Larhyss Journal, ISSN 1112-3680, n°23, September 2015, pp. 203-211

EFFECT STUDY OF WATER INTAKING ANGLE THROUGH FLOW REGIME AT STRAIGHT CANAL

AFROUZ A.[1], HEIDARNEJAD M.[2*], AGHAMAJIDI M.[3], JALILIAN H.R.[2]

[1]Department of Irrigation, Dezful Branch, Islamic Azad University, Iran.
[2]Department of water science engineering, Ahvaz Branch, Islamic Azad University, Ahvaz, Iran.
[3]Department of civil, college of engineering, Sepidan Branch, Islamic Azad University, Iran.

mo_he3197@yahoo.com

ABSTRACT

For 3 dimensional analysis of flow regime, numerical solution of nervier-stocks at T shape intake structure and study effect of diverting angle is necessary. For achieving this purpose, and in order to study effect of angle on flow characteristics at side intakes, numerical simulation had been utilized (with using fluent numerical model). In this software, turbulent model of k-e has been utilized. As a result it has been concluded that, this model has acceptable potential power to predict flow behavior through straight channel.

Keywords: Side intake structure, separation zone, numerical solution, fluent Software.

INTRODUCTION

Flow diverting from main channel for water conveying ,had been done by intakes. The most simplest way for water diversion is diverting from river. For this reason, using water intake structure are common rule, but building these system may caused some problems with complicities. Figure (1) shows that inlet flow through side channel has very powerful momentum through main channel (at direction of main channel) which caused a separation zone near had been created inner side wall of intakes. Side pressure gradient, centrifugal and bed shear force are caused imbalance of inlet flow through diverted channel, therefore, secondary flow has been appeared (Neary et al., 1999). Neary and Sotiropoulos (1996) tried to study flow separation zone. At straight channel (zone B), and diversion channel (zone A). They attempt to study vortex flow and they believed that some points which their value of bed shear stress is zero, are characteristic of in taking point from straight channel. Moreover, many different factors such as: bed form, discharge ratio, depth per width (aspect ratio) and angle of diverting have great effect of side in take structure. Choosing best diverting angle may caused flow (with minimum turbulence) and without separation phenomenon, enter to the intakes. Inlet zone, the separation zone behaves differently because of flow stream dividing.

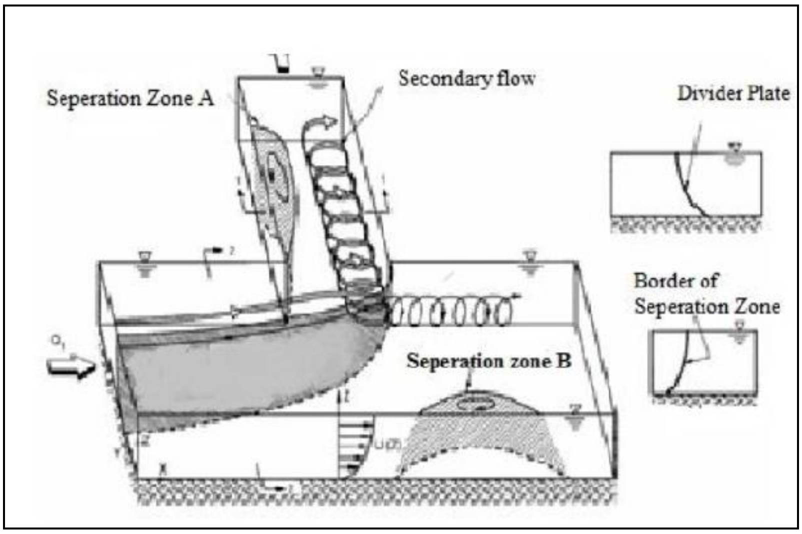

Figure 1: Flow algorithm through intake structure

For the First time, Egyptian engineers (1949) introduced diverting angle as effective factor in order to decrease sediment through diverted channel. Ball (1926) and Schoklich (1937) studied angle of in taking structure but, because of changing location of side intake angle at each project, they didn't introduce specific angle for water in taking (Vanoni, 2006).

Novak (1990) said that the worse angle for water in taking is 90 degree and proposed that angle between 30° to 45° is suitable angle for declining sediment entering side channel. Barkdoll (1997) studied experimentally flow and sediment behavior at 90 degree Branch channel.

