Biological Forum – An International Journal **7**(1): 382-393(2015)

ISSN No. (Print): 0975-1130
ISSN No. (Online): 2249-3239

Numerical Simulation of the Flow Pattern in the Aeration Tank of Sewage Treatment System by the Activated Sludge Process Using Fluent Program

***Hazhar Hadad*[*] and Jamil Ghaderi**[**]**

[*]Department of Civil Engineering, Mahabad Branch, Islamic Azad University, Mahabad, IRAN
[**]Department of Civil Engineering, Mahabad Branch, Islamic Azad University, Mahabad, IRAN

(Corresponding author: Jamil Ghaderi)
(Received 07 January, 2015, Accepted 13 February, 2015)
(Published by Research Trend, Website: www.researchtrend.net)

ABSTRACT: One of the most important parts of the process in water or wastewater treatment plant is in the activated sludge system of the aeration unit. The analysis of the flow pattern and velocity distribution in the tank is one of the important parameters in increasing the treatment efficiency of the unit, therefore identifying the static locations and the places with high velocity can be an accurate design index of this unit. In the present study the simulation of an aerated wastewater treatment tank in terms of two-phase conditions has been evaluated for the first time. One of the three tanks is the wastewater treatment tank located in Pardis Nazlu through the networked Gambit program then the networking was measured by the fluent program. Since the water in the tank is still the simulation and velocity distribution is performed by the static flow and through aeration with a pipe with 280.2 m/h velocity. The simulation was performed at the moment of the air conditioning; the rate and the change of pressure demonstrated an insufficient aeration pipes in the system and the mixing and refining operations are not done in the best manner. The ratio of aeration to the tank area is 1 to 574000. However, if this ratio was 1 to 100, the mixing would be performed properly. It is recommended that the ratio of aeration to the tank area would be increased in the wastewater treatment system.

Keywords: Flow pattern, Tank aeration, Activated sludge, Fluent

INTRODUCTION

All the communities in daily activities produce waste substances that may be solid, liquid or gas form. Liquid waste materials are so-called sewers or sewage. Sewer is the water that is contaminated by various applications. About 99.9 percentage of sewers is water and only about 0.1 percentage of it is consist of impurities such as suspended solids substance and colloidal and solution [1].

Wastewater is not just sewage. All the water used in the home that goes down the drains or into the sewage collection system is wastewater. This includes water from baths, showers, sinks, dishwashers, washing machines and toilets. Small businesses and industries often contribute large amounts of wastewater to sewage collection systems; others operate their own wastewater treatment systems. In combined municipal sewage systems, water from storm drains is also added to the municipal wastewater stream.

Wastewater is about 99 percent water by weight and is generally referred to as influent as it enters the wastewater treatment facility. “Domestic wastewater” is wastewater that comes primarily from individuals, and does not generally include industrial or agricultural wastewater [8].

Understanding specifications of wastewater is of the basic processes in designing sewage purification systems. Without reliable data in this case, designs will be met with many problems such as increasing cost, and operation and maintenance problems and loss in reaching purification goals [2].

According to Miller’s comments, any change in characteristics of air, soil, water and food that have adverse effects on health, environment, human activities and other organisms is called pollution and any water that loses its quality for particular use, is turned into sewers. So the sewer is water which consumed and during this process, some suspended and dissolved substances entered in it [3].

Sludge can be defined as mud or soft sediment resulting from settling of solid substances exists in sewage. Also, the sludge can be assumed as mixtures of massed granular particles, which hydrodynamically exhibit such as a single particle, and observed that these masses can be present, have no contact with other masses [5].

Growing urbanization, population growth and urban and industrial development, problem as water shortage (water scarcity) occurs. To solve this problem, strategies have been contemplated for purifying existing water in environment and transform it into safe and delicious water before delivering industrial or urban drinking water networks and purifying domestic wastewater and industrial effluents before draining them into the nature. The major results of raw water treatment and waste water treatment are including producing water and treatment wastewater, sludge and byproducts [7].

The purified water for urban or industrial and wastewater of treatment plant is transferred to the acceptor water or lands and sludge prior to final disposal is refined for re-use purposes. In conventional wastewater treatment methods such as active sludge processes and stalactite filter, a high volume of initial sludge is added into secondary sediment sludge (active sludge). In active sludge process, secondary sludge is a microbial biomass which is produced by the metabolism of organic matter, and microbial product in sediment wastewater is about 50% and about 20% of this mass returned to the system and with initial sludge guided to disposal system. In Stalactite filters, the hydraulic load is less and less sludge is produced and there is no ancillary returning recycle system. Generally, high volumes of sludge with a concentration of 1 to 4 percent of solid substance are produced in treatment processes which are created one of the main problems of excretion. The reason is that the excess waste sludge is a combination of organic matter and microbial cells that may be degradable with other microorganisms [6].

Planning and designing of wastewater treatment facilities is first and effective step in the implementation of efficient Waste water treatment plants technically and economically. Obviously using systematic and efficient methods have a great impact on success of the project. Waste water treatment plants, consisting of a chain of physical, chemical and biological processes which The main purpose of guiding them is the removal of contaminants present from one side and remove the remaining organic material in the sludge treatment processes on the other hand, before unloading them into the environment. On physical filtration (mechanical) stage of wastewater, the very coarse aggregate solid particles floating or suspended existing in wastewater are removed firstly. Also isolation of sand particles, are often mineral and contaminated with fats and oils, is done. The final stage of physical purifying is related to initial settling (sedimentation) pool. In some cases, to improve efficiency of the solid-liquid phase separation, a small portion of the surplus returned sludge is returned to initial sedimentation pool or Chemical coagulant substances are injected to wastewater at this point. The ability of loading organic pollutant substances depend heavily on the amount of sludge in the aeration pool, because the amount of sludge available at aeration pool depends heavily on the performance of secondary sedimentation basin (pool) under varying hydraulic loading conditions and returned sludge [4].

As Computational Fluid Dynamics (CFD) finds application in an increasing number of scientific and engineering disciplines, the validation of the software is often assumed by the user. However, now that commercial CFD software has the capacity to solve a vast range of engineering problems, the validation material that accompanies them can only ever apply to a subset of these applications. While academic researchers may well accompany CFD simulations with experimental campaigns in order to test a new model or solution technique, industrial users often view CFD as just another design tool. The latter are required to assume that the software, if used correctly, produces results that can be relied on [9].

Wahl *et al.* (2000) compared the results of a CFD simulation against those from a previously validated piece of software called Win Flume. They found reasonably good agreement between the CFD and the much simpler analytical approach of Win Flume [10].

