Cumhuriyet Üniversitesi Fen Fakültesi
Fen Bilimleri Dergisi (CFD), Cilt:36, No: 3 Özel Sayı (2015)
ISSN: 1300-1949

Cumhuriyet University Faculty of Science
Science Journal (CSJ), Vol. 36, No: 3 Special Issue (2015)
ISSN: 1300-1949

Numerical Investigation of Sedimentation Problems in Riverside Pumping Stations

Adel ASNA ASHARİ[1], Edris MERUFİNİA[1*]

[1]*Young Researchers and Elite Club, Mahabad Branch, Islamic Azad University, Mahabad, Iran*

Received: 01.02.2015; Accepted: 05.05.2015

Abstract. In this study, the numerical simulation of the flow hydraulic and sedimentation in an intake with rectangular channel is performed using SSIIM2 model. At the first, the distribution of the profiles of the flow velocity near the water surface, the size of the flow separation zone in the intake and inflow sediment to the intake at various sections of the main channel and the intake for a constant inflow with different turbulence models were simulated in 3D state and compared with the experimental results of other researchers and there were a good agreement between them. Considering the obtained good results, in order to study the flow pattern and the sedimentation in the pumping station inlet, the numerical analysis of the parameters affecting the hydraulic field and sedimentation in front of the inlet of a real pumping station is performed.

Keywords: Lateral intake, sedimentation, pumping stations, SSIIM2 numerical model, K-ω turbulence model

1. INTRODUCTION

Determining the flow structure from the main channel to the lateral channels is very important in many hydraulic installations. Generally the intakes are used in distribution networks, irrigation channels, drainage systems, water treatment plants and sewage installations, the entrance to electric utilities and etc. the deviation of a part of the river water into the channel causes curvature and flow of water. This causes the transverse flows. As shown in Figure1 in the intakes when the fluid flow enters from the main channel into the sub-channels a limited are called the separated zone from the wall in created. In this area of flow the fluid particles are moving around themselves at a distance from the wall such that the area of the lateral channel is not effective on the flow discharge. Failure to control sediment entering the intakes causes the transference of the sediments into the Irrigation and installations channels and leads to problems at different parts. If the flow velocity is too high, the suspended particles cause damages to the facilities such as pumps and turbines. Also, due to changes in the velocity distribution at the intake usually the sedimentation occurs in the intake inlet which decreased the intake efficiency, the entrance of the coarse sediments into the network, the increased cost of operation for reducing the sediments and finally the change of the flow line into the opposite shore in front of the intake. Anything that reduces the secondary flows and vortexes in the intake inlet will lead to the reduced sediments in the inlet and prevents the sediments from entering the intake channel. Figure 1 is the Schematic of the flow separation zone in a 90 degree intake. In this figure B and b are the width of the main channel and intake and Wr and Lr are the width and length of the separation zone, respectively.

*Corresponding author. *Email address*: *edris.marufynia@yahoo.com*

Special Issue: The Second National Conference on Applied Research in Science and Technology