MATERIALS AND METHODS

Regulating and messing solution field

In order to achieve first grid at vertical direction of wall, following procedure should be done.
Firstly, cover friction coefficient (Cf) should be calculated.
For this purpose, velocity Shear (U should be estimated, therefore, Y_1 are calculated as:

$$\frac{cf}{2} \approx 0.0359\ Re - 0.2 = 0.004$$

$$Ut = \left(\frac{tw}{f}\right)^{\frac{1}{2}} = Ue\left(\frac{cf}{2}\right)^{\frac{1}{2}} = 0.015$$

$$Y_1 = 50\frac{v}{ut} = 0.0023\ m$$

In which:
Re is Reynolds number and V is fluid dynamic viscosity.
According to calculated Y_1 Grids of net near wall, is very low and small, far from wall growing rate is higher. This meshing is as below:

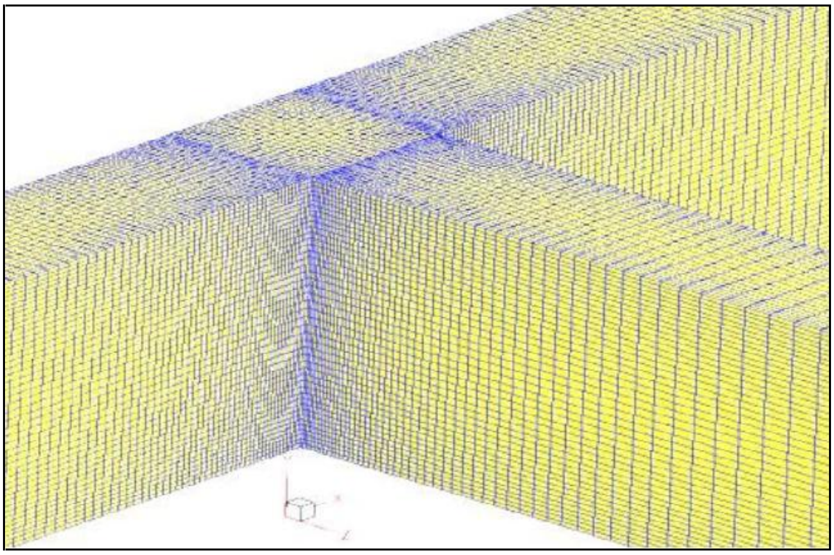

Figure 2: Meshing of solution square

Geometric characteristic of solution square

According to figure (3), physical model of Barkdoll was one T shape flume with (Ratio A/R) IS almost 2, inlet discharge is 11 lit/s, Froude number is 0.13 and Reynolds number (Re) is 50000, therefore, flow is subcritical and turbulent .Inlet discharge of diverted channel per inlet discharge of main channel is about o.31.
In addition, because of development of velocity profile at excrement, for achieving profile, one long channel, with coordinate meshing at width and depth, has been provided, In which: Inlet velocity is about 0.234 m/s.
the average flow depth through total length of canal is 0.31m.
For solving meshing zone, turbulent model of k-e has been used and length of in taking channel increased up to 2.2 meter. The in taking angle, from 15, 30, 35, 45 to 60 degree, have been simulated at this model perfectly.

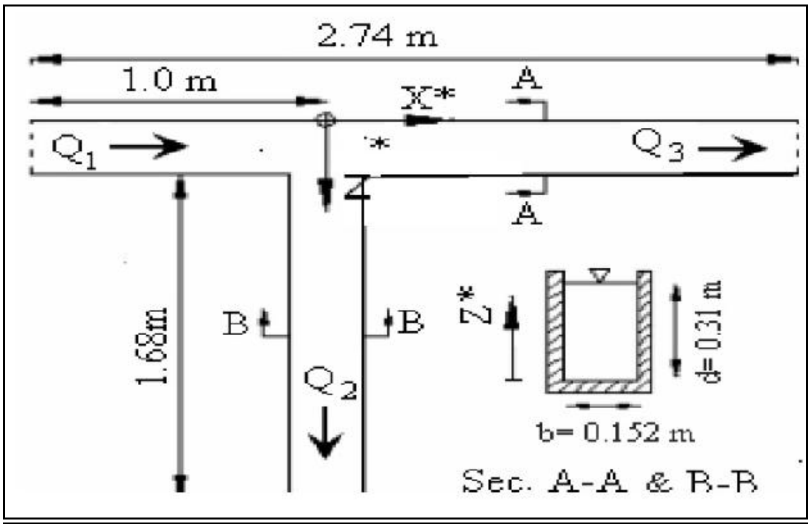

Figure 3: Geometric characteristic flume (2)

Ruling equation

Ruling equations at uncompressible fluid movement with fixed density at turbulent condition is provided with using average value, at time. The continuity and momentum equation are as below:

Continuity equation

$$\frac{\partial u_i}{\partial x_i} = 0$$

Momentum equation

$$\frac{\partial u_i}{\partial x_t} + \frac{\partial u_i u_j}{\partial X_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \mathrm{i} + \frac{\partial}{\partial x_j}(\tau_{ij})$$

In which:
U_i: is velocity element at xi direction, P: total pressure, r: fluid density, G_i: gravity accelerate at xi direction and τij: shear tensor which is known as below:

$$\tau_{ij} = \left[\rho(v + v_t)\left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial X_j}\right)\right] - \left[\frac{2}{3}\rho(k + v_t)\frac{\partial u_i}{\partial x_i}\delta_{ij}\right]$$

