



## ORIGINAL ARTICLE

# The consequences of varying the width ratio parameter of the lateral water intake structure to the main canal

<sup>1</sup>Amin Atarodi, Najaf Hedayat<sup>2</sup>, Mohammad Hadi Tavana<sup>3</sup> and Ali Panahian<sup>4</sup>

<sup>1</sup>Department of Civil Engineering, Islamic Azad University, Kermanshah Branch, Iran

<sup>2</sup>Department of Water Engineering, Islamic Azad University, Dezful Branch, Dezful, Iran

<sup>3</sup>Department of Civil Engineering, Islamic Azad University, Kermanshah Branch, Iran

<sup>4</sup>Department of Civil Engineering, Islamic Azad University, Kermanshah Branch, Iran

Corresponding author Email: amin\_atarodi2005@yahoo.com

### ABSTRACT

*One of the major design features in hydraulic engineering is consideration of the flow pattern in lateral water intake structures that are vital for proper functioning of water conveyance and distribution regimes. This paper aims at systematic analysis of the ways in which varying ratios of two canal widths could affect the water abstraction regime. The methodology involves investigating the hydraulic behavior of three types of turbulent flow regimes with the Froude Numbers of 0.13, 0.19 and 0.27 and five width ratios of 0.5, 0.75, 1, 1.25 and 1.5. The analysis incorporates four discharge ratios of 0.25, 0.375, 0.45 and 0.75, using 3D numerical models with fluent software. The effect of flow velocity and discharge ratio is also used in the analysis. Increase of width ratios had a corresponding increase in length and width of separation or vortex zone around the junction. It was observed that increase in the distributed discharge ratio of two canals decreased the length and width of the separation zone. The overall results was that initial flow velocity in the main channel does not have any bearing on the vortex dimensions generated in the canal junction.*

**Keywords:** Fluent software, Lateral intakes, Numerical model, Turbulent flow.

Received 22 /01/2014 Accepted 13/02/2014

©2014 AELS, INDIA

### INTRODUCTION

Water intake structures are the parts and parcels of water convey and distribution infrastructures designed to supply water for agricultural, domestic and industrial sectors [1], [2], [3]. Optimum utilization of water in these hydraulic appurtenances calls for an analytical investigation of hydraulic flow pattern in intake structures, conveyance and distribution canals [3]. Identification of such hydraulic parameters is very crucial under circumstances of critical hydrological conditions like severe draught.

The commonly-used waterways for this purpose are lateral water intakes structures quite capable of handling complex 3-D turbulent flows that are caused by lateral pressure gradient and shear forces. This in turn, generates secondary flows in the clockwise direction.

The resulting phenomenon becomes instrumental in transporting sediment loads into the vortex area by secondary flows that help their settlement, and as such, undermines the capability of water intake flow into the feeding canal and adversely affecting the conveyance and distribution regime. When flow enters the intake structure, the shear suction pressure at the end of canal junction causes lateral acceleration. This, in turn, makes it possible for the flow to enter the canal junction and the part of which that remains in the main canal, is subsequently conveyed downstream. One of the design features in these structures is the dividing stream surface that separates these two flow sections, known in literature as the dividing stream surfaces [4]. The 3D flow pattern in the lateral water intake structures is schematically shown in fig. 1.

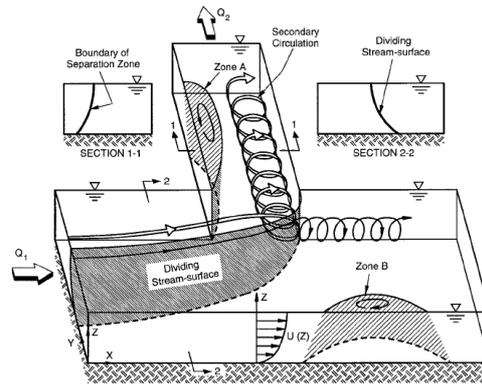


Fig. 1 3D flow patterns in lateral intake

Vortex formation or separation zone is one of the legacies that decreases water flow through the water intake structures and ultimately into the canal systems. That necessitates the establishment of conditions required for maximization of diverted flow into the intake structures with minimum dimensions of separation zones or vortex generation in the area. The establishment of such state of affairs is of vital importance in enhancing the operational efficiency of the structures. For this very reason the hydraulic behavior of flow under such circumstances becomes the cause for investigation in this paper.