Faure *et al.* (2004) used a version of a commercial CFD code that was modified in such a way that the upstream and downstream boundary conditions were appropriate for use in modeling rivers and canals [11].

Sarkar and Rhodes (2004). They compared the position of the free surface profile over a weir in a flume with that predicted by a CFD simulation. For the single flow rate they considered, the agreement was good but they asserted that further work was required at higher flow rates [12].

Chen *et al.* (2002) used CFD modeling to model the turbulent free surface flow over a stepped spillway. The researchers' results generally compared well with experimental data, the position of the free surface was predicted very accurately [13]. Tokyay (2005) investigated the effect of channel bed corrugations on a hydraulic jump experimentally [14].

Esmaili *et al.* (2010) compared the values of pressure and velocity measured in laboratory with values simulated by the fluent model. The results showed that there exists good agreement between the on-the weir flow patterns measured in laboratory and simulated by the model [15].

Karim *et al.* (2009) simulated wave transformation in porous structures to calculate the hydraulic performance of a vertical porous structure. The model was developed using the VOF and two-phase flow [16].

Taylor (1944) first addressed the topic of open-channel junction flow by focusing on the depth ratio between the upstream branches and the downstream channel. The results are a momentum analysis that yields a predictive equation for the depth ratio. Taylor's paper is important for its identification of the need of theoretical description of the open-channel junction and the groundwork it formed for future investigations [17].

Weber *et al.* (2001) performed an extensive experimental stud of combining flows in 90º open channel for the purpose of providing a very broad data set comprising three velocity components, turbulence stresses, and water surface mappings [18].

Huang *et al.* (2002) provided a comprehensive numerical study of combining flows in open channel junction and investigate the effect of the junction angle on the flow characteristics [19].

A numerical model using finite-element methods was presented by Chong (2006), analysis the open channel junction and a simple comparison with experimental data for velocity profiles were done [20].

Pirzadeh and Shamloo (2007) provide application of FLUENT -2D&3D software in simulation of lateral intake flows. Comparisons have been made between numerical results and measured experimental velocities for a lateral intake [21].

Namaee *et al.,* (2014) simulated the 3 dimensional flows on a broad-crested side spillway in a numerical way. RNG K-ε model was used to simulate turbulent and fluid volume model to find free water surface. The simulation results were evaluated using experimental data and showed that existing numerical methods using RANS are useful in designing side weirs [22].

Ramamurthy *et al.* (2009) applied a $k-\varepsilon$ model to determine various flow characteristics such as velocity distribution, water surface profile and pressure distribution over a sharp-crested weir in a rectangular open channel [23].

Research purposes are:

(i) Determining the path length of air mixing with the water treatment tanks.

(ii) Providing the ability of the Fluent program in simulation of the two-phase flow.

(iii) The general study of flow patterns in aeration tanks.

MATERIALS AND METHODS

In this paper, first of all, the location of the free surface of liquid metal was tracked by solving the volume of fluid-function equation, determining computational domain of current time. Second, flow field and temperature field were simulated by using SMPLE algorithm; and temperature was got by solving energy equation. Third, add a time step. Don't stop repeating this process until the filling process finish. Specific process is illustrated in Fig. 1.

FLUENT is the CFD solver for choice for complex flow ranging from incompressible (transonic) to highly compressible (supersonic and hypersonic) flows. Providing multiple choices of solver option, combined with a convergence-enhancing multi-grid method, FLUENT delivers optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in FLUENT allows you to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution- based mesh adoption [24].

FLUENT solves governing equations sequentially using the control volume method. The governing equations are integrated over each control volume to construct discrete algebraic equations for dependent variables. These discrete equations are linearized using an implicit method. As the governing equations are nonlinear and coupled, iterations are needed to achieve a converged solution.

Conservative form of the Navier-stokes equations using the finite volume method on structured orthogonal, Cartesian coordinate's grid system. Turbulent flows can be simulated in FLUENT using the standard k-ε, large eddy simulation (LES), renormalization group (RNG), or the Reynolds-stress (RSM) closure schemes. The model optimizes computational efficiency by allowing the user to choose between various spatial (Second-order upwind, third-order, QUICK) discretization scheme [25].

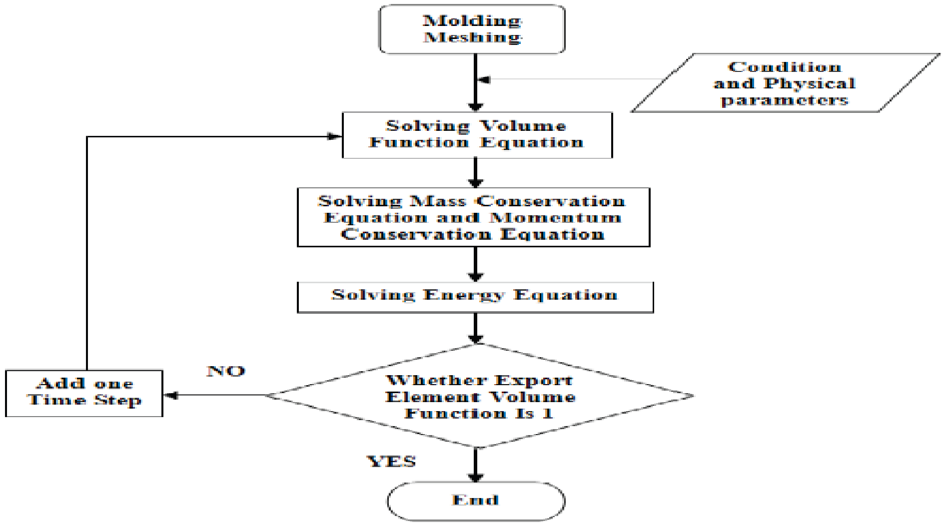

Fig. 1. Flow of Calculation.

The presence of turbulence is determined by Reynolds' dimensionless number. To obtain this value a dimensionless parameter is used which is called Reynolds' dimensionless number and flows more than 2,300 Reynolds number were considered as turbulence streams according to the environmental distribances.

$$Re = \frac{\rho.v.D}{\mu} = \frac{v.D}{\vartheta} = \frac{Q.D}{\vartheta.A} \qquad (1)$$

Where:

V is the velocity of the fluid, D is the diameter of the passage, μ is the viscosity, ϑ is the kinematic viscosity, ρ is the fluid density, Q is the volumetric flow rate and A is the area of the corssing surfa . In the computational fluid dynamics (CFD), many attempts have been done to define a comprehensive model for turbulence and the possibility of this modelling has been provided for the users in a variety of software environments.