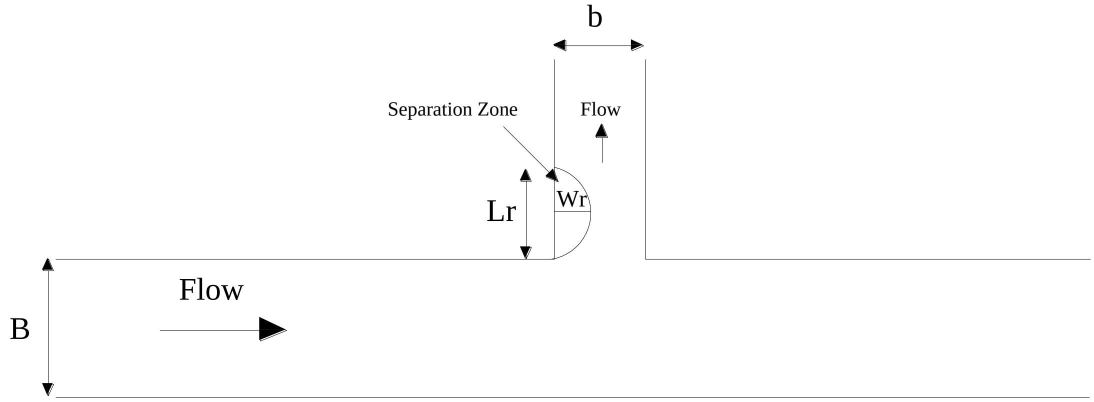


Figure1. The schematic of the flow separation zone at 90 degree intake

2. A REVIEW OF PREVIOUS STUDIES

Neary et al. (1993) conducted experimental studies on the flow hydraulic in the 90° intakes. Nakato et al. (1999) with the construction of physical models for five inlets with sandy beds analyzed the problems of sediments entering to the intake and the ways to reduce them. Behzadipour (1998) studied the behavior of sedimentation in Amirkabir pumping station using a physical model. In this study, in order to investigate the problems of sedimentation in the Agro industrial Amirkabir pumping station in Karoon River the existing physical model with a tilt scale is built and tested [3]. Issa and Oliveira (1994) were the first ones who conducted the 3D simulation of the turbulent flow with T shape geometry. They solved the time averaged Navier-Stokes equations by Reynolds method in 3D state (RANS) with k-ε standard model through wall functions [8].Safarzadeh and Salehi Neyshabouri (2004) analyzed the numerical 3D pattern of flow in the lateral intake [2]. Pirzadeh (2007) examined the flow hydraulic in the lateral intakes using the Fluent program. In this study the analysis of the flow velocity profiles in 3D mode was performed by choosing different turbulence models in the program which was consistent with the experimental results. [1]

The first stage of modeling includes hydraulic numerical simulation and flow sedimentation in the riverside lateral intake from a rectangular channel with straight walls which is done by SSIIM2 program and based on the experimental study of Barkdoll et al. (1998).

3. THE CHARACTERISTICS OF THE EXPERIMENTAL MODEL

In the experimental study of Barkdoll et al. (1998), the length of the main channel was 24m and its width was 1.5m with a height of 0.152. The intake was done through a lateral channel with width of 0.6m, length of 2.45m with 90 ° angle. In this study the average diameter of the bed sediment was (d_{50}) 1 mm, the bed load thickness was 20 cm, the experiment time was 2 hours and the input concentration was 0.3 kg/m^3. The schematic of the channel is presented in Figure 2 [4].

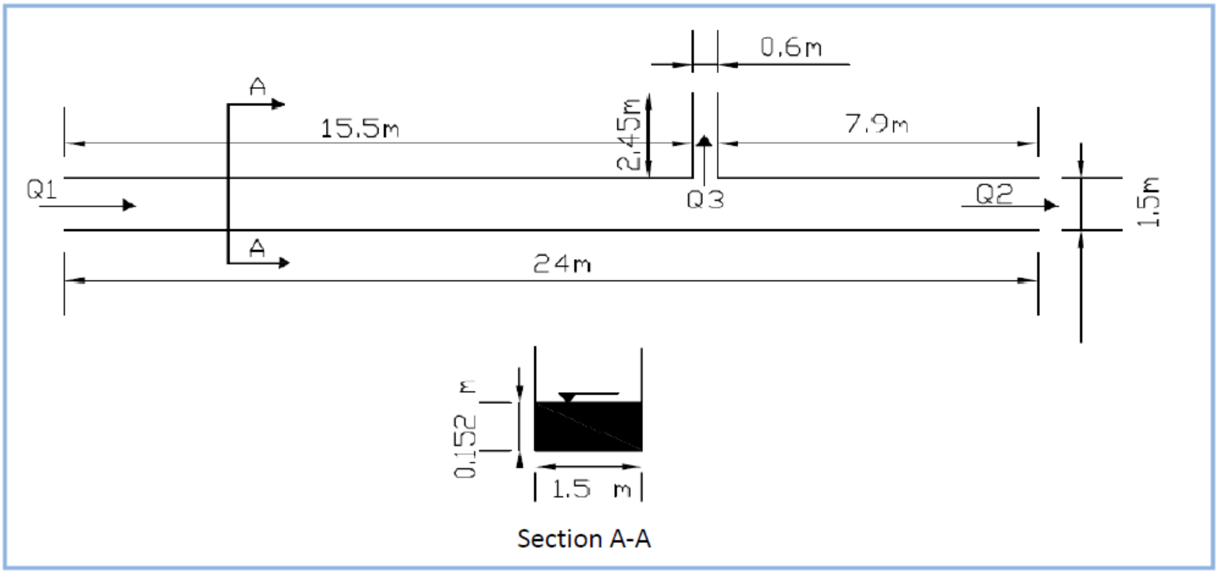


Figure 2. Geometric properties of the flume.