Taylor (1944) experimental works in an open canal showed that for flow entries with a Froude Number less than 0.4 there will be a 2% water level variation around the intake structure. In an experimental work on a 90 degree junction, Kasthuri and Pundarikanthan [6] showed a converse relation between the dimensions of flow separation zone in junction and distributed discharge ratio between the two canals. In their experimental investigations, Barkdoll et al [7] compared the flow in an open-canal with a width ratio at a 1 to 2 depth using a flow pipe with a width to depth ratio of 1 to 4. They attributed the velocity difference around the water surface equal to  $0.36 U_x/U_0$  to the secondary flows generated in the open canals.

Neary and Odgaard [8] analytically investigated the roughness effects of the bed over 3D diverted flows and observed that flow velocity profile with even bed in all flow depths is more uniform than those with rough beds.

Issa and Oliveria [9] modeled 3D turbulent flow patterns in 90 degree intakes. They used Reynolds's average equations accompanied by K- $\epsilon$  turbulent model equations. Results showed a more accurate simulation by the 3D models than the two-D ones.

Neary et al. [8] numerically investigated the lateral intake inflows using 3D two-equation turbulence models without considering the water surface effects. Their findings suggested a close proximity between the two sets of data.

Safarzadeh [10] used 3D numerical model with k- $\epsilon$  and k- $\omega$  turbulence models to predict the flow pattern in the open channel diversion.

Ramamurthy et al. [11] presented experimental data related to 3D mean velocity components and water surface profiles for dividing flows in open channels. The data set presented in their paper was composed of water surface mappings and 3D velocity distributions in the vicinity of the channel junction region. Dimensions of separation zone obtained from the current 2D-numerical model were compared with those of Kasthuri and Pundarikanthan's experiments [6]. Velocity field measurements were compared with measured velocities of Shettar and Murthy (1996). They presented depth-averaged mean velocity profiles in different sections across the main and intake channel.

The aim of the paper is simulation of the flow pattern in the intake structures in straight direction based on the experimental investigation of Barkdoll et al [7] using the Fluent software. For this reason, the velocity profiles of the experimental and numerical models near the water surface were compared to validate the results. The modeling included parameters such as intake rate and related factors such as the ratio of width, flow velocity and the distributed discharge ration in two main canals and junctions. The parameters included turbulent flows with Froude of 0.13, 0.19 and 0.27 and four discharge ratios of 0.25, 0.375, 0.45 And 0.75 as well as five width ratios 0.5, 0.75, 1, 1.25, 1.5.

## MATERIALS AND METHOD

The model was designed on a main canal length of 2.75 (m), the length of diversion weir of 1.68 (m) and the angle of 90 degrees relative to the main canal. The model also had an inflow discharge of 0.011 ( $m^3/sec$ ), depth of flow 0.31 (m), and the width of the two canals at 0.152 (m), the Froude Number of 0.13

and the Reynolds Number of 72320. The flow was therefore turbulent and sub-critical.  $X^*$  and  $Y^*$  are the dimensionless quantities which are derived at from dividing  $x$  and  $y$  by the canal width  $b$ . the ratio of discharge  $q_r$  is the discharge of the diversion canal by the total inflow discharge. The width ratio is derived at by dividing the width of the diversion canal to the main canal (fig. 2).

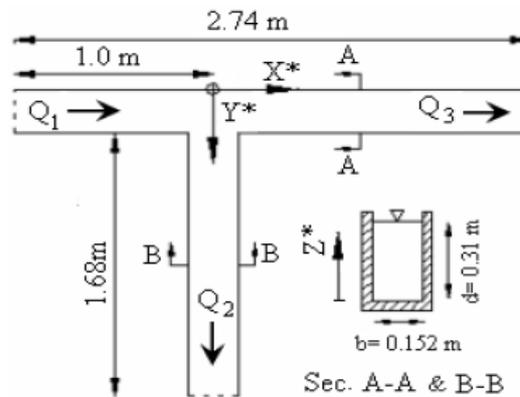


Fig 2. Analysis of the model sensitivity relative to various sediment transports

#### GOVERNING EQUATIONS

The 3D RANS equations for an incompressible fluid, in Cartesian coordinates are given as:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (1)$$

$$U_i \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + g_{x_i} + \mu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{\partial R_{ij}}{\partial x_j} \quad (2)$$