A. Geometry production

For production or draw geometries in Gambit the point, line, page and size commands are used. First the geometry is drawn through pointing in the coordinate system and then by linking the lines the angles of the geometry is drawn. Finally a page is generated based on the area created by the sides. In order to make the geometry three dimensional the volume option is used to link the pages. Thus the geometry of the components will be as follows. After completion of the geometry of the shape will be based on the following figure:

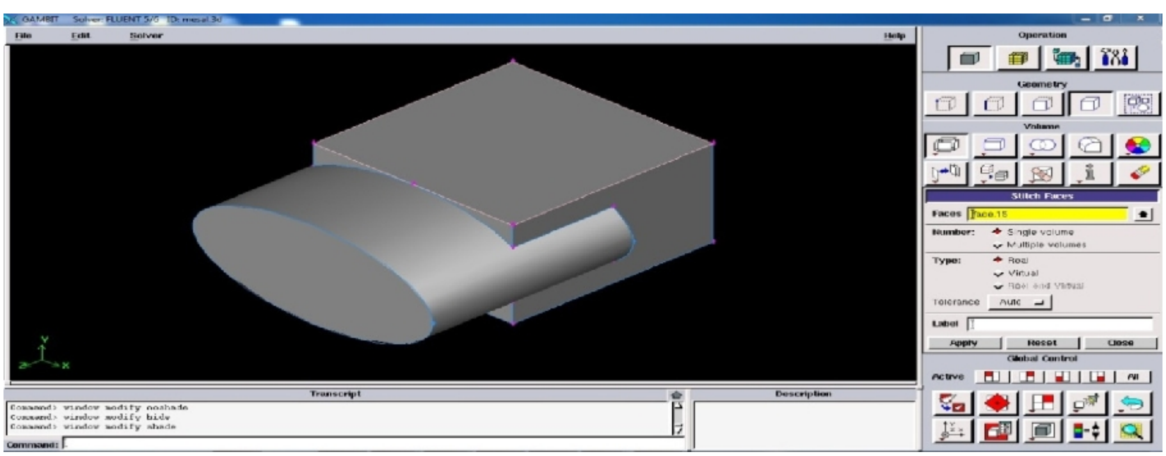

Fig. 2. The final shape of the geometry in the Gambit.

The boundary conditions include wall, the inflows, the outflows, conditioning and etc.

DISCUSSION AND RESULTS

A. The results of the FLUENT program

In fact, this application is related to the mechanics used in multi-phase flows. In this thesis the two phases of air and fluid are used. This software provides many results based on various parameters but the results mentioned in the two-phased flow of air and wastewater are as follows:

(i) The convergence iteration curve in obtaining the solution

(ii) Fluid velocity and air mixing curves and the display of the velocity vectors and speed changes on xy

(iii) The display of the viscosity vectors, viscosity mixture curves of the wastewater and air and showing their mixture on x-y.

(iv) The display of the absolute pressure, static pressure and dynamic pressure and pressure variations on x-y

(v) The display of the mixture density and air density, velocity vector through the density and the density profile curve on x-y

(vi) The curves of volume fraction of air with the fluid and the volume ratio of the air and fluis phases, the curve of volume changes, and the profile curve of volume ratio on x-y.

The curve of the convergence or divergence too obtain the optimal answer in the iteration happens when all the lines of velocity in x, y and z direction reach 0.001 which indicated that the iteration boundary is converged and the answer is optimal. In the simulation of this paper due to the weakness of the PC and the lack of capacity to handle the high volume of the program the iteration has been carried out up to 1000. The following diagrams indicate that in 1000 iteration the amount of changes was reduced which seems enough here (Fig. 3).

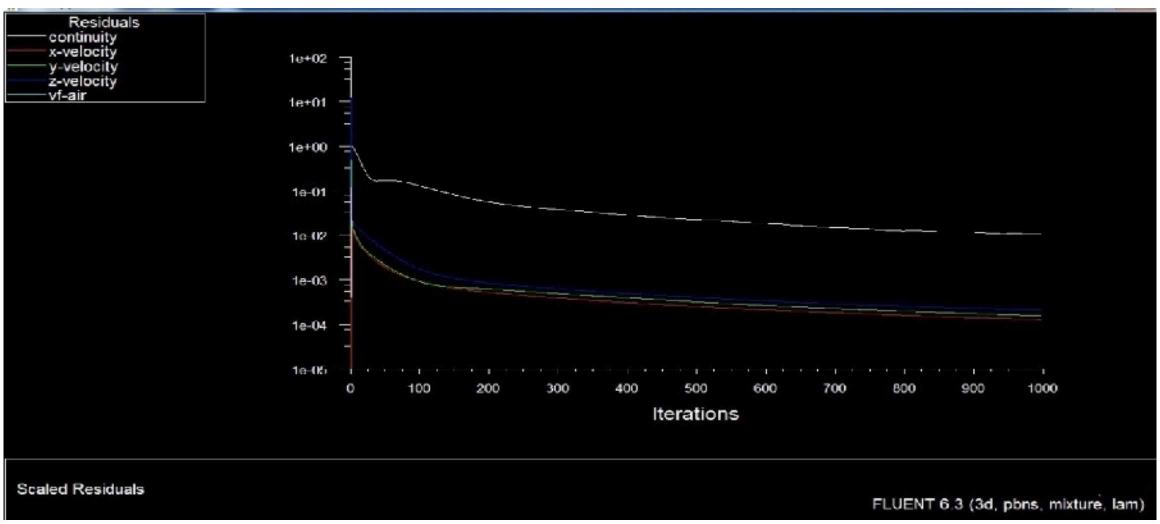

Fig. 3. Convergence iteration up to 1000.

The display of the velocity distribution in the aeration tank is presented in Figure (4). Since the tank is full and wall are introduced in the Fluent in the inlet and the outlet, therefore the velocity of water or wastewater is static but the factor that cause the mixture is the air that can not be displayed in the 3D tank.

Considering the lack of the observation of the velocity in the 3D version in the static flow Figure (4)shows the flat display of the middle of the tank in which the changes happen.

In Figure (5) resulting from the flat view of the tank it is observed that the air injection rate in the inlet is maximum which equals $2.63e^{+2}$m/s. Due to injecting air in the fluid the velocity is gradually reduced. If there are numerous aeration pipes the time of presence of the wastewater in the tank becomes minimum. In this thesis, the number of aeration pipes is more than 1 but unfortunately the computer was incapable of handling more than 1 pipeline and the research is limited to only 1 pipe.