4. DESCRIBING THE NUMERICAL MODEL

The SSIIM model is designed for the simulation of the sediments in the intakes and for sediment and hydraulics engineering, environment and river management. The program is designed based on the finite volume method with an organized three-dimensional network and the Navier-Stokes equations and advection and diffusion equations for sediment transport are solved by SIMPLE algorithm using the volume control method. Turbulence models used in this Model include: The standard k-ε model with RNG expansion, k-ε turbulence model based on water velocity, k-ε turbulence model with the Wilcox's wall rules.

5. THE GOVERNING EQUATIONS

In this model, for the turbulent flow in a three-dimensional geometry, Navier-Stokes equation is used to calculate the flow rate of water according to equation (1).

$$\frac{\partial U_i}{\partial t} + U_i \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left(-\rho \delta_{ij} - \rho u_i u_j \right) \quad (1)$$

Where U is the flow velocity, P is the pressure and δ_{ij} is the Kronecker delta.

In the SSIIM2 model the sediment is divided into the suspended load and bed load. In order to calculate the suspended load the advection-diffusion equation is used descried as follows:

$$\frac{\partial c}{\partial t} + Uj \frac{\partial c}{\partial Xi} + \omega \frac{\partial c}{\partial z} = \frac{\partial c}{\partial Xj} (\Gamma \frac{\partial c}{\partial Xj}) \quad (2)$$

Where c: Sediment concentration, ω: fall velocity, U: Flow velocity, X: distance, Γ: diffusion coefficient.

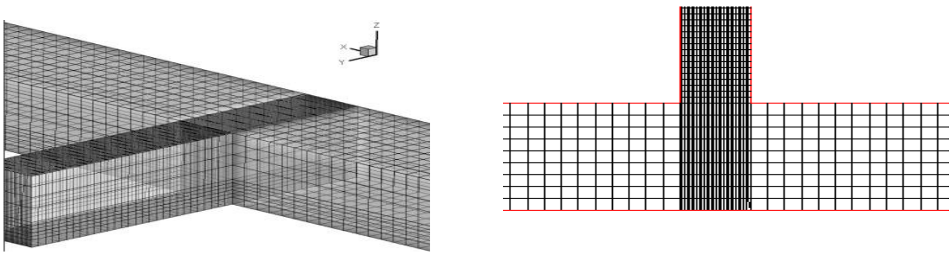
Van Rhine in 1987 suggested an equation for the equilibrium concentration of sediment in the bed for the bed load which is used in this program:

$$c_{bed} = 0.015 \frac{d^{0.3}}{a} \frac{\left[\frac{t - t_c}{t_c}\right]^{1.5}}{\left[\frac{(\rho_s - \rho_w)g}{\rho_w v^2}\right]^{0.1}} \tag{3}$$

In equation (4), d: Sediment particle diameter, a: The base level for the roughness height, τ: bed shear stress, c: critical shear stress ρ_w and ρ_s: Water and sediment density and ν: the water viscosity [7].

6. SIMULATING THE FLOW CONDITIONS IN THE EXPERIMENTAL MODEL USING SSIIM2 MODEL

After testing different values and sensitivity to mesh dimensions, the dimensions of the main channel were set as 8.5*12*12.5 cm before and after the intake and the dimensions of the intake channel were set as 3*12*12.5 cm and the grid cell size were set as 2*8*12.5 as the optimal mesh and the calculations were performed for a total of 84330 cells. The distance between the first node to the bottom is 5.5cm. Figure 3 presents the plan and a three-dimensional view of the computational mesh in a 90° intake.



A B

Figure 3. The generated mesh in the computational domain: (a) plan and (b) 3D view.

7. A COMPARISON BETWEEN THE NUMERICAL AND EXPERIMENTAL MODEL

At this section consider the effect of the turbulence models on the flow velocity field and the inflow rate on the size of the separation zone created in the intake and the ratio of the sediments entering the intake.

8. THE COMPARISON OF VARIOUS TURBULENCE MODELS ON THE FLOW VELOCITY FIELD

This section investigates the effect of various turbulence models on the distribution of flow velocity profile at different sections of the main and intake channels. X * And Y * are obtained by dividing the X and Y by the channel width (b).

