Journal of Civil Engineering and Materials Application
Journal home page: http://journals.lexispublisher.com/jcema

Received: 19 March 2017 • Accepted: 26 April 2017

Research

doi: 10.15412/J.JCEMA.12010104

Modeling of Turbulent Flow Due to the Dam Break Against Trapezoidal Barrier

Behrooz Moradi Mofrad[1*], Sayed Hamid Reza Barnjani[2], Ahmad Safari[1]

[1] Department of Civil Engineering, Yasuj Branch, Yasouj University, Kohgiluyeh and Boyer Ahmad, Iran

[2] Department of Civil Engineering, Bushehr Unit, Islamic Azad University, Bushehr, Iran

*Correspondence should be addressed to Behrooz Moradi Mofrad, Department of Civil Engineering, Yasuj Branch, Yasouj University, Kohgiluyeh and Boyer Ahmad, Iran; Tell: +989173418861; Fax: +987432228450; Email: behrouzmoradi63@gmail.com.

ABSTRACT

Dam is considered as a strategic structure whose collapse and destruction is a catastrophic event and could bring about significant life threatening and financial losses. Also its destruction may cause environmental damages due to uncontrollable exit of large amounts of water and sediment from the reservoir which results into propagation of devastating flood at downstream. Presence of barriers and buildings changes the flow patterns downstream of a dam. Regarding the importance of this issue, in this research modeling of this phenomenon was performed in the presence of a trapezoidal barrier using the finite volume method and OpenFOAM software. Modeling is in 2D form and, for validation of the results, use has been made of the numerical and experimental research conducted by other researchers. The results show that this model has a good performance in simulation of these problems and has been able to simulate the results with a good accuracy, compared to the experimental results. For simulation of other phenomena similar to the dam break, the present model could be developed.

Key words: Dam break, Two-phase flow, Finite volume, OpenFOAM software.

Journal of Civil Engineering and Materials Application *is* published by *Lexis Publisher*; Journal p-ISSN xxxx-xxxx; Journal e-ISSN 2588-2880.

