Journal of Applied Science and Agriculture, 9(3) March 2014, Pages: 1153-1160

AENSI Journals

Journal of Applied Science and Agriculture

ISSN 1816-9112

Journal home page: www.aensiweb.com/jasa/index.html

Simulating the Flow in T-Shaped Elbow Intake Station Using Flow-3D

[1]Roozbeh Aghamajidi, [2]Gh. Beygipoor, [3]Sajjad Amiri

[1]*Young Researchers and Elite Club, Sepidan Branch, Islamic Azad University, Sepidan, Iran.*
[2]*Water Engineering Department, Islamic Azad University, Bandarabbas Branch, Bandarabbas, Iran.*
[3]*Department of Water Science, Shooshtar Branch, Islamic Azad University, Shooshtar, Iran.*

ARTICLE INFO

Article history:
Received 19 December 2013
Received in revised form 20 February 2014
Accepted 30 March 2014
Available online 8 April 2014

Keywords:
Elbow Channel Flow-3D
Flow Pattern T-shaped intake
Channel of Sabily

ABSTRACT

Checking flow pattern in intake is one type of control that should be taken into account during the designing phase. This study attempts to simulate flow pattern by Flow-3D model in lateral T-shaped intake of a sabily pump station which branches from Dez River. Considering the issues of intake in Dez river, this study investigates how to enable using elbow channel. Based on that, velocity profile distribution together with turbulence energy in intake`s opening was investigated with and without the elbow channel. Result showed that in case of no elbow channel, turbulence energy variations from the beginning of intake`s opening until its end was increased about 60% and turbulence energy dissipation was also increased by 31% from the beginning of intake`s opening (in the river) till its end; while quantitatively, maximum velocity gradient in reduced from 61% with elbow channel to 42% without one.

To Cite This Article: Roozbeh Aghamajidi, 2Gh. Beygipoor, 3Sajjad Amiri., Simulating the Flow in T-Shaped Elbow Intake Station Using Flow-3D. ***J. Appl. Sci. & Agric.****, 9(3): 1153-1160, 2014*

INTRODUCTION

In general, intake from rivers for different objectives is done in two ways of Gravitational and Pumping. Although some principles of design based on researches has eliminated this problem, some pumping stations and intake are exposed to lateral erosion and their facilities are exposed to destruction. Moreover many more are facing the sedimentation problem and their suction nozzle is invaded by sediments (Shamloo, 2011). Water deviation from rivers in one the traditional human activities, however standard and particular instruction for water deviation from the river has not been developed. The shape of Hydraulic structures is determined by the local characteristics and applying some local conditions like soil characteristics, foundation and building materials is also guided by different solutions (Neary, 1999). Water deviation from water like any other river engineering operations need various technologies and associations, thus sciences are involved including hydrology and hydraulics, soil and structure mechanics, concrete technology and etc. Location of the sabily pump station in upper side of tuning dam, Dez in Dezful section of Khuzestan is designed in a way that required water for pumping to its main embedded channel is provided by a wide channel with winged trapezoidal cross-section (in order to facilitate dredging (Barkdoll, 1997). Width of this channel is about 386m. design level of the intake channel`s floor for transferring water from is 126 and level of the river bottom is about 123 to 124. Investigating the flow pattern of initial water inflow in the transfer channel to determine hydraulics parameters in intake channel`s cross section in the primary goal of this study. (Safarzadeh *et al* 2009) modeled the 3D hydrodynamic structure of in lateral intake from the river bend in two phases and showed that the comparison of modeling results together with expermental results and verifying the numeric model significantly help analyzing fluid motion pattern in different levels, especially near bed level. (Kuroush Vahid *et al* 2010) discussed experimental formation of water surface profiles in a novel intake method by bottom intake with porous environment and showed that flow profile uniformity in low flow is less that uniformity in high flow. In case of low flow due to low velocity on porous surface, more flow is passed through. (Moazen and Shafaie Bejestan 2007) studied irrigation efficiency of vortex tube at the opening of the channels and concluded that increasing flow drop number causes reduction in water loss from the vortex tube. Moreover increasing the drop number, first increases the tube irrigation throughput and then decreases this procedure. (Sajadi and Shafaei Bejestan 2009) evaluated the precipitation irrigation tanks design models and shows that some empirical models are much faulty due to the flow pattern not matching these models in practice. (Keshavarz *et al* 2009) performed 3D numeric simulation of flow pattern surrounding perpendicular to the shore and oblique breakwaters considering different border conditions and based on their result in which free surface modeling is used assuming rigid lid is