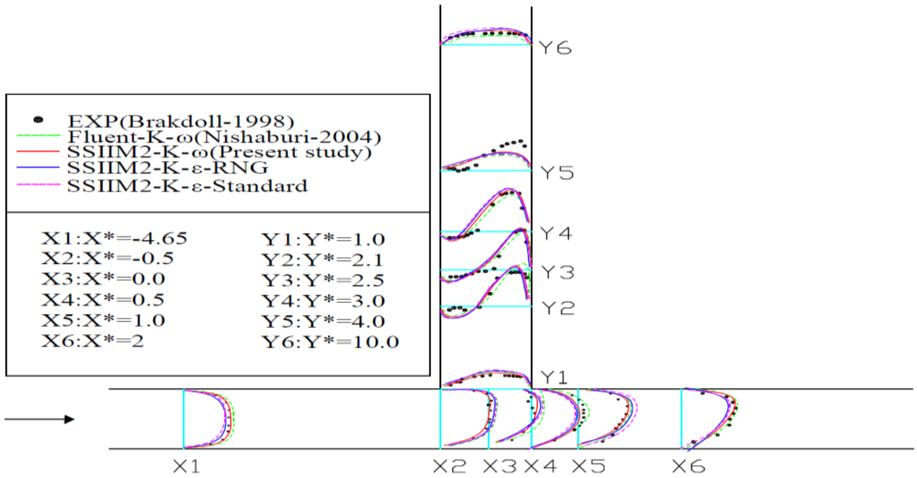
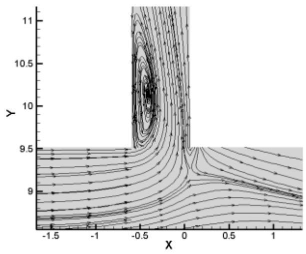


Figure 4. Comparison of the velocity near the water surface in the main channel and the intake for the discharge ratio 0.31.

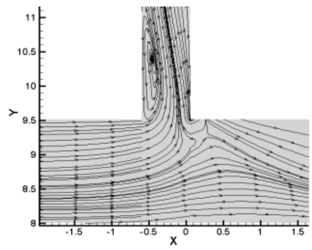
According to figure 4 until a distance before the intake inlet, velocity profile keeps its extended status and by approaching to the inlet deviate into the lateral channel due to the suction pressure applied by the intake and the maximum velocity is transmitted into the intake inlet (X*=-0.5 section). The k-ω turbulence model simulated better than the two other models and considered as the selected turbulence model in modeling process.

9. THE FLOW SEPARATION ZONE AND SEDIMENTATION ENTERING TO THE INTAKE

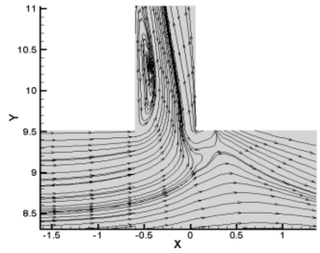
In this part, the effect of five discharge ratios 0.2, 0.43, 0.62, 0.83 and 0.9 (R) was studied for a discharge of 0.104 m^3/s for a constant Froude number (Fr) of 0.375 on length and width of the separation zone (Lr and Wr) and the Q_{sr} entering to the intake.



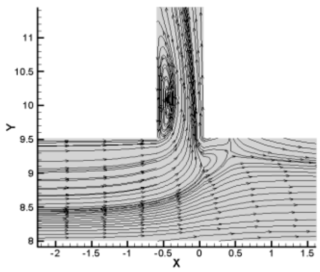
R=0.2



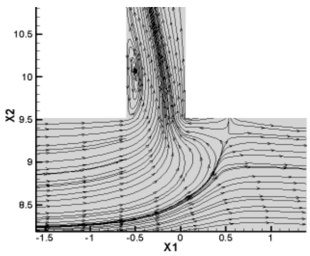
R=0.43



R=0.62



R=0.83



R=0.9

Figure 5. Flow lines of created in lateral channel.

According to Figure 5, with increasing the diverted discharge ratio, the component of transverse velocity in the inlet is increased and a higher amount of sediments enter to the intake channel. Figure 6 presents different Q_{sr} entering to the intake channel based on different ratios of diverted discharge and at a 90 degrees angle in both experimental study and numerical simulation.