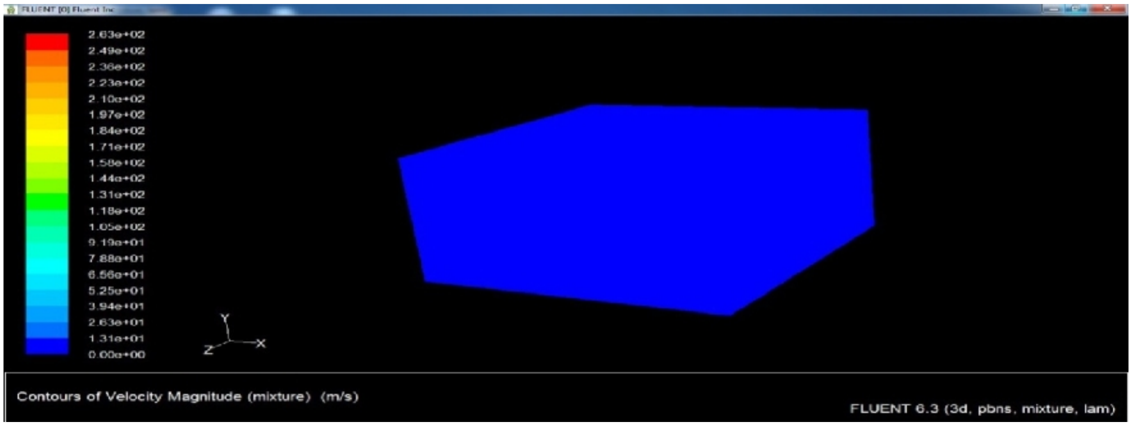

Fig. 4. The velocity distribution in the tank.

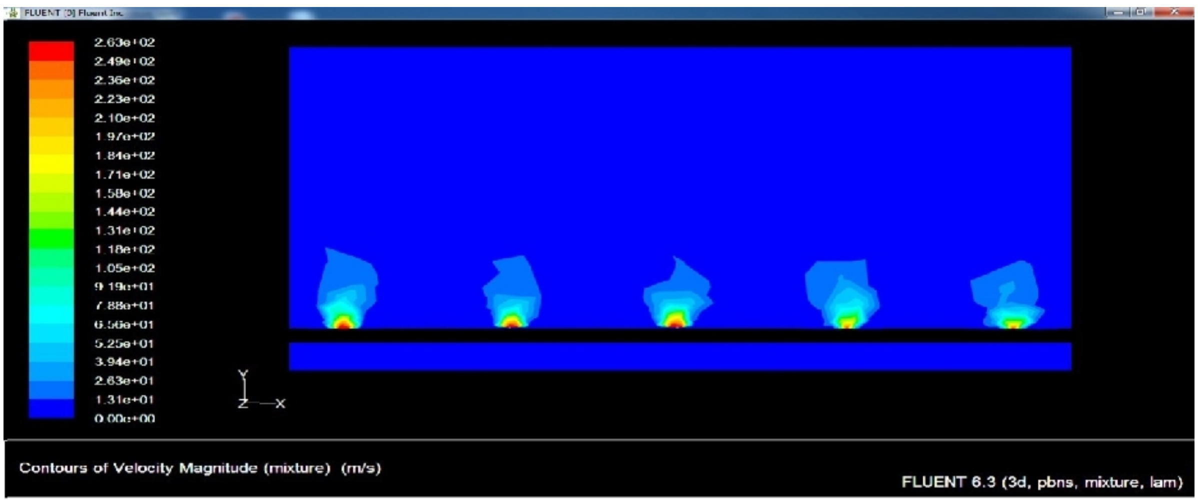

Fig. 5. The velocity distribution in the Tank.

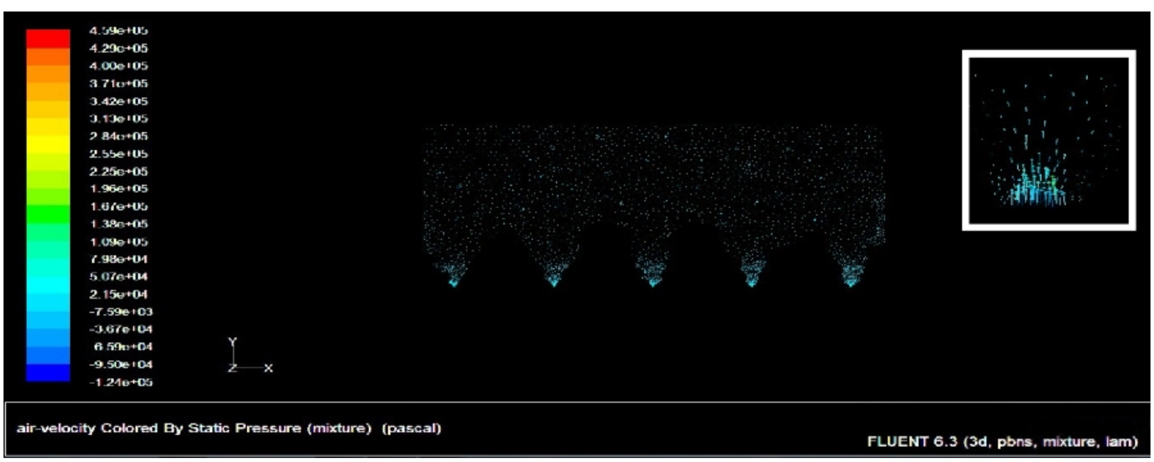

Fig. 6. The static distribution of air into the tank.

Fig. 6 displays the static air velocity vector. The velocity vectors are extended from the exhaust point that is considered for a certain width range. In this figure the static pressure is the aeration hole is $5.07e^{+4}$pa.

Fig. 7 displays the velocity distribution vector. The magnification of it is presented in the upper side of the figure. At the beginning of the air outlet in the holes of the aeration pipe it is shown as the compact vector with the maximum velocity.

The profile of the velocity vector is shown in x-y direction in Fig. 8 which consists of two diagrams. The upper diagram is the flat diagram of the tank which presents the aeration velocity in each aeration hole that passes through the aeration pipe for the iteration 1000. The velocity in the inlet which is located at the center of the coordinate system is higher and it is reduced gradually in the holes at the end of the pipe. The lower diagram presents the velocity in the whole tank. In this diagram it is observed that the velocity around the walls and above the tank is zero which confirms the problem. The velocity is $2.5e^{+2}$m/s around the aeration holes.

