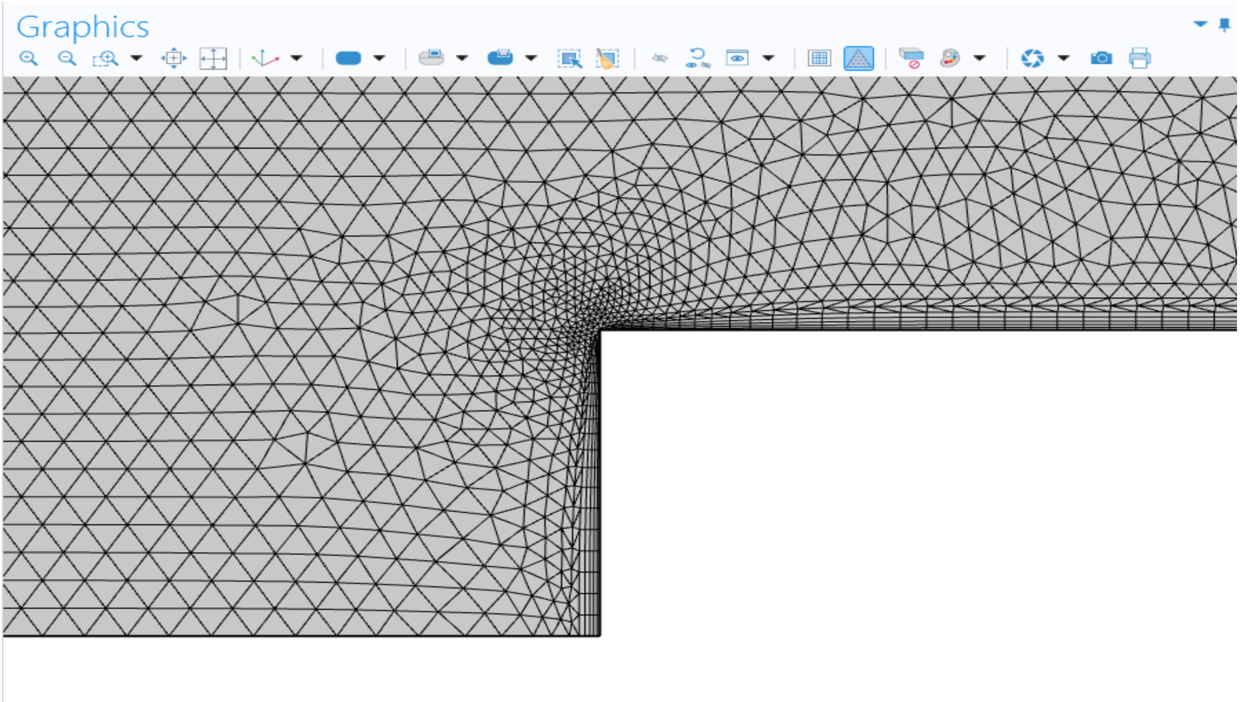

Fig. 4. Mesh generated near front area of the weir model.

For 2D geometries, the mesh generator discretizes the domains into triangular or quadrilateral mesh elements. If the boundary is curved, these elements represent only an approximation of the original geometry. The sides of the triangles and quadrilaterals are called mesh edges, and their corners are mesh vertices. A mesh edge must not contain mesh vertices in its interior. The boundaries defined in the geometry are discretized (approximately) into mesh edges, referred to as boundary elements (or edge elements), which must conform with the mesh elements of the adjacent domains.

For 3D geometries, the mesh generator discretizes the domains into tetrahedral, hexahedral, prism, or pyramid mesh elements whose faces, edges, and corners are called mesh faces, mesh edges, and mesh vertices, respectively. The boundaries in the geometry are discretized into triangular or quadrilateral boundary elements. The geometry edges are discretized into edge elements. Similar to 2D, the geometry vertices are represented by vertex elements.

In current research was used physic-controlled meshing. It is the default approach when mesh is controlled by the physics module applied to the model. The mesh is then adapted to the current physics settings in the model. For example, for a fluid-flow model you get a somewhat finer mesh than the default with a boundary layer mesh along the no-slip boundaries. If we want to modify the overall element size of the physics-induced mesh we select another element size in the settings window of the main mesh node and rebuild the mesh. In current research we built our mesh in "Extra Fine" predefined element size.