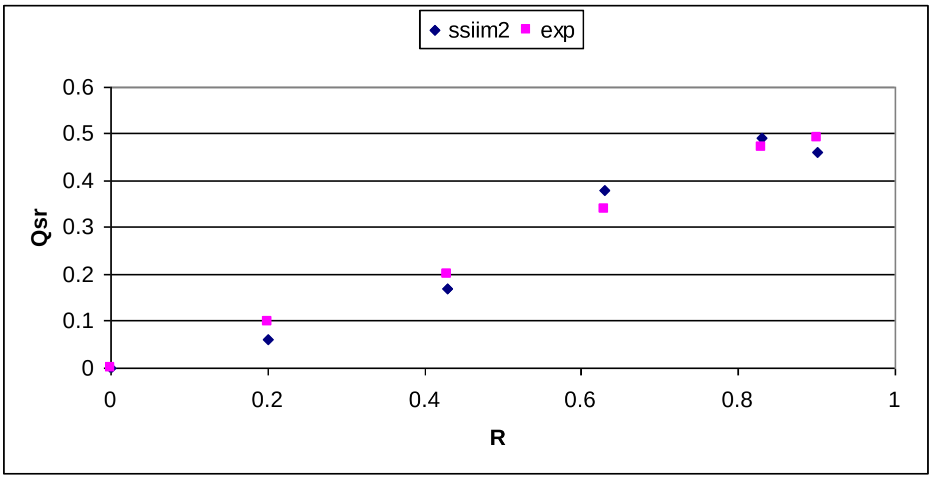


Figure 6. Q_{sr} versus R in experimental and numerical model for an angle of 90 degrees.

10. THE CHARACTERISTICS OF THE EXPERIMENTAL MODEL OF THE PUMPING STATION

In this section the experimental model of Emamgholi Zadeh et al (2007) is simulated to analyze the flow pattern and the deposition of the pumping station inlet. In this experimental model the length, width and depth of the main channel were 8 m, 1.2 m and 0.6 m. The dewatering was performed by an intake with the internal and external width of 1.20 and 0.75 m and the length of 0.22m which is placed in 4.8 meter downstream of the main channel with a 60° angle. In this modeling the average diameter of bed load sediment particles is 0.018mm and the average diameter of suspended load sediment particles is 0.05m, the inflow suspended load sediment is 30 g/sec and the sediment concentration of the inflow is 580 mg/lit and the needed time for numerical modeling is 6 h. Figure 7 is the schematic experimental flume of the pumping station with the intake inlet [5].

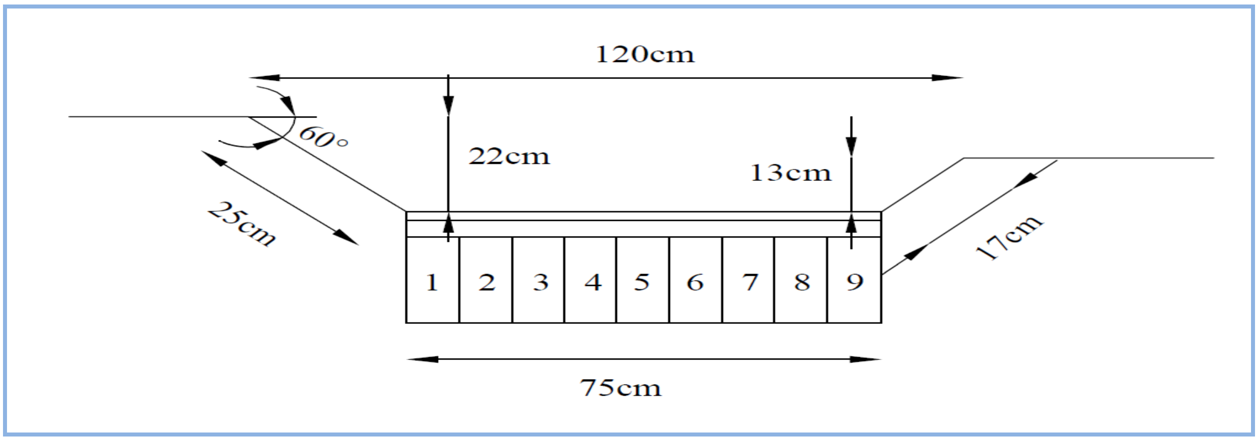


Figure 7. The schematic of experimental flume characteristics (EmamgholiZadeh et al., 2007)

In the this section, according to the good results obtained from the first section the numerical simulation of the effect of various parameters affecting the hydraulic field and the accumulated sediment in front of the Vase pumping station inlet on the Karun River has been done to simulate the flow pattern and sedimentation.