Adjusting boundary condition

In order to solve equations, boundary condition should be convinced for inlet of square, Boundary condition is defined as inlet vertical velocity. Moreover, for calculation turbulent parameters; turbulent intensity and hydraulic diameter are choosing 3% and 31% respectively. For outlet of main channel and intake channel Boundary condition is defined. (zero gradient has been used).
Taylor (1944) concluded that if discharge in taking ratio is less than 0.45 and Fr≤0.4 then, flow depth variation near water intakes, at straight direction, is less than 2 %. According to this, neglect able variation, symmetry Boundary condition had been utilized for water surface profile (Neary and Sotiropoulos, 1996).
It should be mentioned, that water level profile is not neglected but this changing indirectly will be used. Boundary condition of wall are defined as rigid Boundary, so, simple algorithm method have been used in order to couple velocity elements and pressure terms. Besides, Apoyand system with degree 2, has been applied for equations movement.

Software introduction

For geometrical generation, meshing and ruling boundary condition of flow, gambit software (2.3.16) has been used effectively. For 3pimensional Analysis, fluent software, version 2.3.16 used too. At the model, for solving square, continuity equation, navies-stocks, for flow analysis were used. for turbulent condition, ruling equation were transformed to Reynolds equation and solving square had been done by finite volume method.

RESULTS AND DISCUSSION

After each run, for each model, in order to study effect of diverting angle on flow regime, flow stream at level of y=.02m, y=.3m for each six models, have been investigated flow division sheet (sheet which distinguish flow regime at intake channel and main channel), separation zone A (zone of separation specialize flow outer which because of zero Shear stress could be a place for sediment deposition), separation zone B: (separation zone of flow at outer edge of flow stream after intake structure near diverting channel), saddle point (zone of flow stream which some part of it direct to this point), this points (saddle points) are located at downstream of side intake structure at bottom of channel, are studied very carefully.

Figures (4) and (5) shows flow stream changing at canal bottom. By increasing angle between flow at channel bottom; width of vortex zone decreased inversely (angle will be increased). Separation zone near bottom are plotted with blue color.

Dimensions of zone at bottom and surface at inner side of channel are completely clear. Orange color shows that from blue to red color, velocity are increased. It’s crystal clear that both of separation zones are decrease when dimensions increase naturally. For instance at angle of 60°degree, seperation zone B is very small it should be considered that maximum Flow are observed at side intakes structure.

Particles near bed profile are come from up stream of main channels. total mount of particles are came from upstream of water intakes and contact to each other at saddle point, which are knowable point for sediment deposition.

