International Journal of
Science and Engineering Investigations vol. 6, issue 61, February 2017
ISSN: 2251-8843

The Comparison of Modeling Profile of Velocity Inside Turbidity Currents Using FLOW-3D and Fluent Software

Ahmad Abdipoor[1], Ehsan Hajibabaei[2], Seyed Abbas Hosseiny[3]
[1]Ph.D., Technical and Engineering Department, Islamic Azad University, Science and Research Branch, Tehran, Iran
[2]Ph.D. Candidate, Technical and Engineering Department, Islamic Azad University, Science and Research Branch, Tehran, Iran
[3]Assistant Prof., Technical and Engineering Department, Islamic Azad University, Science and Research Branch, Tehran, Iran
([2]hajbaba2@yahoo.com)

Abstract-Turbidity currents are gravity currents in which the difference of density or weight difference between two fluids is because of suspended sediments. Considering sedimentation of turbidity currents in dams' reservoirs, hydrodynamic recognition of these currents will have great effect on increasing useful life of dams. So identifying hydrodynamic and characteristics of turbidity currents is very important in different fields. Using FLOW-3D and Fluent software velocity in the body of turbidity current has been modeled and compared to each other in this paper. The obtained results of numerical models of velocity profile in the body in inner regions (between bed toll maximum velocity) and in outer region (between the maximum velocities until the velocity gets almost zero) have relatively good compliance.

Keywords-*Turbidity Currents, Hydrodynamic, Velocity Profile, FLOW-3D, Fluent*

I. INTRODUCTION

Velocity distribution vertically in turbidity currents has important and main role in distributing sediments in the current and therefore it affects many other aspects of sedimentation [1]. Sedimentation in river bends has been considered by Ketabdar and Hamedi [2] and Ketabdar [3] experimentally and numerically respectively. Sedimentation and velocity changes may create different types of damages such as erosion, abrasion, cavitation, and some instability, which are extensively reported for marine structures [4-6]. One of measuring methods of these currents is numerical modeling, implementing and executing numerical models is very less expensive than experimental ones and their function isn't limited to only a specific sample. Such a modeling has been applied in the other fields and compensate more of the effort for experimental works like to work carried out by Darvish et al. [7-11]. Moreover, maneuverability on the channel geometry is very high in numerical models and channel can be simulated with those very real dimensions of that. As result, all characteristics of current can be simultaneously simulated in these types of models but the biggest weakness of these models is lack of considering all effective parameters on nature. In another word, only main effective factors on flows are modeled in these models. On the other hand, in numerical models, there might be errors while environment boundaries are implemented, dominant equations are prepared for solution using the method in the model as well as numerical solving of obtained equations. Each one of these errors can be reduced through some measures. However, these errors will never eliminate. Therefore, obtained answers from model, in initial using that, are better to be analyzed and controlled with experimental results to specify the weaknesses and strengths of model. Hamedi et al. [12][13] considered the energy loss experimentally then used the numerical methods to simulate the flow to show the capabilities of this kind of method. In fact, numerical method completes other methods well but it can replace them because a theory and natural affairs and experimental experiments will be always required.

In this paper, a numerical model of non-conservative turbidity currents is sought to be found using Flow 3-D and Fluent software. Numerical modeling in Flow 3-D software was conducted through VOF method and modeling in Fluent software was conducted in two phases through Euler-Lagrange method. In order to verify numerical modeling in determining hydraulic parameters of turbidity currents such as velocity profiles in the body of turbidity current, these two models were compared with each other. How turbidity currents move with numerical modeling and its body and head parts have been shown in figures 1 and 2. The movement of head and body of a permanent turbidity current has been clearly proposed in the figure and it is seen that the height of body is almost fixed but the head of current has variable shape and height along the path of channel.

II. THEORETICAL PRINCIPLES

Dominant equations on fluid current include continuity equations and size of movement. Continuity equation of current is obtained from the law of conservation of mass and through writing the equation of mass equilibrium for a fluid element. This equation is generally written as follows:

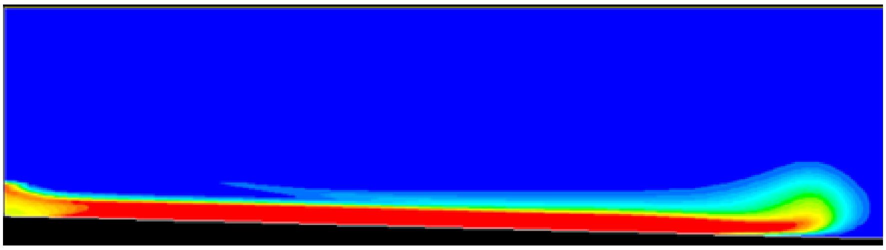

Figure 1. Schematic view of turbidity current with numerical modeling (Fluent)