11. NUMERICAL RESULTS

Due to the unavailability of the output results from the experimental modeling of the pumping station for the adaptability and comparison with the numerical results, regarding the good agreement of the numerical and experimental results in the first part of the study and ensuring the ability of the model, only the numerical results are analyzed.

11.1. The intake bays maneuver and their effect on the sedimentation in front of the pumping stations inlet

Here the bays maneuver of the inlet and their effect on the sedimentation in front of the pumping stations inlet (regularly) on the Q_{sr} for a constant rate of flow 0.043 m^3/s, diverted discharge ratio (R) 0.15 and three Froude numbers 0.0248, 0.0296 and 0.0309 were analyzed. The listed Froude numbers were selected based on the flow depth changes. Figure 8 shows the changes of the rate sedimentation entering the intake in terms of the number of open bays (N).

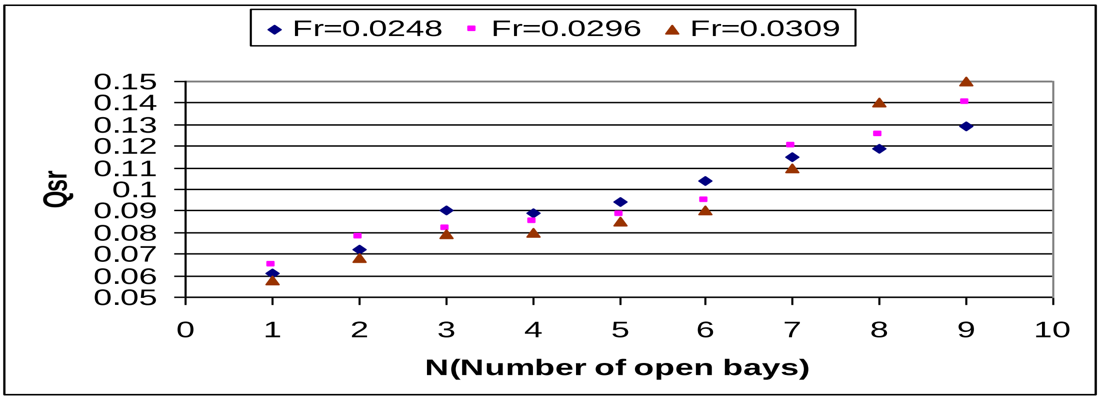


Figure 8. The changes of the rate sedimentation entering the intake in terms of regular maneuvering of the watering inlets.

11.2. The effect of different diversion angles (upstream slope of the intake wall) on the Qsr entering to the pumping station intake

In this study the effect of the intake angles of the downstream (θ) 45, 60, 75 and 90 ° for R= 0.15 and three Froude number 0.0248, 0.0296 and 0.0309 on the Q_{sr} of the pumping station was analyzed.

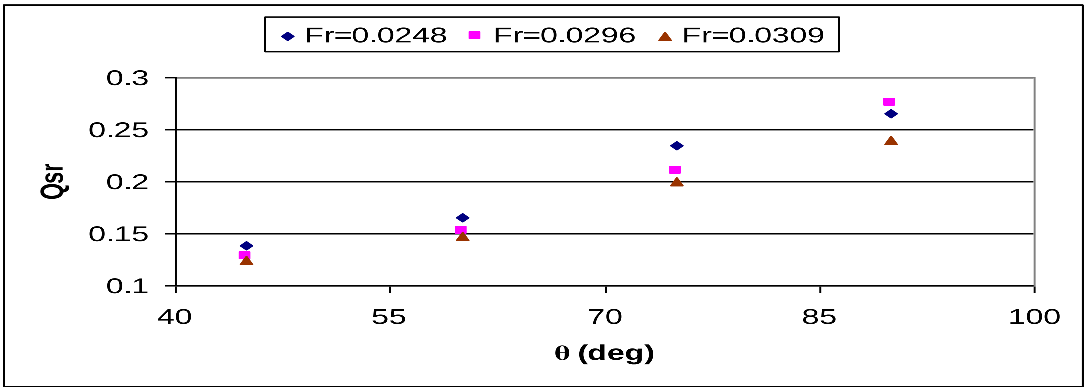


Figure 9. Changes of the Q_{sr} of entering to the pumping stations intake based on different diversion angles.