Where  $U_i$  = mean velocity,  $X_i$ = position,  $p$ = the mean pressure,  $\mu$ = the dynamic viscosity and  $R_{ij}$ = the Reynolds stress tensor and should be modeled using turbulence closure equations [12].

#### NUMERICAL METHOD

The commercial mathematical model of FLUENT, version 6.3.26 is used for this study. Geometry of model is constructed using Gambit software, version 2.3.16. In this study all simulations are computed using the RSM model. The pressure-velocity coupling is achieved using the SIMPLE algorithm, and the discretized equations are solved with Second order upwind scheme.

#### BOUNDARY CONDITIONS

Velocity inlet boundary condition is specified at the main channel inlet. Appropriate conditions must be specified at domain boundaries depending on the nature of the flow. Outflow boundary condition used for two outlets at all of runs. The length of the main and branch channels were chosen properly; therefore sufficient distance is provided between the junction and two outlets to ensure that the flow returned to the undisturbed pattern. The no-slip boundary condition is specified to set the velocity to be zero at the solid boundaries and assumed to be smooth.

Taylor showed that for  $Fr$  between 0 and 0.4, there is less than 2% variation in flow depth in the vicinity of diversion [5]. Therefore, symmetry boundary condition has been applied for free surface. Wall boundary condition was applied for the solid boundaries.

It is also important to establish that grid independent results have been obtained. The grid structure must be fine enough especially near the wall boundaries and the junction, which is the region of rapid variation. Various flow computational trials have been carried out with different number of grids in  $x$  and  $y$  directions. It was found that results are independent of grid size, if at least 137000 nodes are used.

#### MODEL VALIDATION

Fig. 3 shows the comparison of the velocity profiles derived from the experimental and numerical models for the discharge ratio of 0.31 near the water surface. Results suggested a close proximity between the observed data and the model data which confirmed the validity components of the findings.

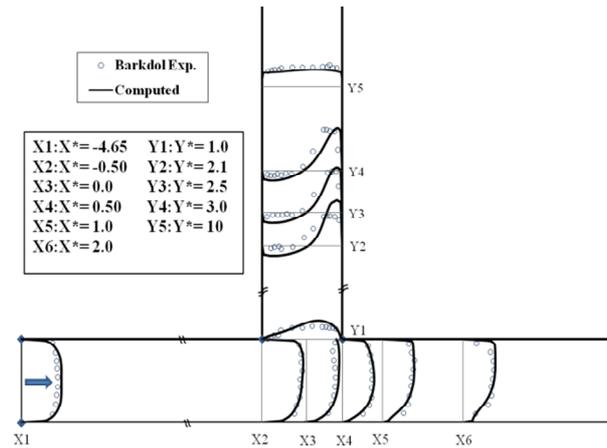


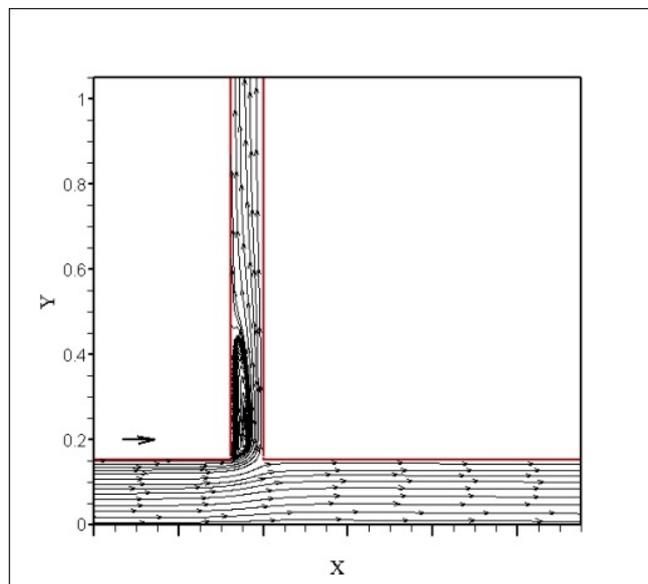
Fig 3 .Comparison between predicted and measured velocity profiles near the surface