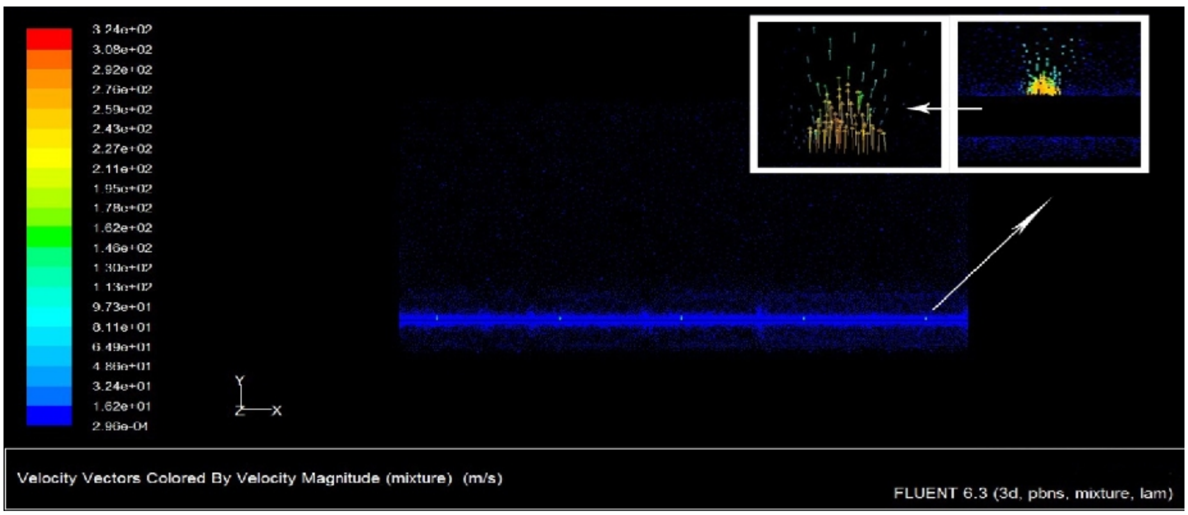

Fig. 7. The velocity vector distribution.

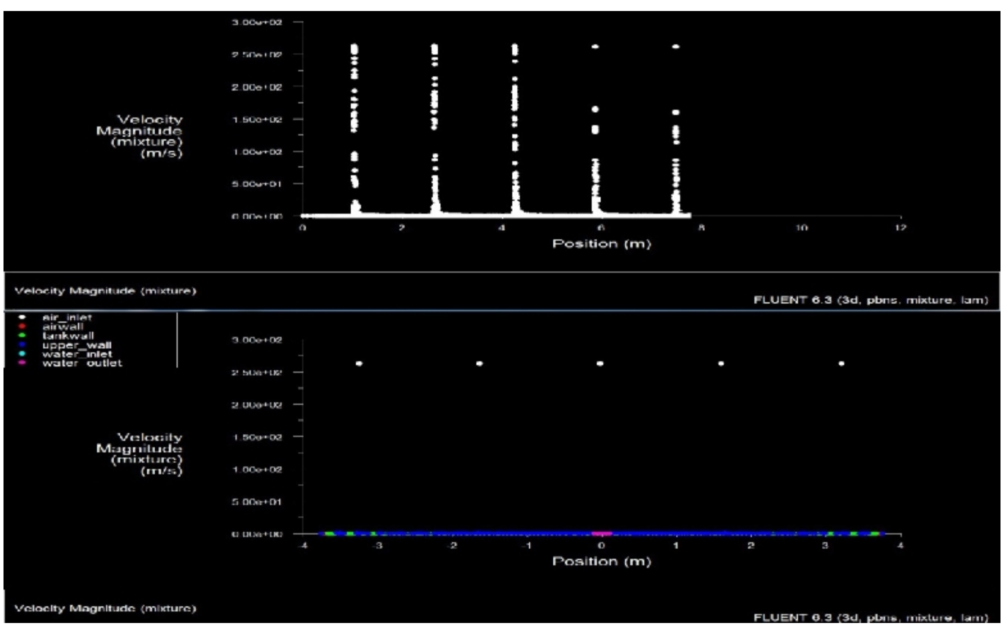

Fig. 8. The velocity profile in the aeration and tank.

Fig. 9 shows the absolute pressure. According to the figure the pressure is higher around the aeriation holes. The minimum pressure distribution in the aeration holes is $1.01e^{+5}$paand the maximum pressure distribution in the aeration holes is $2.32e^{+5}$pa. Wherever the location and environment do not interfere the pressure, the pressure is the absolute pressure.

The static pressure in Fig. 10 indicates that where the aeration is higher the static pressure is lower and where the aeration becomes lower due to the waste the static pressure will be higher. In this figure, it is observed that the maximum static pressure in the first aeration holes is $3.79e^{+4}$pa and $1.84e^{+5}$pain the last whole which indicates the maximum static pressure.

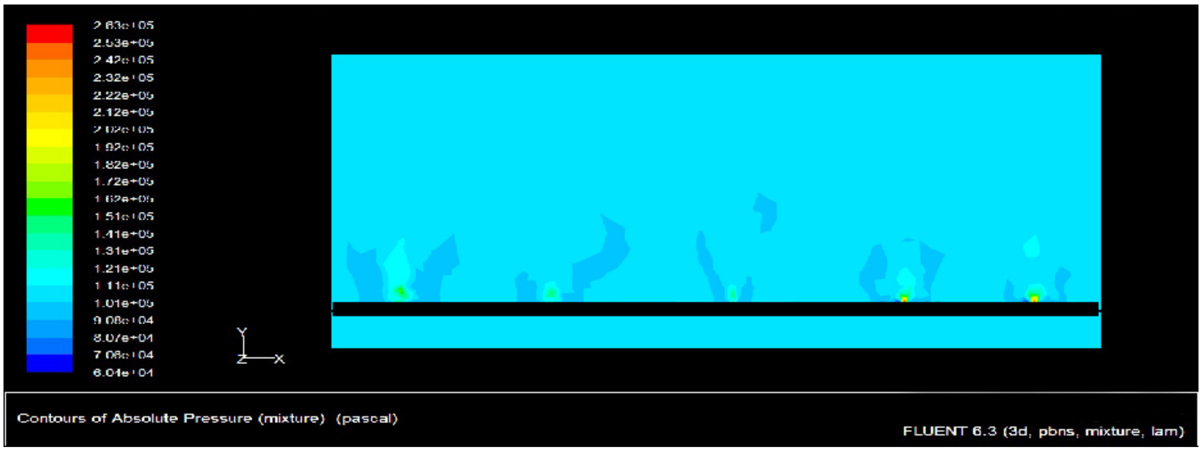

Fig. 9. The absolute pressure.