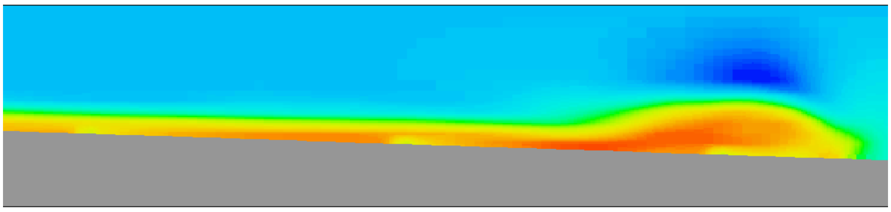

Figure 2. Schematic view of turbidity current with numerical modeling (FLOW-3D)

$$V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + R\frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) + \xi \frac{\rho u A_x}{x} = 0 \quad (1)$$

In which V_F is the ratio of fluid volume passing an element to the total volume of element and ρ is fluid density. Velocity components (u, v, w) are in directions of (x, y, z). Axis the ratio of area of passing fluid from an element to the total area of element in the direction of x, Az and Ay are similarly the levels of current in directions y and z. R and ξ are related to the type of coordinate system and Cartesian coordinates are R=1 and ξ =0. Navier-Stokes equations of fluid with velocity components (u, v, w) in 3 dimensional coordinates are shown as below:

$$\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial u}{\partial x} + vA_y R\frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x \quad (2)$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial v}{\partial x} + vA_y R\frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + G_y + f_y \quad (3)$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial w}{\partial x} + vA_y R\frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z \quad (4)$$

In these equations, (Gx, Gy and Gz) are terms of mass acceleration, (fz, fy and fx) are terms of viscose acceleration [14].

III. Initial Conditions

Numerical modeling in this paper will be conducted for a gravity current containing particles in a channel of 12-meter length and 75-centimeter height. The concentration of particles is 0.5 percent and flow rate will be about 20 liters on minute. The slope of channel is also 2 percent. Modeling was done two dimensionally. Turbidity current with fixed flow rate gets into standing water in the channel from 1 centimeter part below the valve and when it reaches the end of channel, modeling will be stopped. Then the velocity of turbidity current is calculated in middle depths of channel transversely.

Initial conditions for modeling have been shown in table 1.

Considering table 1, C_0 is initial concentration, ϱ_0 is density fluid inflow rate, U_0 is the velocity of inflow, b_0 and h_0 are respectively the width and height of input valve, g_0 is reduced gravity velocity, B_0 is input floating flux, Re_0 is Reynolds number of inflow, Ri_0 is initial Richardson number of flow and Fr0 is initial Froude number of flow.

The velocity of reduced gravity (g'_0) and input floating flux (B0) and initial Richardson number (Ri_0) are:

$$g'_0 = \frac{\rho - \rho_0}{\rho_0} g \quad (5)$$

$$B_0 = b_0 . h_0 U_0 . g'_0 \quad (6)$$

$$Ri_0 = \frac{g'_0 . h_0 . \cos\theta}{U_0^{\ 2}} = \frac{1}{Fr_0^{\ 2}} \quad (7)$$

IV. Numerical Solution of Equations Through Flow 3D Software

Boundary conditions are introduced in numerical model in software Flow 3D as below:

1- The velocity of density inflow under the valve into the channel (obtained by dividing inflow rate on input section level)

2- The concentration of density flow that is obtained from multiplying the percentage of volume concentration in sediments particles density. The density of sediments particles is 2648 kg/m3.