**RESULTS AND DISCUSSION**

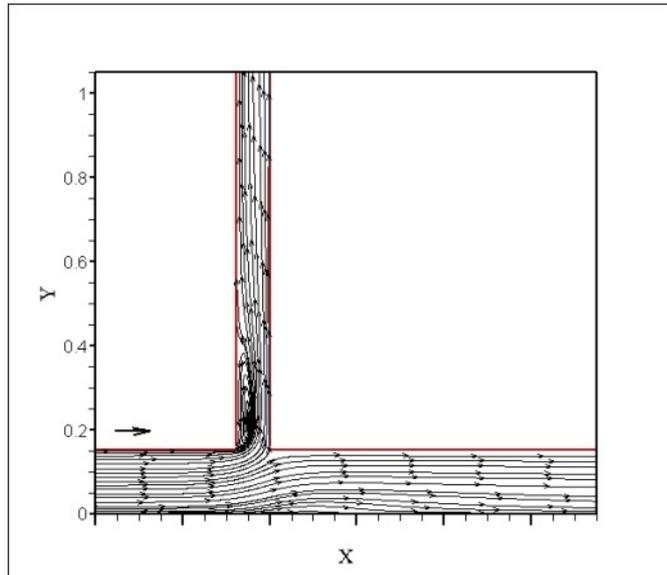
Results show that vortex formation or separation zone in the water intake spreads too much of the width at the beginning of the intake structure. Assuming a constant width ratio in all cases, the discharge ration has an inverse relation with the length of vortex. In other word, as the discharge ratio increases the width and length of the vortex and the separation zone decreases. It was also found that increase in discharge ratio has a direct relation with the distance of the separation zone from the internal wall of the main canal (fig. 4).

Results indicated that under certain condition of high discharge ratio, a new separation zone in the main canal and downstream of the canal reach establishes. The underlying reason for the formation of this separation zone is explained by the suction effects of the intake structure. It is for this reason that its dimension increases with an increase in the discharge ratio.

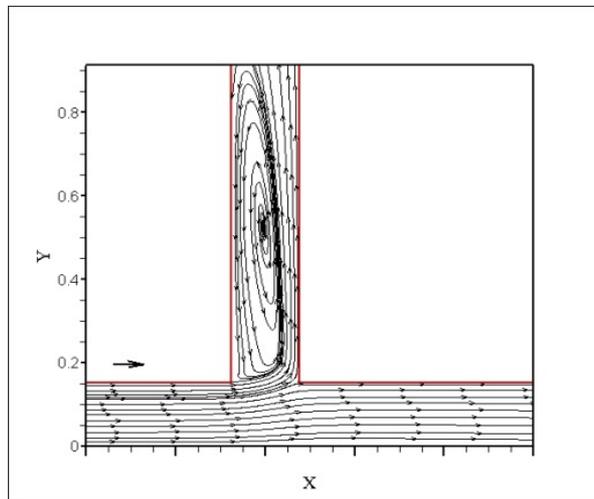
Results (fig. 5) show that dimensions of vertex formed in intake structure in all states increases with an increase in junction’s width.