1. INTRODUCTION

Dam is an engineering structure built in a valley or natural depressions and is a conventional method for storage of water flow in rivers. Regarding the high volume of used materials and high costs of constructing a dam, also its economic functionality and aspects of maintaining energy, water and job creation, it is considered as a strategic structure. Thus collapse of a dam or its destruction would be a catastrophic event and could bring about significant life threatening and financial losses. Also its failure may cause environmental damages due to uncontrollable exit of large amounts of water and sediment from the reservoir which results in downstream propagation of devastating flood (1). In addition to the complex dam break flow, presence of the barriers and buildings, changes the pattern of rapid flow at downstream of the dam which makes the problem more complicated (2). Therefore researchers and engineers do their best in comprehensive investigation of the problem and limiting the associated damages. Lauber and Hager (1998), investigated the effect of bed slope on dam break in a horizontal canal with a dry bed at downstream. They performed the experiment in a rectangular canal with 14m length and 0.5m width. Applying a simple static system, they were able to change the bed slope from horizontal to maximum 0.5 (26.5 degrees). They analytically determined the positive and negative wave fronts based on the characteristic equations which exhibited good agreement with the experimental results. Particle image velocimetry (PIV) method was applied for measurement of wave velocities (3). Ferrari et al. (2010), investigated wave propagation on a dry bed after the dam break. They used two models for comparing the simulation of dam break phenomenon. They developed the 2D model of dam break based on the shallow water assumptions and the 3D model based on the Smoothed-particle hydrodynamics (SPH) method. The comparison of results showed that the calculated results by SPH 3D model were superior to those obtained by the 2D shallow water model (4). Ozmen and Kocaman (2010), investigated the initial stages of dam break over a dry bed. The experiment was performed in a rectangular canal with 9m length, 0.3m width and 0.34m height. They installed the gate in a distance equal to 4.65m from the end of upstream canal and simulated the process of dam break by abrupt removal of the gate. The experimental results were compared to the results of commercially available CFD programs, solved Reynolds-averaged Navier–Stokes (RANS) equations, k-ε turbulence model, and shallow water equations. Measurement and calculation of the water level profile at the initial stages of dam break show that, while both models predict dam break flow with acceptable precision, RANS model exhibits a better compatibility (5). Biscarini et al. (2010), presented the numerical solution of flow due to dam break and compared it with the 3D simulated model using the shallow water equations. Their research was based on solution of the Depth-integrated Reynolds-averaged Navier–Stokes (DIRANS) equations using the Volume of Fluid (VOF) method. The results show that the shallow water equation method, although being able to properly model the main aspects of fluid flow, miss some 3D phenomena due to inaccurate idealization of shallow water flow, which ignores the 3D effects of gravity force (6). Bellos and Hrissanthou (2011) numerically simulated the flood wave propagation due to concrete dam break assuming one dimensional transient flow. They developed two numerical models based on the one dimensional shallow water equations (SWE) and Saint-Vansant equations using Lax-Wendroff and McCormack numerical models, respectively. The algorithm could successfully simulate flow over the bed with a triangular barrier and a high negative slope without any complicated consideration. They observed a high compatibility between the numerical and experimental results (7). Soares (2007), experimentally investigated the effect of bed slope on propagation of waves over the dry bed. He performed this using an example of one dimensional flow due to dam break over a downstream triangular barrier. In this study he used two measurement tools to obtain variations in the water level. The first was application of 3 gauges for measurement of water level and the second was application of the high speed digital camera where shooting was performed through the canal glass walls. Ultimately this method was successfully validated by comparing the gauges results (8). Marsooli et al. (2011), applied two 2D models for simulating transient flow due to the dam break. The first model which was the 2D horizontal model solved the shallow water equations by Godunov or upwind finite volume method. The second model was the 2D vertical model which solved Navier-Stokes equations using the VOF finite difference method to calculate the free water surface. Both models were examined for dam break waves flowing over the dry bed, humid bed and bed with triangular barrier. Also it was demonstrated that the 2D vertical model could investigate variation in the water level with more details, compared to the 2D horizontal one (9). Ozmen and Kocaman (2011) experimentally and numerically investigated the dam break flow over an initially dry bed with a barrier and examined formation and propagation of the negative wave behind the barrier. The flow was numerically simulated by VOF, based on the commercially available CFD program, FLOW3D. Comparison between the calculated results and experimental data showed that RANS model could appropriately model the flow with acceptable accuracy, while the simple SWE model exhibited some incoherence, especially concerning the negative wave propagation (10). Oertel and Bung (2012) investigated the dam break flow over a barrier in 2D form using the physical and numerical models and compared the results with those of Volume-of –fluid (VOF) method in the commercial code FLOW3D. It was found that the numerical modeling was consistent with the physical model results of the wave surface and velocity profile including the gate effects, whereas the analytical models were not able to account for the gate effects (11). Ozmen and Kocaman (2012), experimentally and numerically investigated dam break flow over a dry canal with abrupt contraction in a section of canal. The results showed that abrupt contraction in the canal section reflected a portion of flood waves toward the contraction and created a negative hole which was reflected upstream, whereas the remained portion moved downstream (12). Regarding the importance of this issue, reducing the hazards and associated damages due to dam break has been under focus of attention of engineers and researchers. In this research, regarding the vital importance of this problem, attempt is made that, by incorporating the finite volume method and OpenFOAM software and modeling of this phenomenon in the presence of a trapezoidal barrier, the probable associated damages are reduced as possible.

2. NUMERICAL MODELING

OpenFOAM which is open source software is a Computational Fluid Dynamics (CFD) code, which is able to model any problem related to the partial differential equations including numerical solution of the fluid flow problems, both the simple and complicated ones. The structure of this software is based on solution of 3D problems and takes advantage of a tensor with various orders to describe the physical characteristics of the problem. The available codes in OpenFOAM software are in C++ language. OpenFOAM uses the Finite Volume Method (FVM) for solution of partial differential equations (13).