Corresponding Author: Ruzbeh Aghamajidi, Assistant professor, department, Islamic azad university (I. A. U), Sepidan Branch. Iran.
E-mail: roozbeh1381@yahoo.com

used, computation time is decreased significantly. However results are not obtained using a logical model and in comparison to empirical results bring many errors. Although logical results were obtained using fluid volume method for aforementioned surface in comparison to experimental results, so that predicted numeric model error for vortex width concerning perpendicular to the shore breakwater was about 0.9% and this amount of error was about 7% in numeric modeling of other researchers. (Shamloo *et al* 2011) studied the effects of geometrical and hydraulic parameters on separation zone`s dimensions in a lateral intake using Fluent and concluded that choosing an appropriate turbulent model with flow conditions is very important and investigated effects of different geometrical and hydraulic parameters on the ratio of branch channel`s width to main channel`s width, flow deviation angle, flow drop number and flow distribution to separation zone`s dimensions ratio in the junction showed that in all cases, increasing the flow distribution ratio, reduces the separation zone`s dimensions in the junction and if conditions at the both end is considered so that the amount of flow distribution ratio is stable, separation zone`s dimensions in the junction will not depend on inflow conditions and it will be the ratio of two channel width to flow deviation angle. (Ghodsian *et al* 2011) experimented the effects of relative curvature of the channel on bed scour in a 90 degree curve and concluded that in most cases, location of the maximum depth of scour with increased flow and flow drop number and in time is transferred to lower levels. Location of the maximum depth of scour is at about 60-90 degrees of the beginning of the curve and in most cases at about 80-90 degrees. (Pirestani *et al* 2009) analyzed the lateral intake flow in arch channel using a physical model and concluded that regarding the linear velocity and water surface variations, increasing the linear velocity in the opening of the intake, reduces the water surface and vice versa, which is verified by the special energy equation and variable location flow. Salehi (Neishaburi *et al* 2005) studied numeric modeling of 3D flow pattern in lateral intakes and concluded that regarding a 3D flow pattern in a lateral intake located in an open channel using turbulent model k-ε, this model determines the velocity profile along main channel accurately, but predicts the developed vortex zone inside the intake poorly and the velocity profiles only match the expermintal values up to a low distance from the intake opening and the determined vortex width is obtained less that the experimental value.

METHODS AND MATERIALS

As mentioned earlier, the goal of this study is to investigate and simulate the flow pattern in elbow T-shaped intake channel of Sabily pump station, thus for this study purposes, mathematical model Flow-3D has been used. This model is multi aspect and consistent with complex flow conditions in 2D and 3D simulations. Equation solution methods in this model are based on limited volume which solves flow equations including continuity and momentum equations and other equations (e.g. Diffusion equation). Flow continuity equation is acquired from mass conservation law and mass balance equation for a fluid element. General form of this equation is as follows:

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

In this equation, Vf is a fraction of open flow amount along x. Ax, Ay and Az are similarly a fraction of open flow surface along x, y and z. Moreover R and ξ coefficient are coordinate system operators. While coordinate system is Cartesian, R is equal to one and ξ equals to zero. The first term on the right of equation (2) is the turbulent diffusion term and is calculated as follows:

$$R_{DIF} = \frac{\partial}{\partial x}(\nu_p A_x \frac{\partial \rho}{\partial x}) + R\frac{\partial}{\partial y}(\nu_p A_y R\frac{\partial \rho}{\partial y}) + \frac{\partial}{\partial z}(\nu_p A_z \frac{\partial \rho}{\partial z}) + \xi \frac{\rho \nu_p A_x}{x} \quad (2)$$

In (2) ν_ρ coefficient is equal to $c_\rho \mu / \rho$ which μ is the momentum diffusion coefficient and c_ρ is a coefficient depending on turbulent Schmit number. This mass diffusion term is related to evaluating turbulent mixing stages and is used only in case of functional fluid density being ununiformed. Last term in (2) is related to source and can be uses in case of two-phased fluids. Navier-Stockes equations of motion in a fluid with velocity components (u,v,w) in 3D coordinates is another case of flow solution equations that is shown in the following form:

$$\frac{\partial u}{\partial t} + \frac{1}{V_f}\left[uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z}\right] = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x \quad (3)$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_f}\left[uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z}\right] = -\frac{1}{\rho}\frac{\partial p}{\partial y} + G_y + f_y \quad (4)$$

Journal of Applied Science and Agriculture, 9(3) March 2014, Pages: 1153-1160

$$\frac{\partial w}{\partial t}+\frac{1}{V_f}\left[uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right]=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_z \qquad (5)$$

In these equations, (Gx, Gy, Gz) are mass accelerations and (fx, fy, fz) are viscosity accelerations. Thus in this research, intake of Sabily pump station is simulated based on aforementioned equations. Figure (1) represents a view of Sabily pump station.

Fig. 1: a view of Sabily pump station

As we can observe from figure (1), Sabily pump station intake is provided water in a T-junction from Dez River. In current conditions, this intake has many issues due to the sediments at its inlet. Thus organizing this intake based on constructing an elbow channel at its inlet. This study simulated flow pattern in two cases. In the first case, T-shaped channel is modeled without the elbow channel and in the second, it is simulated using an elbow channel. Figure (2) represents the schematics of an elbow channel.

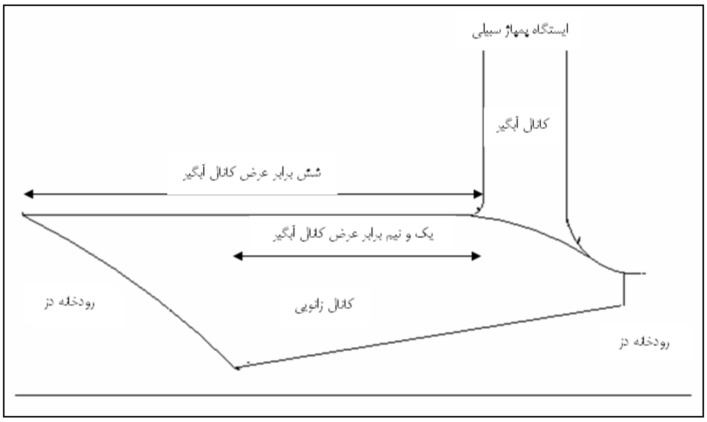

Fig. 2: elbow channel schema in Sabily pump station intake and Dez river

Discussion and Results:

This study researches the effect of elbow channel on flow pattern in Sabily pump station. Therefore both cases of intakes with and without elbow channel is considered. Figure(3) shows the simulation results for the case of no elbow channel:

Journal of Applied Science and Agriculture, 9(3) March 2014, Pages: 1153-1160

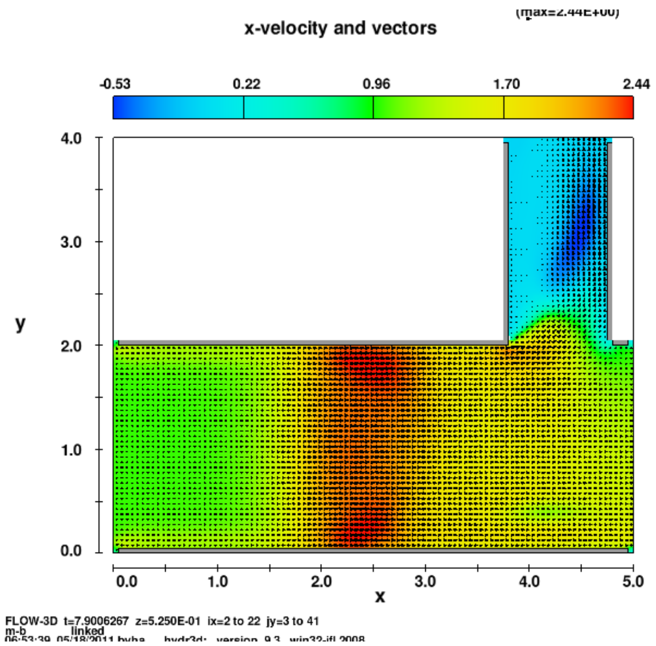

Fig. 3: flow pattern in case of not using an elbow channel

As we can observe in figure (3), the points in the intake channel`s location have the least flow which cause sedimentation in the T-shaped channel leading to Sabily pump station. To further discuss this, deep velocity profile was drawn along flow at the points close to the T-shaped intake channel, the data in which is represented in figure(4). Moreover the average hydraulic parameters including velocity, turbulent velocity and turbulent dissipation at the intake inlet is show in table (1).

Table 1: hydraulic characteristics of river flow at the intake inlet without the elbow channel

X	U_{ave}	Tke	dtke
3.825	1.961101	0.010738	0.01511
3.925	1.931423	0.011474	0.015676
4.025	1.88628	0.011967	0.015685
4.125	1.805836	0.012202	0.015182
4.225	1.67497	0.012274	0.014411
4.325	1.475544	0.014463	0.015948
4.425	1.168398	0.015949	0.017628
4.525	0.928202	0.017109	0.018251
4.625	0.740048	0.018304	0.017625
4.725	0.606701	0.023044	0.020984
4.825	0.689537	0.025023	0.022173

In table (1), location is along the river at the intake inlet, Uave is the average velocity at each point X (X=3.825 riverside at the intake channel`s inlet is in Flow-3D`s defined geometry), tke is the turbulent kinetic energy and dtke is the turbulent kinetic energy dissipation. Turbulent energy includes turbulent transference equation in case of kinetic energy together with turbulent velocity fluctuations of the flow in which the below equations are valid:

$$K_T = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \qquad (6)$$

In this equation, u' و v' و w' are the turbulent velocity fluctuations. 0.5 coefficient is based on the sum of kinetic energy due to the turbulence in three different Cartesian directions. Note that turbulent energy dissipation using defined k-e model is considered for the simulation. Figure (4) represents the velocity profile at the intake inlet:

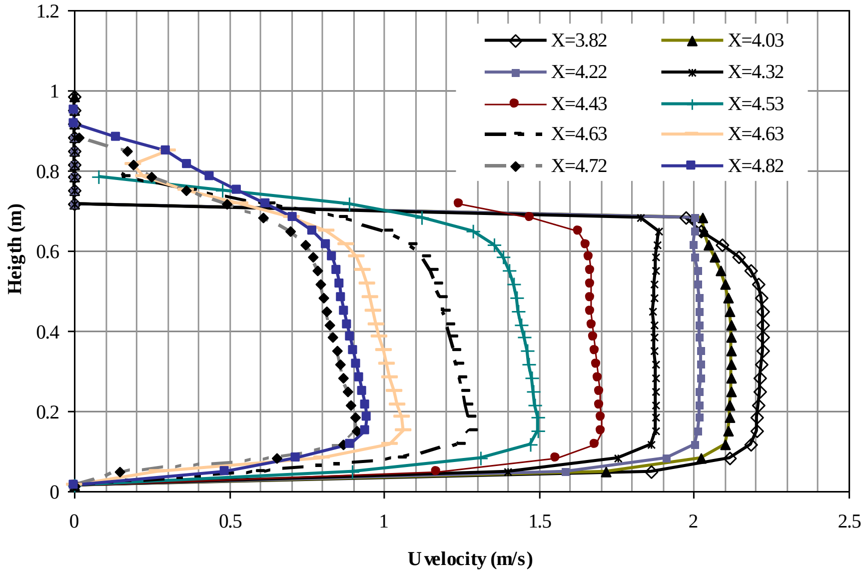

Fig. 4: velocity profile variations at the intake inlet

As it is shown in figure (4), maximum velocity at the intake inlet is at the beginning, while at the end, the velocity is reduced to less than half. This speed variation from maximum to minimum, i.e. from 2.2m to 0.85m causes gradient at the inlet which makes a vortex flow and thus turbulent energy dissipation is occurred. Quantitatively, turbulent energy variations from the beginning of the inlet to it end increases to about 60%, while turbulent energy dissipation at the beginning of the inlet (river) to its end increases about 31%. Figure (5) shows the velocity profile variations at the intake inlet. Moreover table (2) represents the average hydraulic parameters inside the T-shaped Sabily intake inlet:

Table 2: hydraulic characteristics of flow in the intake channel`s inlet itself

X	U_{ave}	tke	dtke
3.8250	0.0020	0.0015	0.0194
3.9250	0.0313	0.0535	0.0535
4.0250	0.0419	0.0860	0.5702
4.1250	0.0344	0.0729	0.8784
4.2250	0.0250	0.0496	1.0665
4.3250	0.0194	0.0342	1.0849
4.4250	0.0265	0.0411	0.8152
4.5250	0.0320	0.0464	0.6099
4.6250	0.0331	0.0427	0.3999
4.7250	0.0071	0.0097	0.0915

According to result shown in table (2), it is observed that turbulent energy from the beginning of the inlet is increases along linear direction to its end for about 85% and the amount of energy dissipation is increased 78% because of it. In the second case, using an elbow channel has been taken into account. Figure (5) shows a view of executing the model in case of elbow channel which is considered as baffle at the top of the intake channel:

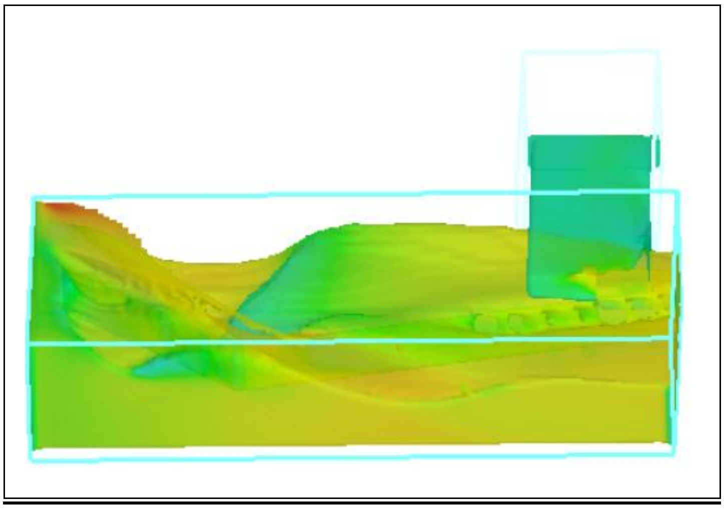

Fig. 5: a view of the mathematical model execution in case of elbow channel

To further investigate the subject of river velocity profiles at the intake channel`s inlet is represented in figure (6) in which velocity profile variation is shown from the beginning to the end:

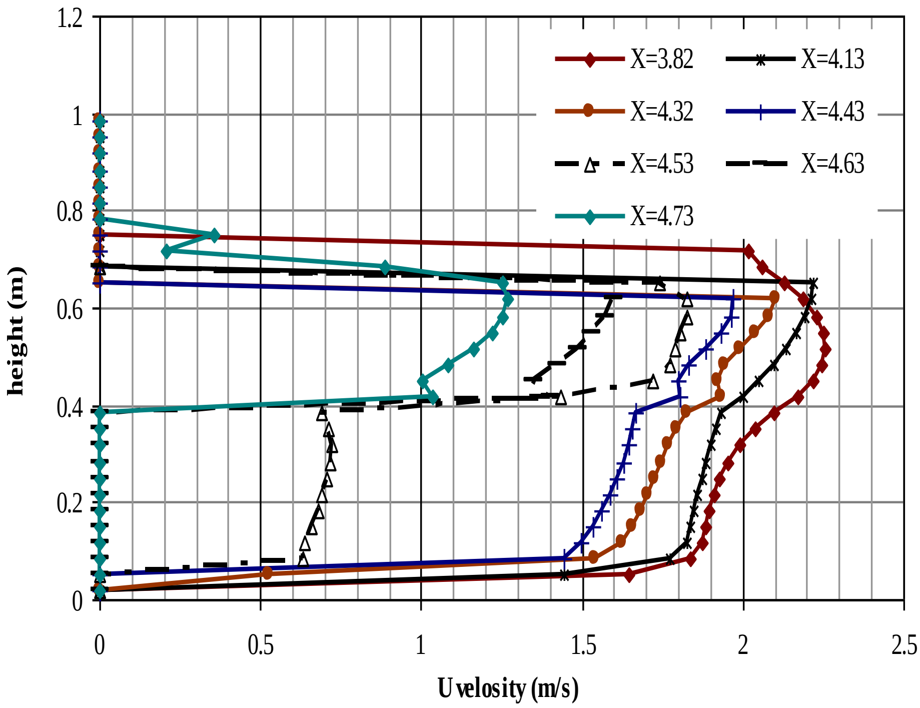

Fig. 6: velocity profile variation at beginning of the intake inlet (river)

As it is shown in figure (6), it is obvious that due to the baffle at the intake inlet, velocity profile is jagged. In general from the beginning of the inlet to its end, flow velocity decreases from 2.25 to 1.3 m/s. in comparison to the case of no elbow channel, velocity gradient is increased in this case, thus torsional velocities decrease by which sediments are introduced into the intake channel. Quantitatively, maximum velocity gradient without elbow channel reduces from 61% to 42% in case of elbow channel in place. Accordingly, results show that using an elbow channel is effective in modifying the flow pattern to reduce torsional velocities. Table (3) shows the result related to turbulent parameters, turbulent energy dissipation and also average velocity at the intake inlet.

Journal of Applied Science and Agriculture, 9(3) March 2014, Pages: 1153-1160

Table 3: hydraulic characteristics of river flow at the intake channel`s inlet with an elbow channel

X	Tke	dtke	Uave
3.8250	0.0386	0.0527	1.8620
3.9250	0.0430	0.0636	1.8370
4.0250	0.0460	0.0748	1.7647
4.2250	0.0512	0.1120	1.6159
4.2250	0.0436	0.0972	1.6515
4.3250	0.0484	0.1162	1.5802
4.4250	0.0612	0.4741	0.8137
4.5250	0.0612	0.4741	0.8137
4.6250	0.0330	0.1477	0.4538
4.7250	0.0336	0.0833	0.4469
4.7750	0.0506	0.1039	0.4568

As it is shown in table (3), along the river flow at the intake inlet, maximum velocity is reduced in average from 1.86 to .45 m/s. moreover turbulent energy variation in case of elbow channel is increased 23%, while under the same conditions without the elbow channel the various increases 60% which shows the efficiency of the elbow channel. Moreover for turbulent energy dissipation, in case of an elbow channel, the turbulent energy dissipation is about 50%, while in case of elbow channel, the corresponding value is about 31% which proves the vortex flow potential is decreased when using the elbow channel.