According to Figure 9, for a diverted discharge ratio, by increasing the intake wall angle (upstream wall slope of the station) the concentration of the entering sediment is increased. The reason for this is that, increasing the intake angle reduces the length of the vortex in the intake inlet and increases its width. Increased vortex in the inlet leads to increased sediment accumulation in the intake inlet and increased the sediments entering to the intake channel.

11.3. The effect of the ratio of the width of the pumping station intake to the main channel on the Qsr of the pumping station

Considering the geometry of the effect of six different ratios of width 0.25, 0.5, 1.5,2, 2.5 and 3 (b / B) for three ratios of flow distribution 0.32, 0.52 and 0.81 (R) and three types of inflow with Froude number such as 0.0248, 0.0296 and 0.0309 on the Q_{sr} entering to the pumping station intake was studied. According to Figure 10 we find that by increasing the width of the intake to the main channel (b / B), the Q_{sr} into the intake channel increases the main reasons of which is the constant inflow and increased width of the intake channel.

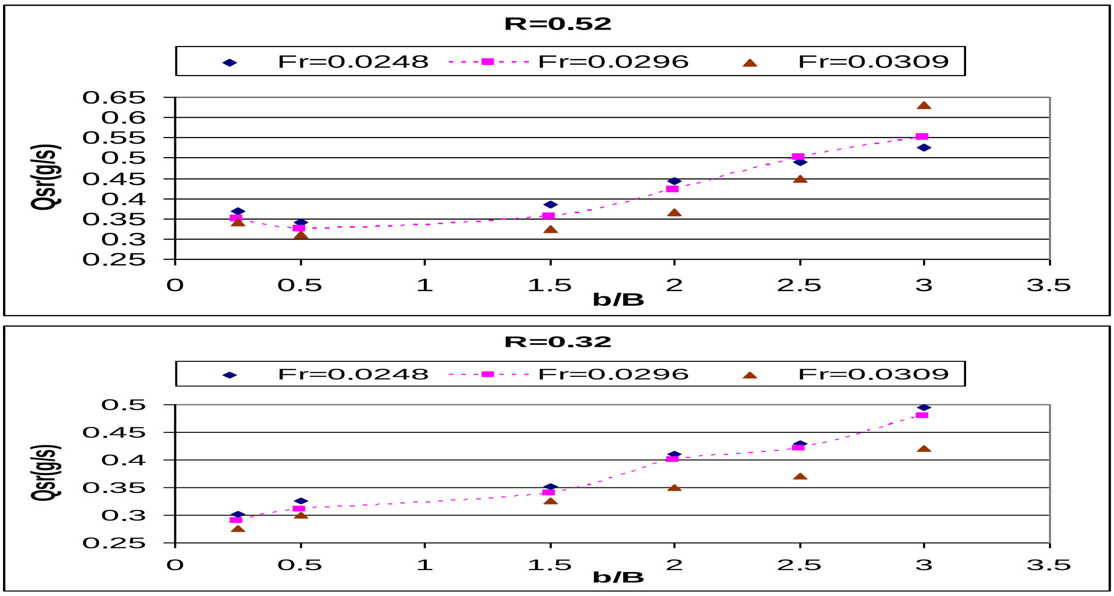


Figure 10. Changes of Q_{sr} entering into the pumping station intake based on various ratios of b/B.

By generalizing the results, the equation 4 is suggested for determining the Q_{sr} entering the into the pumping station intake within the experimental studies conducted at a constant intake angle.

$$Qsr = 0.36(\frac{b}{B})^{0.293}.Fr^{-0.0022}.R^{-0.0417} \qquad r^2 = 0.85 \tag{4}$$

Then, to determine the sediments concentration entering to the pumping station intake we just present the changes of sediments concentration in three depths of 0.1, 0.3 and 0.5 for a constant flow rate of 0.043 m^3/s and the width ratio of 2.5 (b/B) and we ignore the graphic representation due to the lack of space. Figures 11 (a) and 11 (b) represent the sediments concentration in the channel before entering to the intake in different depths and ratios of width 0.5,2,2.5 and 3 and for the inflow ratio of 0.32 in the pumping station.