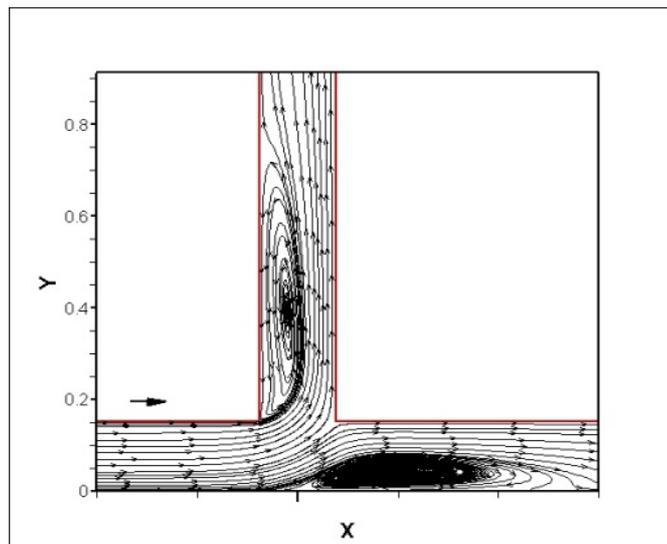
a)  $b/B = 0.5, qr = 0.25$



b)  $b/B=0.5, qr=0.45$



c)  $b/B=1, qr=0.25$



d)  $b/B=1, qr=0.75$

**Fig4 . 2D streamline plot at near water surface**

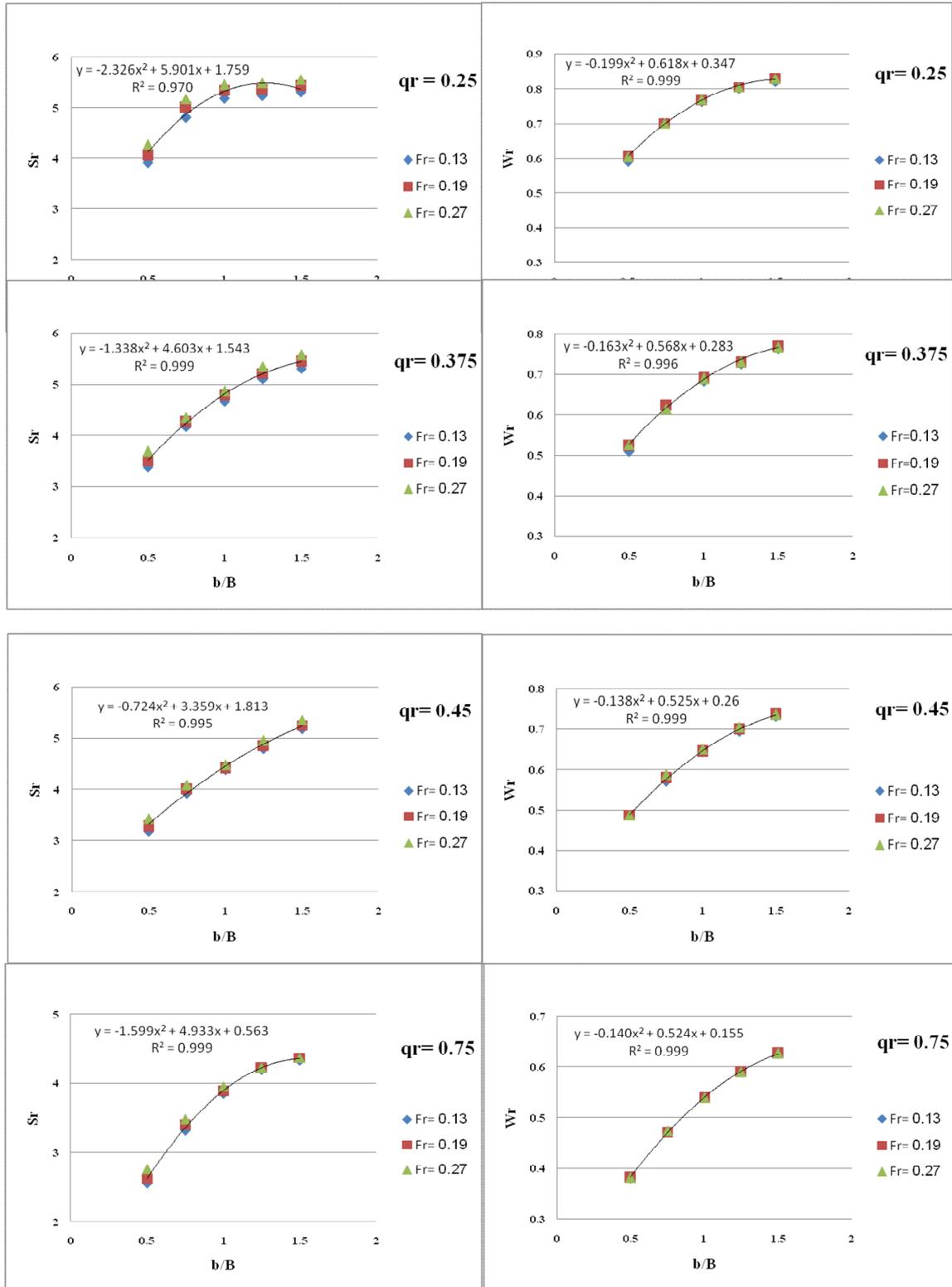


Fig 5. The width ratio- vertex dimension curve of numerical models

The flow velocity entering in the main canal does not seem to have significant effects on the vertex formation in the junction.

**CONCLUSIONS**

There is a close correlation between the results of modeling and those of the experimentations, as is suggested by the close proximity between the two sets of data. Results also suggest an increasing trend on the length and width of the vortex formed in the vicinity of the water intake structures due to increase in width ratio. The sedimentation in the junction increases as a result of decrease in the water intake. The increase in the discharge distribution ratio the dimension of the separation zone in lateral intake gets smaller and as such, the rate of water intake improves. In higher discharge ratios a new vortex in the main canal is formed the dimension of which has a direct relation with the increase in discharge ratio. It was also found that the inflow velocity has no significant effect on the dimension of the vortex or separation zone.

**REFERENCES**

1. S. Ashkarian, H. Kiamanesh and N. Hedayat, "Simulation of the flow pattern using Flow 3D software, a case study in the Dez diversion Weir in Dezful", Iran. Proceedings: National Conference Soil, water, Plant and Mechanization Conference, Islamic Azad University, Dezful, Iran, 10-12 march 2010.
2. H. Shirin, N. Hedayat, H. Kiamanesh. and Z. Tamoradi, "Application of the SHARC model for simulating sediment transport at Eastern Intake of the Dez Diversion Weir in Dezful, Iran", Proceedings of the WASET Conference in Paris, 28-30 July 2010, Paris France.