Fig. 10. The static pressure.

The dynamic pressure is the amount of drop within the path presented in Figure (11). Wherever the movement of the air or the air velocity is high the dynamic pressure is high and wherever the movement of the air or the air velocity is low the dynamic pressure is low as well. It has been observed that the dynamic pressure at the beginning of the pipe is higher than its end and covers a wider space in the tank. The highest dynamic pressure is at the beginning of the air injection which equals $4.73e^{+5}$pa after mixing the two phases of air and water it is reduced around the aeration pipe and becomes$4.98e^{+4}$pa. Finally the dynamic pressure around the walls and above the tank becomes$4.09e^{+4}$pa.

The static pressure is obtained by the dynamic pressure based on the following equation:

Dynamic pressure $= \frac{v^2}{2g} + \mathrm{P}$ (2)

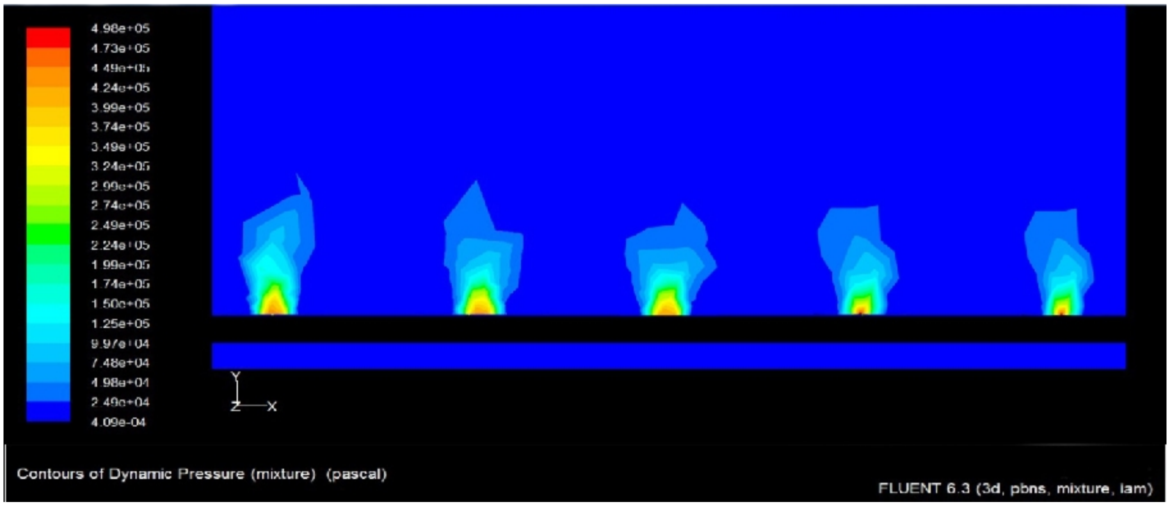

Fig. 11. The dynamic pressure.

Dynamic pressure vector variations in the x-y direction are presented in Figure (12) which includes two shear section pressure and the total tank pressure. It can be observed that in the section of aeration pipe in the first diagram the dynamic pressure is higher at the beginning of the pipe compared to the end.

The second diagram presents the dynamic pressure of the tank which indicates that the wall, inlet and the outlet pressure is zero considering the wall in the boundary conditions but there is dynamic pressure in the pipes and aeration holes the value of which is presented. Obviously, the pressure in the aeration inlet is higher than the outlet.

Fig. 13 analyzes the aeration tank in the two phases of water and wastewater. Considering the results obtained from this figure the density is lower than the tank at the air inlet and fluctuate between $6.06e^{+2} kg/m$ and $1.20e^{+3} kg/m$. It is concluded from this figure that the number of holes is less than the ideal condition because the air- wastewater mixing is not done optimally and this mixture does not affect the tank and it has acted as a mass.

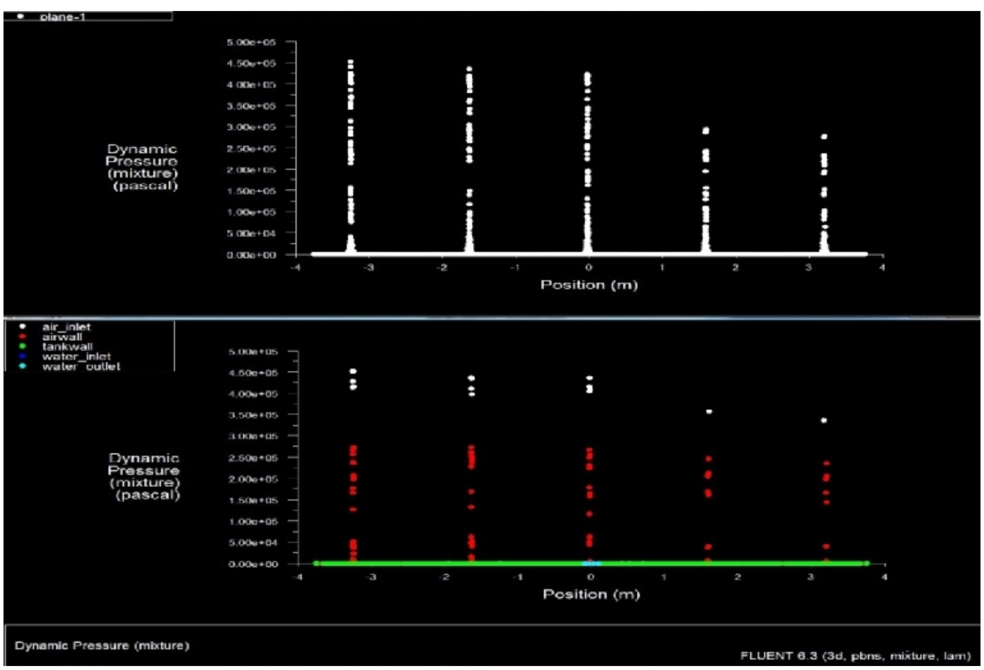

Fig. 12. Dynamic pressure at the aeration and the whole tank in x-y.