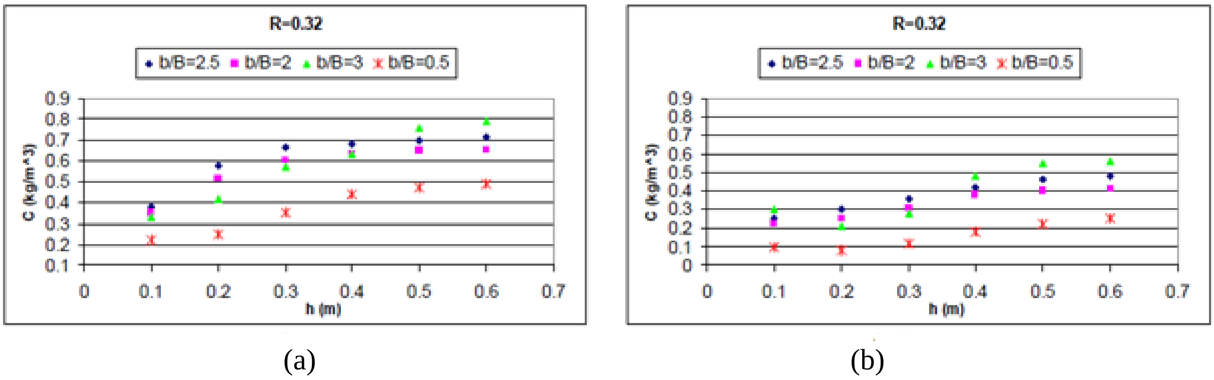


Figure 11. Changes in the sediment concentration in the main channel before the intake channel (a) and the intake inlet (b) based on the various depths

12. CONCLUSION

- The k-ε-standard and k-ε-RNG turbulence models were weak in predicting the flow velocity profile and the vortex inside the intake but the k-ω model which was used as the turbulence model in this study had a better performance due to the ability to predict the flows with lower Reynolds number, the absence of the wall function and the ability to predict the areas affected by the adverse pressure gradient.
- The flow ratio is one of the most important parameters affecting the size of the separation zone and increased Q_{sr}.
- For a constant inflow rate entering to the main channel, by opening the intake inlet the Q_{sr} entering the intake pumping channel increases.
- The decreased intake angle (the upstream wall slope of the station) in addition to decreasing the width and length of the separation zone within the intake channel, it leads to the smoothness of the flow lines at the downside of the inlet and decreased Q_{sr} entering to the intake channel.
- For a constant inflow rate entering to the main channel, by increasing the intake width ratio of the pumping station to the width of the main channel the Q_{sr} entering to the intake channel increases because of the fixed inflow at the same time of increasing the intake width.

REFERENCES

1. Pirzadeh, B. 2007. Numerical Investigation of hydraulic flow in the riverside lateral intakes using the Fluent software. Fourth National Congress on Civil Engineering, Tehran University ,Iran
2. Safarzadeh and SalehiNeyshabouri. 2004. Numerical modeling of tripartite model - dimensional flow in the lateral intake. First National Congress of Civil Engineering, Sharif University of Technology
3. Behzad Pour. 1998. The study of sedimentation Amir Kabir pumping station intake using a physical model. MS Thesis, Sharif University of Technology, Faculty of Civil Engineering, Tehran, Iran
4. Brakdoll, B.D., Ettema.R.O. , Odgaard,A.J. (1998) Sediment Control At Lateral Diversions:Limits And Enhancements To Vane Use. J.HYDR.Eng.ASCE, 124(1), 92-95
5. Emamgholizadeh. , Torabi. (2006) Experimental Investigation Of The Effects Submerged Vanes For Sediment Diversion In The Veis Pump Station. journal of Applied Sciences 8(13).2396-2403
6. Neary and A.J.odgaard (1993).”Experimental studies on flow hydraulic in intakes”.journal of Hydraulic engineering ASCE, 119(2), 94-103
7. Olsen, N.B.R.(2006).A” three-dimensional numerical model for simulation of sediment movements in water intakes with moltiblockoption”.Department of hydraulic and environmental engineering ,the Norwegian university of science and technology.
8. Issa, R, I., and Oliveira, P. J., “Numerical Prediction of Phase Separation in Tow-Phase Flow ThroughTJunctions", Comp. and Fluids, V.23, No. 2, 1994, pp. 347-372.