TABLE I. INITIAL CONDITIONS FOR NUMERICAL SIMULATIONS

RUN.NO	*Slope (%)*	C_0 *(gr/cm3)*	Q_0 *(lit/min)*	b_0 *(cm)*	h_0 *(cm)*	U_0 *(cm/s)*	g'_0 *(cm/s2)*	B_0 *(Cm4/s3)*	Re_0	Ri_0	Fr'_0
No.1	2%	0.005	20	20	1	16.67	3.0618	51.03	1618.2	0.0110	9.525

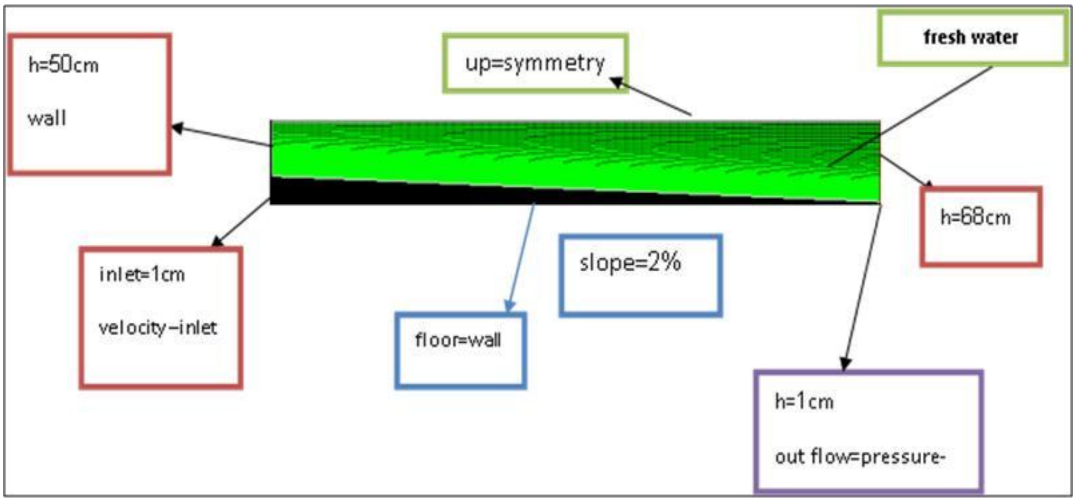

Figure 3. The geometry and boundary conditions of current

Two fluids have been used for modeling that one is standing water with density of 998.53 kg/m3 and viscosity of $1.014*10^{-1}$ and the other the flow of density which is located under stagnant water.

Considering previous experiences and investigated turbulence models for turbidity currents, in order to simulate experimental turbidity current, RNG turbulence models and scour model have been used in the current research [15].

V. NUMERICAL SOLUTION OF EQUATIONS THROUGH FLUENT SOFTWARE

Fluent is strong software in the field of CFD that has been written with programming language of C and uses all the capability and flexibility of this language. Also, Zeidi et al. [16-19] generated an Eulerian-Lagrangian approach for tracking bubble besides developing a comprehensive model inside Ansys-Fluent, which had been very helpful to set up the current geometry and mesh topology.

Continuity and Navier-Stokes equations are used in analyzing the current. In case the current is turbulent then dominant equations turn to Reynolds ones and one-equation, two-equation, five or six-equation models are used for determining eddy viscosity that user determines the type of model [20]. Fluent converts dominant equations to algebraic equations through the method of finite volumes and then solve them [21].

Considering that Fluent software doesn’t generate network itself and the network should be generated in another software, Fluent is capable of reading mesh from other software such as Gambit, Geomesh and ANSYS. In this research in first step, the shape of problem is drawn in AutoCAD software and in the second step, meshing was formed on general geometry in Gambit software [22], meshing is non-uniformly and rectangular. Rectangular mesh has been used by Baqersad et al.[23] and Fesharaki and Hamedi[24] previously. Then the type of the boundaries of solution zone such as wall, input and output were specified, generated mesh is read in Fluent software and controlled in this software based on the research work of Hamedi et al. and Hamedi and Ketabdar [25][26] in case of probable problems such as negative volumes.

Geometry and boundary conditions of current have been represented in figure 3.

VI. COMPARING THE RESULTS OF MODELING

A density flow consists of two main and important parts of head and body. The body of density flow is semi-uniform part of flow in which flow is almost permanent in that area. In another word, time and place changes of velocity in the body of flow are almost trivial. According to conducted researches in past, a velocity profile in the body consists of two inner and outer regions that are separated from each other by maximum velocity Um that is often defined as the nose of head of turbidity current. Inner region or that very gradient wall area has a positive velocity and turbulence is generated caused by floor roughness. The sediments are sucked from floor in this region and velocity distribution in this region is a logarithmic distribution. The thickness of this area is less than half thickness of external region. A longitudinal velocity profile has been proposed in figure 4. The velocity field, also, has been considered in vortex flow by Sarkardeh et al. [27].