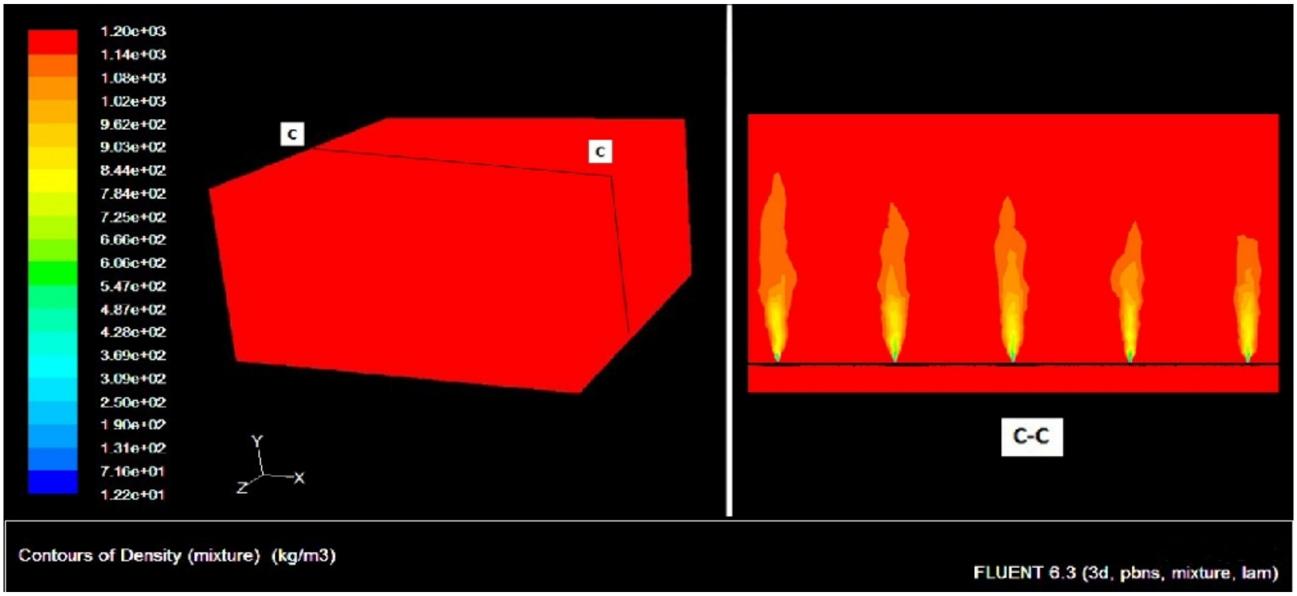

Fig. 13. The density of air and wastewater mixture.

Fig. 14 displays the density of air. It is observed that the mixture is not accomplished and due to the lack of the aeration pipes in various areas this result seems rational. According to the figure the density of blue, green and yellow colors is equal everywhere the value of which is $1.23e^{+00}\,kg/m$ and they vary in small decimal digits which depends on the accuracy of the calculation and the lack of aeration pipes. The profile density is presented in Fig. 15, the first diagram shows the air density curve in the mixing mode in the B-B section and the second diagram shows the density curve in the whole tank.

It can be seen that the density of the sides of the tank and the walls is $1.20e^{+3}\,kg/m^3$ which is consistent with Fig. 15.

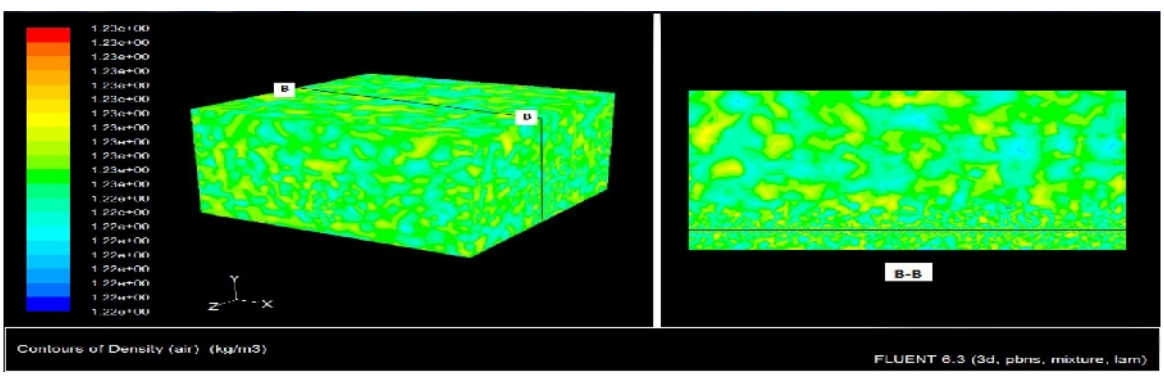

Fig. 14. The density of the air in the tank.

SUMMARY OF THE RESULTS

(i) Fluent program has a specific ability of simulation in the static flow in a closed structure in two different (biphasic) fluids.

(ii) The changes of velocity in the two environments of aeration and wastewater through the aeration pipes are presented in the form of graphics within the tank.

(iii) Due to the lack of aeration pipes and the holes related to the aeration the system does not have the capacity of optimal mixing with the air.

(iv) At the level of air- wastewater mixture, generally due to the lightness of the air in the whole system it can be observed in the long term but a longer time should be considered.

(v) The Fluent program is capable of presenting the changes in velocity and pressure at different times and any section of the structure.
(vi) Due to the lack of access to accurate information of the treatment plants and their lack of cooperation generally simulation of the wastewater system is a demanding process but the Fluent program in case of having access to computers with a powerful processor can perform the simulation at different forms and velocities.