3. GOVERNING EQUATIONS

The continuity equation for incompressible flow is as follows (13):

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

In the above equation, u and v are the components of velocity vector in the x and y directions, respectively. The general form of incompressible Navier-Stokes equations is as follows which includes four unknowns; u, v, w and P and, by adding the continuity equations, they could be solved:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+=g_x-\frac{1}{\rho}\frac{\partial P}{\partial x}+\frac{\mu_e}{\rho}\left(\frac{\partial^2 u}{\partial x^2}-\right. \qquad (2)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+=g_x-\frac{1}{\rho}\frac{\partial P}{\partial y}+\frac{\mu_e}{\rho}\left(\frac{\partial^2 v}{\partial x^2}+\right. \qquad (3)$$

Where g_x and g_y are the gravitational acceleration components, ρ is the water density and μ_e is the effective viscosity, where in laminar flow is equal to the fluid dynamic viscosity μ, and in turbulent flow is equal to sum of the dynamic viscosity plus the turbulent viscosity ($\mu_e = \mu + \mu_t$) (14). The k-ε model is a semi-empirical model based on the model transport equations where the turbulent kinetic energy, (k), and its dissipation rate (ε), are obtained by combination of the flow governing equations according to the experimental observations and mathematical relations (15).

$$\frac{\partial(\rho k)}{\partial t}+u_i\frac{\partial(\rho k)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right)+G-\rho\varepsilon$$

$$G=\mu_t\left[\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right]\frac{\partial u_i}{\partial x_j} \qquad (4)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+div(\rho\bar{u}\varepsilon)=\frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial}{\partial x_i}\varepsilon\right)+c_1\frac{\varepsilon}{k}G \qquad (5)$$

μ_t is obtained from expression (6):

$$\mu_t=\rho c_\mu\frac{\varepsilon^2}{k} \qquad (6)$$

4. MODEL PREPARATION

For modeling of dam break flow over a trapezoidal barrier the experimental study by Ozmen and Kocaman (2011) is utilized (10). This experiment is performed in a flume with 9m length and 0.3m width and 0.34m height. The gate is located in a distance equal to 4.65m from the beginning of the canal and the initial water depth behind the gate was 0.25m, also the bed is dry at downstream. Figure 1 shows the laboratory canal characteristics and initial conditions of the barrier.

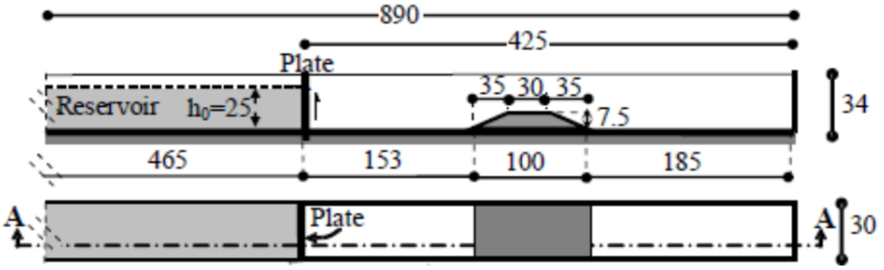


Figure 1. Section and plan of the laboratory canal and trapezoidal barrier

For geometric discretization of the problem and meshing use has been made of Gambit software. Using Gambit, the intended geometry is built and meshing is performed with arbitrary dimensions, also the boundary conditions are specified (16). In this research, applying the trial and error method, meshing is done with 1cm distance between flowlines (Figure 2).

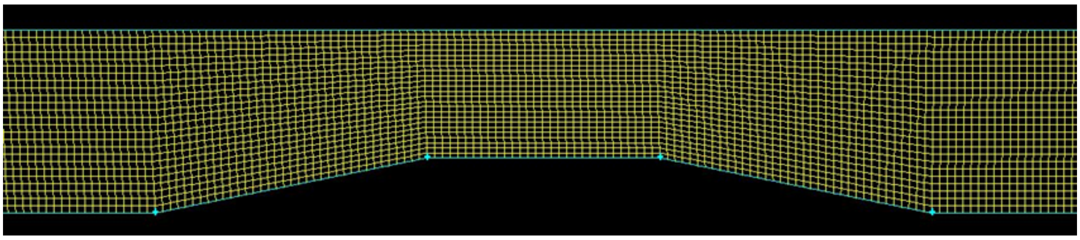


Figure 2. Computational grid around the trapezoidal barrier

The VOF method is used to determine the water level profile at the border between water and air. Parameter α in this method is taken equal to unity for water particles which occupy the intended cells and is taken equal to zero for air particles. For those cells which partially include water and air the parameter α would vary between zero and one. It should be noted that in all control volumes the sum of volume-of-fluid ratio of all existing phases is equal to unity (17). The density and kinematic viscosity of water are taken 1000 and 0.000001, respectively. For the air fluid, these values are taken 1 kg/$m^3$and 0.0000014 m^2/s, respectively. The time interval is taken in a way that the Courant number is smaller than unity for all the models (18). For discretizing the finite volume method, the

SIMPLE algorithm is incorporated. Also modeling of turbulent flow in this research is performed by k-ε model with constant coefficient according to Table 1 (15).