Outer region of gradient has negative velocity that this negative gradient is because of backward flow of environmental fluid. Turbulence in this region is obtained from mixing with upper stagnant fluid. Velocity distribution in this area is a normal distribution of Gasyn. The results show that velocities near shear layer, between density and stagnant fluid are less than middle scale of flow while a part of existing suspended sediments in density flow is defused into upper stagnant fluid and mix with it and therefore cause clouding of the clear water and move with it as backward flow while it isn't a part of density flow [1].

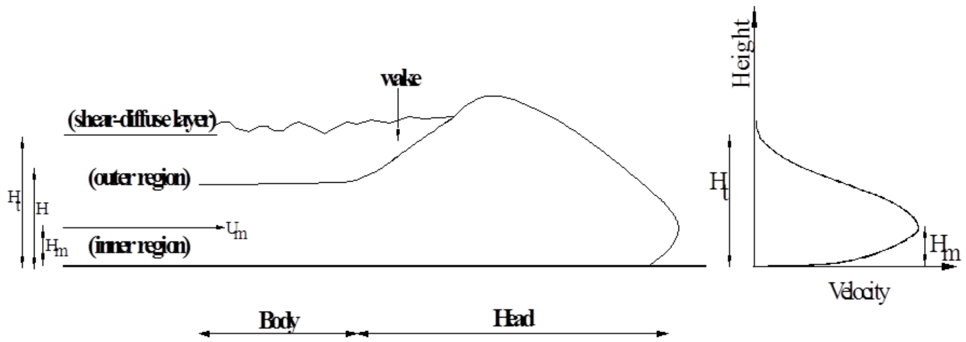

Figure 4. Longitudinal velocity type profile with specific heights in density flow [1]

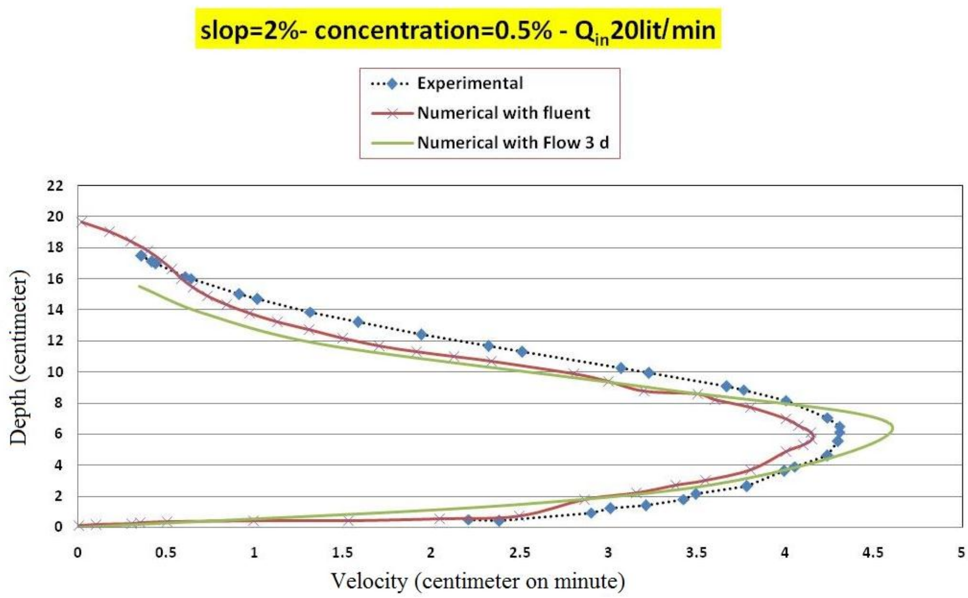

Figure 5. The comparison of numerical modeling of velocity profile in body through Fluent and Flow 3D software for slope of 2%, concentration of 0.5% and flow rate of 20 liters on minute

In all three levels of numerical modeling, density flow with specific flow rate and concentration enters into the channel from 1 centimeter below the valve, when the flow reaches to the end of channel, modeling will be stopped and velocity profiles in middle depths of channel are measured transversely and compared with each other.

To compare numerical solution, experimental results conducted by Hosseini et al that had been accomplished in

Center of Energy Pole of Sharif University were used. As it can be seen in figure above, the results of numerical modeling have good compliance with the experimental results.