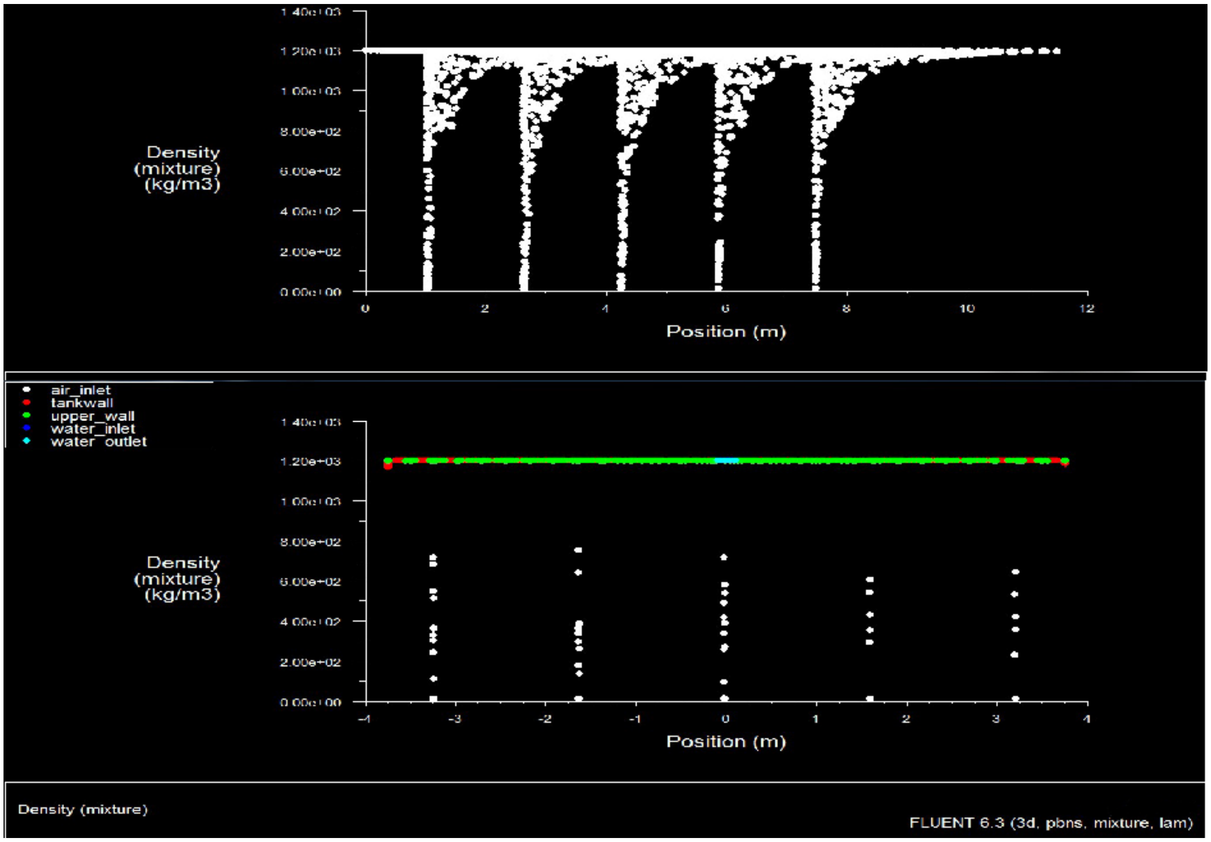

Fig. 15. The air density profile curve in x-y.

REFERENCES

Nadafi. K, (2011). "Wastewater Treatment", construction and training organization, the Ministry of Energy, Iran.

Afuoni. M, Erfanmanesh. M, (2012). "Environmental Pollution" Arkaneh Danesh Publications, Isfahan, Iran.

Tovrovski, E.S. Maathai, P.K. , (2010). "Treatment of Sewage Sludge", Mousavi. GR (Translator), Hafeez Publications, Iran.

Bedelians Golikandi. G, (2012), "Wastewater Treatment Plant Designing", Aeezh Publication, Tehran, Iran.

Metkaf and Eddy, (2011), "Wastewater Engineering", Markazeh Nashre Daneshgahi Publishing, Design and Research Consulting Engineers Company of Isfahan Water and Wastewater, Iran.

Gourjar, B.R. (2012), "Sludge Treatment and Disposal", Mokhtarzadeh Azar.R (Translator), Hafeez Publishing, Rural Water and Wastewater Company of West Azerbaijan Province, Iran.

Monzavi.MT, (2011), "Municipal Wastewater", Tehran University Publications, Iran.

Sarafraz, S., Khani M.M. and Yaghmaeian K. (2007). Quality and quantity survey of hospital wastewaters in Hormozgan province. *Iran. J. Environ. Health. Sci. Eng.*, **4**(1): 43-50.

Youngs DL (1982). Time-dependent multi-material flow with large fluid distortion. Num. Meth. for Fluid Dynamics. Academic Press.

Wahl TL, Repogle JA, Wahlin BT and Higgs JA (2000). New developments in design and application of long-throated flumes. *Joint Conference on Water Resources Engineering and Water Resources Planning and Management. Minneapolis*.

Faure J.-B, Buil N and Gay B (2004). 3-D modelling of unsteady free-surface flow in open channel. *J. Hydraulic Res.* **42**(3): 263–272.

Sarkar MA and Rhodes DG (2004). Calculation of free-surface profile over a rectangular broad-crested weir. *Flow Measurement and Instrumentation*, **15**: 215–219.

Chen Q, Dai G and Liu H (2002). Volume of fluid model for turbulence numerical simulation of stepped spillway overflow. *J. Hydraul. Eng.* **128**(7): 683–688.

Tokyay, N.D., “Effect of Channel Bed Corrugations on Hydraulic Jumps”, Impacts of Global Climate Change Conference ,EWRI, 15-19 May, Anchorage, Alaska, USA, 2005.

Esmaili, K., Naghavi, B., KouroshVahid, F. and Yazdi, J. (2010). “Experimental and numerical modeling of flow pattern on circular weir”, *Journal of Water and Soil*, **24**(1): 166-179.

Karim, M.F., Tanimoto, K., Hieu, P.D., (2009). Modelling and simulation of wave transformation in porous structures using VOF based two-phase flow model. *Appl. Math. Model.*, **33**: 343–360.

Taylor, E.H. (1944). "Flow characteristics at rectangular open-channel junctions", *Transactions, ASCE*, Vol. **109**, pp. 893–902.

Weber, L. J., Schumate, E. D., and Mawer, N. (2001). "Experiments on flow at a 90◦ open-channel junction.", *Journal of Hydraulic Engineering, ASCE*, Vol. **127**, no. 5, pp. 340-350.

Huang, J., Weber, L.J., Lai, Y.G. (2002), "Three dimensional numerical study of flows in open channel junctions.", *Journal of Hydraulic Engineering. ASCE*, Vol. **128**, no.3, pp.268-280.

Chong, N.B. (2006), "Numerical simulation of supercritical flow in open channel." M.S.C thesis, University Technology Malaysia.

Pirzadeh, B. and Shamloo, H. (2007). "Numerical investigation of velocity field in dividing open channel flow." *Proceedings of the 12th WSEAS International Conference on applied mathematics, Cairo, Egypt*, December 29-31, 2007, pp.194-198.

Namaee MR, Rostami M, Jalaledini S, Habibi M. A 3-Dimensional numerical simulation of flow over a broad-crested side weir. Advances in Hydroinformatics 2014; pp: 511-523.

Ramamurthy J., Tadayon A. S. R., Chen Z., 2009: Numerical simulation of sharp-crested weir flows. *Canadian J. of Civil Engg*., **36**, 9, 1530–1534.

FLUENT user’s guide manual-version 6.3., Fluent Incorporated, Lebanon, N.H., 2013.

Neary, V.S. and Odgaard, A.J. (1993), "Three dimensional flow structure at open-channel diversions. ", *Journal of Hydraulic Engineering. ASCE*, Vol.**119**, no.11, pp.1223- 1230.