Table 1. Constant coefficient in the k-$^{\varepsilon}$ model

$C_{2\varepsilon}$	$C_{1\varepsilon}$	C_{μ}	σ_{ε}	σ_{k}
1.92	1.44	0.09	0.76923	1

The initial conditions of velocity is in a way that at t=0, the velocity is assumed equal to zero for all the points. The pressure in all the points is also taken zero at time t=0. By start of the solution and fall of the water column under the gravity effect, the velocity and pressure are computed at each time step by the corresponding equations. the velocity boundary conditions for the walls at both the left and right hand borders are taken zero to satisfy the no-slip condition of fluid at the vicinity of walls. The velocity boundary condition at the upper boundary is not specified and velocity is computed by the corresponding equations. In some models where the right hand boundary is defined as output, the velocity gradient value is taken zero. The pressure boundary condition in all borders is assumed zero. In addition to the velocity and pressure, there is need to assign the initial condition for parameter α, which, depending on the problem conditions, those cells which contain the fluid are designated with unity and those cells which contain air are designated with zero (19). The boundary condition for parameter α is assumed to have a zero gradient. Figure 3 shows the initial conditions for the finite volume model.

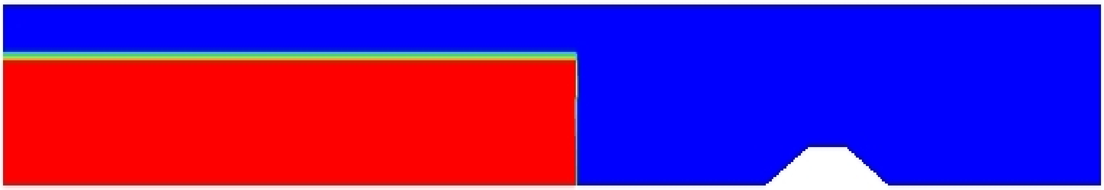
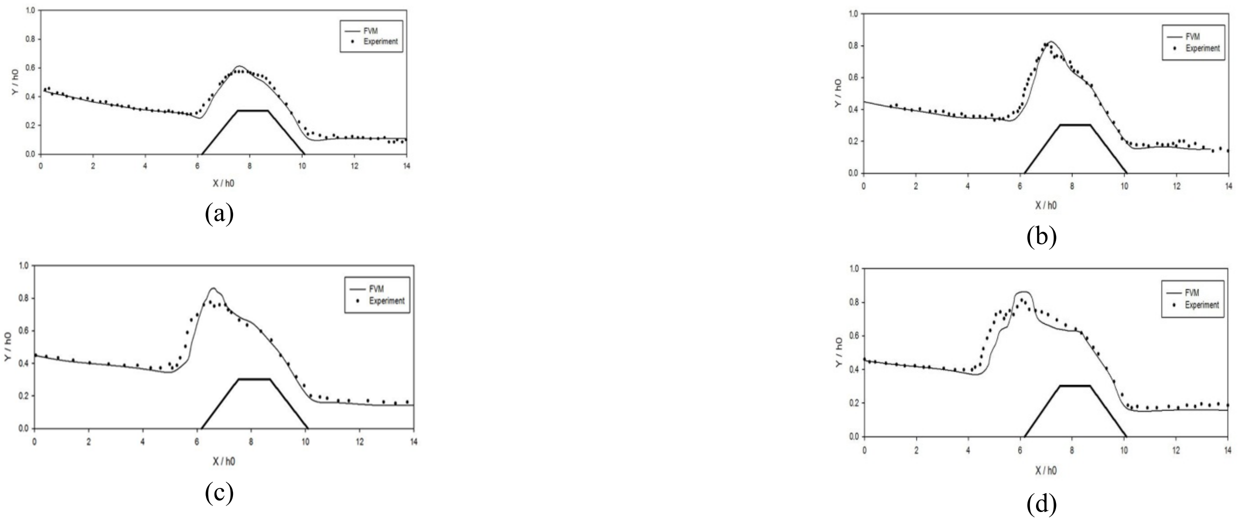