Calculated velocity profiles due to central line of channel are measured to determine the average height (H) and velocity (U) of layer (depth) in each section and used by the following equations.

$$UH = \int_0^\infty \bar{u}(z)dz = \int_0^{H_t} \bar{u}(z)dz \quad (8)$$

$$U^2H = \int_0^\infty \bar{u}^2(z)dz = \int_0^{H_t} \bar{u}^2(z)dz \quad (9)$$

In which, u(z) is average longitudinal velocity of flow at the distance of z from bed and H is the height of whole flow and place where the velocity of flow gets almost zero. The values of whole flow height are H, flow maximum velocity is U_{max} and the similar height of the maximum velocity is Hm which have been shown in table 2 and compared with the results of two modeling. Furthermore, It is recommended to use artificial intelligence and compare the results with numerical simulation [28].

TABLE II. COMPARING THE RESULTS OF WHOLE FLOW HEIGHT, MAXIMUM VELOCITY AND ITS SIMILAR HEIGHT

Description	*Type of comparison*	*(Experiment)*	*(Flow 3D)*	*Error percentage*	*(FLUENT)*	*Error percentage*
S2C5Q20	Ht(cm)	21.7	19.5	-10.14	19.67	-9.35
	Umax(cm/s)	4.1	4.59	11.95	4.15	1.22
	Hm(cm)	7.7	6.65	-13.64	5.7	-25.97

VII. CONCLUSION

The velocity in the body of turbidity current has been modeled through Fluent and Flow 3D software and then compared to each other. Hydrodynamic quantities of average flow including height, velocity, maximum velocity and corresponding height with that by the help of obtained velocity profiles were calculated and compared with each other in numerical models.

Through the comparison of velocity profiles between Fluent and Flow 3D software in the body of density flows in a common experiment, relatively good compliance can be seen.

REFERENCES

[1] Hosseini. N.R, 2005. A laboratory study on the hydrodynamic flows using sonic speedometer. PhD Dissertation, Department of civil and environmental engineering, Sharif University of Technology.

[2] Ketabdar, M. Hamedi, A. 2016 Intake Angle Optimization in 90-degree Converged Bends in the Presence of Floating Wooden Debris: Experimental Development. Florida Civil Engineering Journal, 2, 22-27.

[3] Ketabdar, M. 2016. Numerical and Empirical Studies on the Hydraulic Conditions of 90 degree converged Bend with Intake. International Journal of Science and Engineering Applications, 5(9), 441-444.

[4] Champiri, M. D., Sajjadi, S., Mousavizadegan, S. H., Moodi, F., "Assessing Distress Cause and Estimating Evaluation Index for Marine Concrete Structures, American Journal of Civil Engineering and Architecture, Science and Education Publishing, 4(4), 142-152, 2016, doi: 10.12691/ajcea-4-4-5.

[5] Champiri, M. D., Mousavizadegan, S. H., Moodi, F., "A fuzzy classification system for evaluating the health condition of marine concrete structures", Journal of Advanced Concrete Technology, Vol. 10, No. 3, 95-109, http://doi.org/10.3151/jact.10.95.

[6] Sajedi, S., Huang, Q., Gandomi, A. H., Kiani, B. "Reliability-Based Multiobjective Design Optimization of Reinforced Concrete Bridges Considering Corrosion Effect". ASCE-ASME Journal of Risk and Uncertainty in Engineering Systems, Part A: Civil Engineering, 2016, 04016015.

[7] Darvish S, Sabarou H, Saxena SK, Zhong Y. Quantitative defect chemistry analysis and electronic conductivity prediction of La0.8Sr0.2MnO3±d perovskite. J Electrochem Soc 2015;162:E134e40. http://dx.doi.org/10.1149/2.0361509jes.

[8] Darvish S , Saxena SK, Zhong Y. Quantitative Analysis of (La0.8Sr0.2)0.98MnO3±δ Electronic Conductivity Using CALPHAD Approach. The 39th International Conference on Advanced Ceramics and Composites (ICACC); 2015;179-189. http://dx.doi.org/10.1002/9781119211747.ch15

[9] Darvish S, Karbasi A, Saxena SK, Zhong Y. Weight Loss Mechanism of (La0.8Sr0.2)0.98MnO3±δ During Thermal Cycles. The 39th International Conference on Advanced Ceramics and Composites (ICACC); 2015; 93-99. http://dx.doi.org/10.1002/9781119211310.ch11