Conclusions:

According to discussions in this study, flow pattern result for intakes using elbow channel in comparison to intakes without it were generally investigated. The summary of the results are as follows:

- In case of not using elbow channel, velocity variation at the intake inlet decreases from 2.2m/s to 0.85m/s which leads to velocity gradient at the inlet and consequently vortex flow. Results showed that quantitatively, maximum velocity gradient in case of not using elbow channel reduces from 61% to 42% in case of using one.
- Result show that in case of no elbow channel, turbulent energy variations from the beginning of the intake inlet to its end increases for about 60%, while the amount of turbulent energy dissipation from the beginning to the end is about 31%.
- Results show that in case of elbow channel along intake flow, maximum speed is decreased in average from 1.86 to 0.45 m/s. moreover turbulent energy variations in case of elbow channel is increased by 23%. Moreover for turbulent energy dissipation in case of elbow channel, turbulent energy drop variations are about 50%, while without the elbow channel this value is about 31%.

REFERENCES

Kurosh, Vahid, 2010. “experimental investigating of water surface profiles formation in a novel intake from river by floor intake with a porous environment”, Eighth National Conference of Civil Engineering, Shiraz University.

Moazen, Iman, 2007. “Evaluation irrigation efficiency of vortex tube at the beginning of the channel “, national National Conference of Irrigation and Drainage Network Management, Shahid Chamran University, Ahvaz

Keshavarz, Mohammad Hosein, 2009. “numeral simulating of 3D flow pattern surrounding perpendicular to the shore and oblique breakwaters considering different border conditions”, Iran marine engineering community, marine engineering magazine, fourth year, NO-8.

Shamloo, Hamid, 2011. “investigating the effects of geometrical and hydraulic parameters on separation zone`s dimensions in a lateral intake using Fluent”, Civil Engineering and topography magazine, faculty of Engineering, ،44NO4 ،P487-495.

Pirestani, Mohammad Reza, 2009. “analyzing the lateral intake flow in arch channel using a physical model”, Water and Soil magazine (Science and Industry of Agriculture), Issue 22 NO2.

Salehi Neishaburi, Seyed Ali Akbar, “Numeral Modeling of 3D Flow Pattern in Lateral Intake”, First National Conference of Civil Engineering, Sharif University of Engineering.

Barkdoll, B.D., 1997. "Sediment Control at Lateral Diversion", PhD dissertation, Civil and Environmental Engineering, University of Iowa, Iowa City, Iowa.

Shettar, A.S., and K.K. Murthy, 1996. “A Numerical Study of Division of Flow in Open Channels", J.Hydr. Res., 34(5): 651-675.

Issa, R.I., and P.J. Oliveira, 1947. “Numerical Prediction of Phase Separation in Tow-Phase Flow through TJunctions", Comp. and Fluids, 23(2): 347-372.

Neary, V.S., *et al*., 1999."Three-Dimensional Numerical Model of Lateral-Intake Inflows", J. Hydr. Engrg, ASCE, 125(2): 126-140.

Journal of Applied Science and Agriculture, 9(3) March 2014, Pages: 1153-1160

Lee, D., and J.J. Chiu, 1992. "Computation of Physiological Bifurcation Flows Using a Patched Grid", Comp. and Fluids, 21(4): 519-535.

Hsu, Chung-Chieh,Tang. Chii, Jau, Lee, Wen-Jung and Shieh, Mon-Yi, 2002. “Subcritical 90. Equal-Width open-channel dividing flow.” ASCE, J. Hydr. Eng., 128(7): 716-720.