Figure 3. Initial conditions for the finite volume model

5. RESULTS AND DISCUSSION

Comparison between the experimental results and those obtained by modeling using the finite volume method is shown in Figure 4. The results demonstrate that by occurrence of dam break, the flow, after encountering the barrier, increases the specific energy and passes over it. But presence of the barrier creates a negative wave and where with passage of time moves upward and increases the flow height in front of the barrier. In fact, after passing of approximately 42 seconds, prior to encountering, the barrier accommodates itself with it and by increasing the depth passes over it. Then the flow accommodates itself with downstream slope and flows downward. Also the induced negative wave in moving upward and encountering the flow creates turbulence in the form of surface waves. The simulation results show that in the numerical model at the moment the wave moves upward a lower velocity value is observed in comparison to the experimental results. But from the non-dimensional time T=26.69 on, the results become more compatible with the experimental study results.



(a)

(b)

(c)

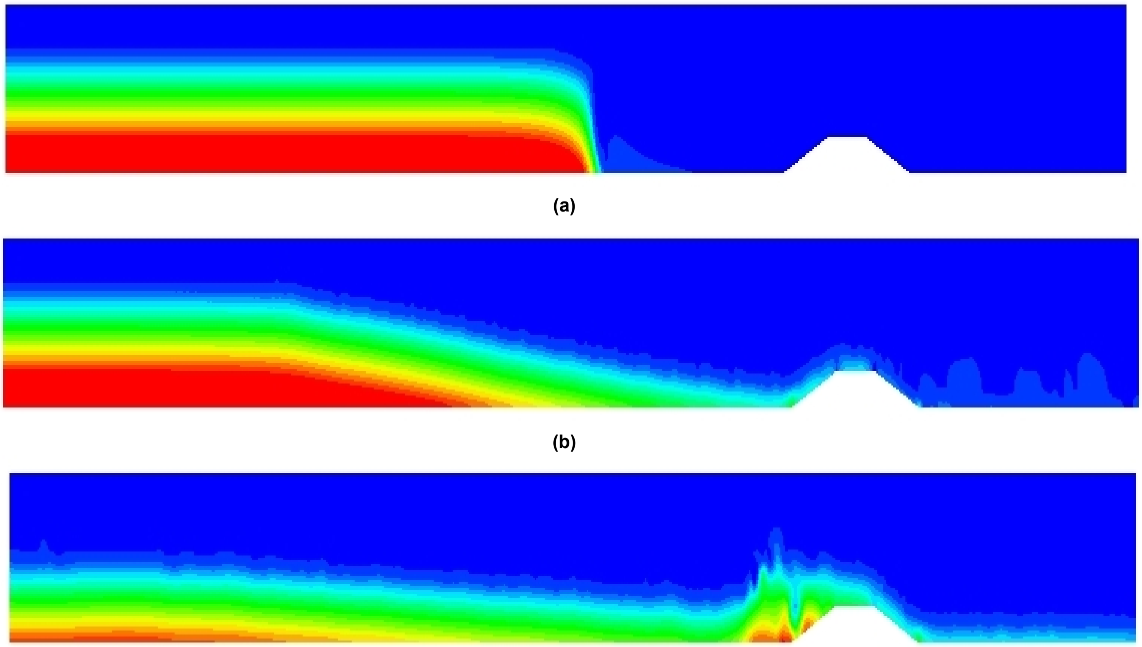
(d)

(e)

(f)

Figure 4. Comparison between the water surface profiles measured in the laboratory and computed using the finite volume method with trapezoidal barrier; a) T=11.9 s, b) T=17.54s, c) T=20.67s, d) T=23.05s, e) T=29.69s f) T=41.84s

Figure 5 shows the pressure contours at different times during the modeling process. As is seen, the pressure at the initial moments is in hydrostatic form but, with passage of time and reduction in the water column height, the pressure decreases. Also at the encounter of flow and the barrier, dynamic pressure is increased, as seen in the figure.