[10] Darvish S, Asadikiya M, Hu B, Singh P, Zhong Y. Thermodynamic prediction of the effect of CO2 to the stability of (La0.8Sr0.2)0.98MnO3±δ system. International Journal of Hydrogen Energy. 2016;41:10239-48. http://dx.doi.org/10.1016/j.ijhydene.2016.05.063

[11] Darvish S, Gopalan S, Zhong Y. Thermodynamic Stability Maps for the La0.6Sr0.4Co0.2Fe0.8O3±δ–CO2–O2 System for Application in Solid Oxide Fuel Cells. Journal of Power Sources. 2016; 336: 351–359.http://dx.doi.org/10.1016/j.jpowsour.2016.10.004

[12] Hamedi, A., Mansoori, A., Shamsai, A., & Amirahmadian, S. 2014. The Effect of End Sill and Stepped Slope on Stepped Spillway Energy Dissipation. Journal of Water Sciences Research, 6 :1-15.

[13] Hamedi, A., Ketabdar, M., Fesharaki, M., & Mansoori, A. 2016. Nappe Flow Regime Energy Loss in Stepped Chutes Equipped with Reverse Inclined Steps: Experimental Development. Florida Civil Engineering Journal, 2:28-37.

[14] Abdipour. Ahmad, 2010. Numerical modeling of Hydrodynamic flows by Flow 3D software. M.S thesis, University Of Science and Research branch.

[15] Theory of Flow 3-D Manual Ver.8.2

[16] Zeidi, S. M. J, Mahdi, M., 2015, "Evaluation of the physical forces exerted on a spherical bubble inside the nozzle in a cavitating flow

with an Eulerian/Lagrangian approach", European journal of physics, 136(6).

[17] Zeidi, S. M. J and Mahdi M, 2015, Investigation effects of injection pressure and compressibility and nozzle entry in Diesel injector nozzle's flow J. Appl. Comp. Mech. 2 83–94.

[18] Zeidi, S. M. J and Mahdi M, 2015, "Effects of nozzle geometry and fuel characteristics on cavitation phenomena in injection nozzles", The 22st Annual International Conference on Mechanical Engineering-ISME 2014, available online at "http://www.civilica.com/EnPaper--ISME22_394.htmlhttp://www.civilica.com/EnPaper--ISME22_394.html".

[19] Zeidi, S. M. J and Mahdi M, 2015, "Investigation of viscosity effect on velocity profile and cavitation formation in Diesel injector nozzle", 8th International Conference on Internal Combustion Engines 2014, ISBN 978-600-91530, available online at http://www.civilica.com/EnPaper--ICICE08_055.html.

[20] Georgoulas,N. et al, 2010. 3D numerical modelling of turbidity currents. Laboratory of Hydraulics and Hydraulic Structures, Department of Civil Engineering Democritus University of Thrace.

[21] FLUENT 6.3.26 User's Guide (2008)

[22] Gambit 2.2, Tutorial Guide.

[23] Baqersad, M., Eslami, A., Haghighat, Rowshanzamir, M., Mortazavi Bak, H., 2016. Comparison of Coupled and Uncoupled Consolidation Equations Using Finite Element Method in Plane-Strain Condition. Civil Engineering Journal, 2: 375-388.

[24] Fesharaki, M., Hamedi, A. 2016. Effects of High-Speed Rail Substructure on Ground-Borne Vibrations. Florida Civil Engineering Journal, 2:38-47.

[25] Hamedi, A., Hajigholizadeh, M., & Mansoori, A. (2016). Flow Simulation and Energy Loss Estimation in the Nappe Flow Regime of Stepped Spillways with Inclined Steps and End Sill: A Numerical Approach. Civil Engineering Journal, 2(9), 426-437.

[26] Hamedi, A., Ketabdar, M. (2016) Energy Loss Estimation and Flow Simulation in the skimming flow Regime of Stepped Spillways with Inclined Steps and End Sill: A Numerical Model. International Journal of Science and Engineering Applications, 5(7), 399-407.

[27] H. Sarkardeh, E. Jabbari, A. R. Zarrati, S. Tavakkol, Velocity field in a reservoir in the presence of an air-core vortex, Journal of Water Management, Vol. 164, No. 4, pp. 193-200, 2013

[28] Bardestani, S., Givehchi, M., Younesi, E., Sajjadi, S., Shamshirband, S., & Petkovic, D. 2016. "Predicting turbulent flow friction coefficient using ANFIS technique." Signal, Image and Video Processing, 1-7.