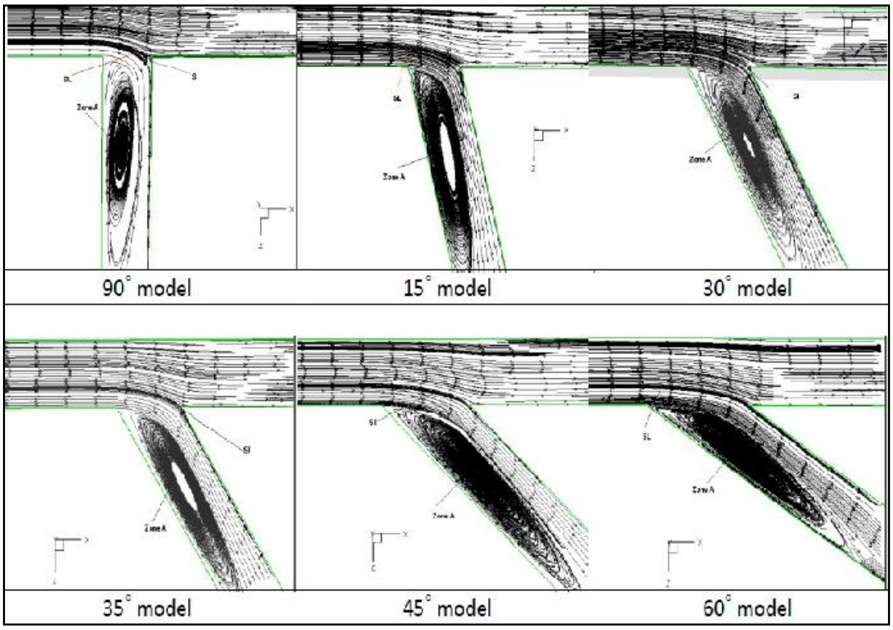

Figure 4: From 0.3 meter above bottom

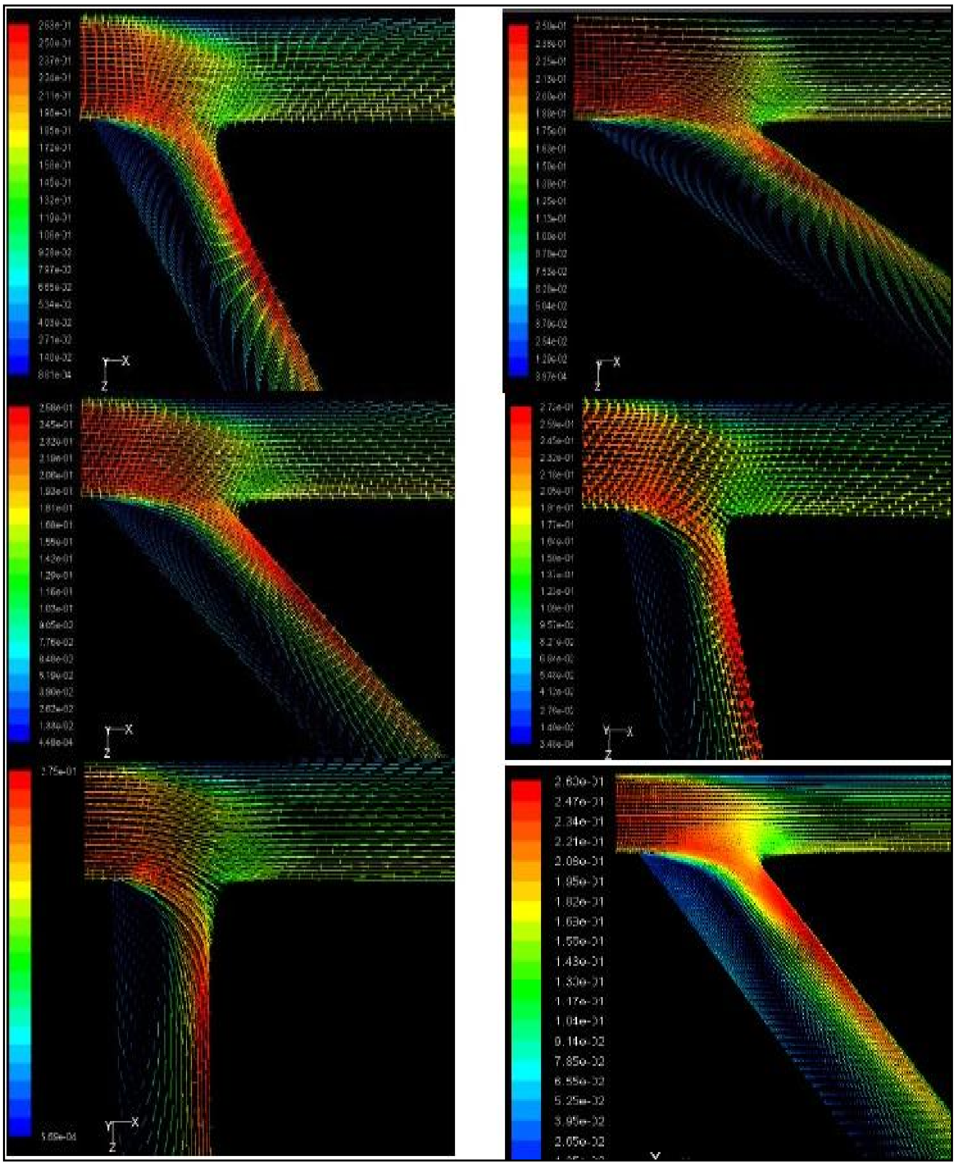

Figure 5: Flow stream at inlet point at 0.02m above bottom with diverting angle: 30, 15, 90° and at left side from down to up diverting angle 60°, 45°, 35°

CONCLUSION

1) Turbulent model k-e is very suitable model for predicting vortex and separation zone.

2) When diverting angle increased, division length of flow is change little from inner point of channel.
3) Increasing diverting angle may cause separation zone decreased normally.

REFERENCES

BARKDOLL B.D. (1999). Sediment Control at Lateral Diversions: Limits and Enhancements to vane Use, J. Hyd. Eng., ASCE, Vol.125, Issue 8, 826-870

NEARY V. SOTIROPOULOS F., ODGAARD A.J. (1999). Three-dimensional numerical model of lateral-intake in flows, J. Hyd.. Eng., ASCE, Vol.125, Issue 2,126-140.

NEARY V.S., SOTIROPOULOS F. (1996). Numerical Investigation of Laminar Flow Through 90-Degree Diversions of Rectangular Cross-Section, J. Comp. Fluids, Vol.25, Issue 2, 95-118.

VANONI A. (2006). Sedimentation engineering: Manuals and Reports on Engineering Practice, N°54, ASCE Publication, New York.

NOVAK P., MOFFAT A., NALLURI C. (1990). Hydraulic Structures, Pitman, London, 546p.