(a)

(b)

(c)

Figure 5. pressure at different flow points at non-dimensional times: a)T=0.63, b) T=7.5, c) T=25

6. CONCLUSION

One method for energy dissipation of flow due to dam break is creation of barrier along the wave alignment. In this research, using the OpenFOAM software and finite volume numerical methods and incorporating the k-ε turbulence model, fall of water against a trapezoidal barrier is investigated. The results showed that a portion of flow has passed over the barrier and a portion of it has not had enough energy to pass the barrier. Presence of the barrier has created a negative wave which moves upward and encounter of this returning wave with the flow creates small surface waves. By increase in the specific energy, the flow completely passes the barrier. The pressure is at first in hydrostatic form. By passage of time pressure is reduced, and at the point where the flow encounters the barrier pressure is increased in dynamic form. Comparing the experimental and numerical model results, it is found that the fine volume numerical model simulates the dam break phenomenon well against the trapezoidal barrier and the results show a good compatibility.

FUNDING/SUPPORT

Not mentioned any Funding/Support by authors.

ACKNOWLEDGMENT

Not mentioned any acknowledgment by authors.

AUTHORS CONTRIBUTION

This work was carried out in collaboration among all authors.

CONFLICT OF INTEREST

The author (s) declared no potential conflicts of interests with respect to the authorship and/or publication of this paper.

REFERENCES

1. Fraccarollo L, Toro EF. Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems. Journal of hydraulic research. 1995;33(6):843-64.
2. Soares-Frazão S, Zech Y. Experimental study of dam-break flow against an isolated obstacle. Journal of Hydraulic Research. 2007;45(sup1):27-36.
3. Lauber G, Hager WH. Experiments to dambreak wave: Horizontal channel. Journal of Hydraulic research. 1998;36(3):291-307.
4. Ferrari A, Fraccarollo L, Dumbser M, Toro E, Armanini A. Three-dimensional flow evolution after a dam break. Journal of Fluid Mechanics. 2010;663:456-77.
5. Ozmen-Cagatay H, Kocaman S. Dam-break flows during initial stage using SWE and RANS approaches. Journal of Hydraulic Research. 2010;48(5):603-11.
6. Biscarini C, Di Francesco S, Manciola P. CFD modelling approach for dam break flow studies. Hydrology and Earth System Sciences. 2010;14(4):705.
7. Bellos V, Hrissanthou V. Numerical simulation of a dam-break flood wave. European Water. 2011;33:45-53.
8. Soares-Frazão S. Experiments of dam-break wave over a triangular bottom sill. Journal of Hydraulic Research. 2007;45(sup1):19-26.
9. Marsooli R, Zhang M, Wu W, editors. Vertical and horizontal two-dimensional numerical modeling of dam-break flow over fixed beds. World Environmental and Water Resources Congress 2011: Bearing Knowledge for Sustainability; 2011.
10. Ozmen-Cagatay H, Kocaman S. Dam-break flow in the presence of obstacle: experiment and CFD simulation. Engineering Applications of Computational Fluid Mechanics. 2011;5(4):541-52.
11. Oertel M, Bung DB. Initial stage of two-dimensional dam-break waves: laboratory versus VOF. Journal of Hydraulic Research. 2012;50(1):89-97.
12. Ozmen-Cagatay H, Kocaman S. Investigation of dam-break flow over abruptly contracting channel with trapezoidal-shaped lateral obstacles. Journal of Fluids Engineering. 2012;134(8):081204.
13. Holzinger G. CD-Laboratory-Particulate Flow Modelling Johannes Kepler University, Linz, Austria. 2014.
14. Shih T-H, Liou WW, Shabbir A, Yang Z, Zhu J. A new k-ϵ eddy viscosity model for high reynolds number turbulent flows. Computers & Fluids. 1995;24(3):227-38.
15. Bremhorst K. Modified form of the kw model for predicting wall turbulence. Journal of Fluid Engineering. 1981;103:456-60.
16. Europe F. GAMBIT users’ manual. Version. 2001.
17. Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics. 1981;39(1):201-25.
18. Chang S-C. Courant number insensitive CE/SE schemes. AIAA paper. 2002;3890:2002.
19. Jasak H. OpenFOAM: open source CFD in research and industry. International Journal of Naval Architecture and Ocean Engineering. 2009;1(2):89-94.