3. N. Hedayat, "Improving the performance of water Delivery in the Dez and Moghan Irrigation Schemes in Iran", PhD Thesis, Cranfield University, UK, 2005.
4. V. S. Neary, F. Sotiropoulos, and A. J. Odgaard, "Three dimensional model of lateral intake inflows", J. Hydraul. Eng., ASCE, vol. 125, no. 2, 1999, pp. 126-140.
5. E.H. Taylor, "Flow Characteristic at rectangular open-channel junctions", Trans ASCE, Vol. 109, 1944, pp. 893-912.
6. B. Kasthuri, and N.V. Pundarikanthan, "Discussion on separation zone at open channel junction", J. Hydraul. Eng. vol. 113, no. 4, 1987, pp. 543-548.
7. B.D. Barkdoll, B.L. Hagen, A.J. Odgaard, "Experimental comparison of dividing open-channel with duct flow in T-junction". J. Hydr. Eng. ASCE, vol 124, no. 1, 1998, pp. 92-95.
8. V. S. Neary, and A.J. Odgaard, "Three dimensional flow structure at open channel diversions", J. Hydraul. Eng., ASCE, vol. 119, no. 11, 1993, pp. 1224-1230.
9. R. I. Issa, and P. J. Oliveira, "Numerical prediction of phase separation in two-phase flow through T junctions". Comp. and Fluids, vol. 23, no. 2, 1994, pp. 347-372.
10. A. Safarzadeh, "Numerical simulation of flow pattern at lateral intake in 180-degree bend", Msc thesis, Civil Engineering, Tarbiat Modares University, Tehran, Iran, 2005.
11. A.S. Ramamurthy, J. Ou, and D. Vo, "Numerical and experimental study of dividing open-channel flows", J. Hydraul. Eng, vol. 133, no. 10, 2007, pp. 1135-1144.
12. Fluent 6.3 Inc., "Fluent User's Guide", 2501 pp., September, 2006.

**How to cite this article:**

Amin A, Najaf H, Mohammad Hadi T and Ali P. The consequences of varying the width ratio parameter of the lateral water intake structure to the main canal. Bull. Env. Pharmacol. Life Sci. 3 (4) 2014: 48-54