Numerical simulations in COMSOL Multiphysics simulation software were performed for short-crested weir with model where height (H_0) = 0,3 m and L = 0,6 m. Thus, H_0/L was equal to 0,5. The discharge of the flow via the weir was (Q) – 0,02 m^3/s. During the simulations was obtained numerical model of the flow with all parameters. The main aim of the research is to compare the free-surface profiles obtained during the experimental research performed by (LVIV) and profiles obtained with numerical simulations in COMSOL Multiphysics simulation software.

Thus, in this study, the discharge over the broad-crested weir was 0.02 m^3/s. The comparison between the free-surface profiles obtained in numerical simulations (performed by the standard *k-e* turbulence closure model and SST *k-w* turbulence model) and experimental profiles for thought three serious is shown in figure 3. Chen et al. [53] performed similar investigations with broad-crested weirs in different configurations. According to their results the free-surface profile can be divided into two parts based on the gradient. The first part is a slowly descending part and for the second part a sharply descending is the main characterization criteria. Also, their results shows that the curvature of the former part is close to zero, whereas that of the latter part decreases quickly with a hydraulic drop [53].

The results of the research: experimental and computer simulations are represented in figure 3. The free-surface profiles obtained in this study (Fig. 3) are consistent with experimental data and fully confirm the conclusions from previous investigations.

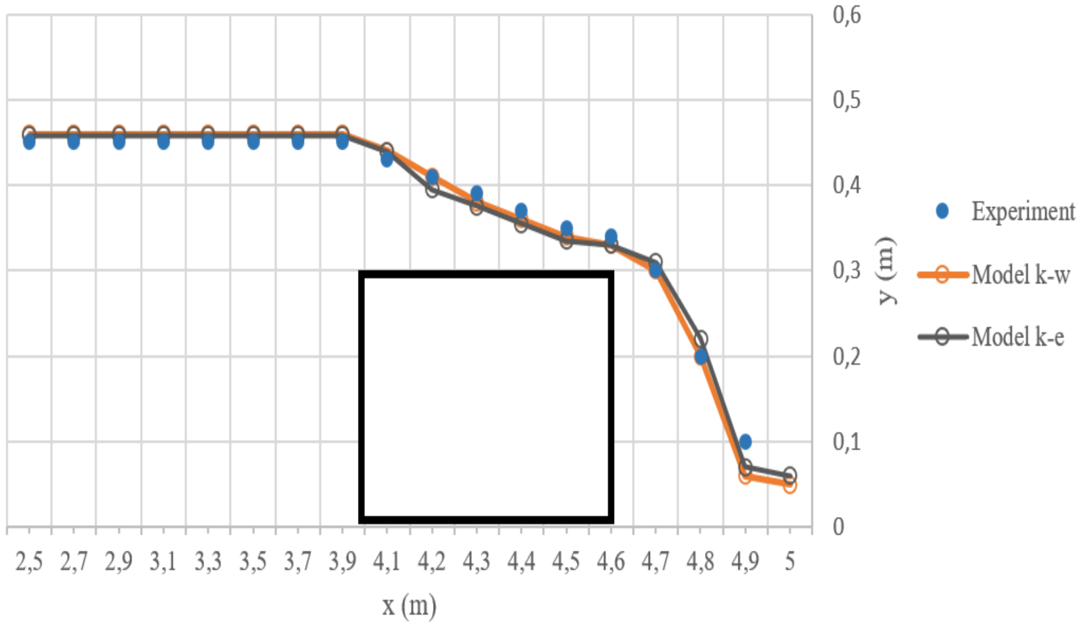

Fig. 5. Free-surface profiles obtained by experiment and computer modeling by using k–w and k–e turbulence modules from COMSOL Multiphysics simulation software with discharge (Q) – 0.02 m^3/s.

In order to perform further verifications of the numerical models and data received from computers simulation, the following parameters should be analyzed: the mean absolute percent error (MAPE) and the root mean square error (RMSE). Both statistical indicators were used to evaluate the accuracy of the numerical results obtained during the simulations. Both statistical indicators (the RMSE and the MAPE) were small which strongly confirms good correlations between the numerical and experimental results.

The values received from the statistical indicators for the free-surface profiles show a perfect match. For the standard *k-w* model, the range of the root mean square error parameter is between 0.4% and 0.74%, and the range of the mean absolute percent error is between 0.7% and 1.8%; for the *k-w* turbulence model, the range of the root mean square error is between 0.32% and 0.71%, and the value of the mean absolute percent error is between 0.64% and 1.53%. Statistical analysis of the results shows that both turbulence models (*k-e* and *k-w*) are able to perform computing simulations and describe the physical picture of the process with great alignments in terms of representing the free-surface profile over the broad-crested weir.