International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-8 Issue-3, September 2019

Flow-field Near Forty-Five Degree Dividing Open Channel

K. K. Pandey, Amiya Abhash, Ravi Prakash Tripathi, Shambhu Dayal

ABSTRACT: Dividing flows are most common in open channel flow in hydraulic engineering system. Turbulent flows are most common through lateral intakes adjoining rivers and canals. Lateral intakes are used for the distribution of water for irrigation system, power plant, public supply etc. Sedimentation remains the most prominent problem in and around the intake structures in water diversion engineering. Sediment entering the water conveyance system leads to reducing the effective length of the waterway and also the closure of entrances of intake structures. Diversion works or intake works are structures provided to draw in water from the main river or channel into conveyance systems for meeting different uses such a irrigation, drinking water requirements etc. In the present study, experimental and numerical modelling study has been made to model an intake at 45^0 and velocity is measured experimentally. These velocities are then compared with the velocity obtained using the FLUENT software in the main and branch channel along with the junction point. The study shows good agreement between the numerical simulation and experimental data. And found the variation in separation zone at different discharge ratio and compare with the separation zone which is found by the previous study around 90^0 water intakes.

Keywords: Dividing channel, Numerical study, RANS Model, Velocity profile

I. INTRODUCTION

The diversion of flow from the main reservoir or river to the conveyance supply system, canals and hydro-power turbines requires the design of water intake. The flow pattern at the water intake is very complex due to flow separation phenomenon. This flow-separation is influenced by the alignment of intake with flow direction in the main channel. In large cases, the water intakes are installed at ninety degrees to the direction of flow in the river or main channel. Various studies in past have been carried out for understanding flow field at these right-angle intakes [1] - [9].

These types of alignments create a large separation zone and hence decreasing the discharge. The optimum angle of installation of intake structure as such is a important criteria for minimizing the separation zone at the intake structure.

Very few studies of flow fields at the intakes aligned at angle other than right-angle have been reported in the literatures. Some flow characteristics of 45 degree channel are shown in Fig.1.

The dividing stream line, near the intake junction, divides the flow into two areas and creating a stagnation zone prone to sedimentation near the downstream corner of the junction. Two separation zones, one in main channel at immediate downstream of junction and other at intake-junction are formed. The zone of separation in the intake reduces the flow area in the intake and thus, this need to be minimized. In our study the angle of intake structure having the minimum separation zone is defined as the optimized water intake structure.

Reference [10] presented the result of an experimental study of flow separation at the water intakes aligned at 45, 56, 67, 79 and 90 degree angles. The result of statistical analysis shows the size of minimum separation zone at 55 degree angle of water intake.

In present study, the experimental and numerical study of 3D flow behavior of a clear water 45 degree lateral channel has been conducted for three different discharges. The experimental data consist of free-surface velocity fields and velocity distributions near the intake junction. A numerical modeling of free-surface turbulence has been developed. Volume of fluid (VOF) scheme has been used under the FLUENT-3D platform to solve the numerical model. The experimental and numerical results of velocity profiles and free surface velocity fields have been compared.

II. EXPERIMENTAL STUDIES

Test setup

The experiment was conducted in horizontal rectangular flumes. The width of main and branch channels were 100 cm and 50 cm respectively. Both the main and branch flumes were made of brick masonry with smooth cement finish. The length of the main channel upstream of the junction was 200 cm and the length of the branch channel was 321 cm. The main channel extended downstream of the junction to a distance of 392 cm. There were two rectangular notches used in main channel. First notch was placed at 160 cm upstream of junction to measure the incoming discharge while second notch was placed at 355 cm downstream of junction to measure the residual discharge in the main channel after feeding the branch channel. Also, a rectangular notch was used in branch channel at a distance 235 cm downstream of the junction. The height of each notch was 9 cm.

Manuscript published on 30 September 2019
* Correspondence Author
K. K. Pandey*, Associate Professor, Department of Civil Engineering, IIT (BHU) Varanasi, 221005, Uttar Pradesh, India Email- kkp.civ@itbhu.ac.in, Mob-9415760115
Amiya Abhash, Research Scholar, Department of Civil Engineering, IIT (BHU) Varanasi, 221005, Uttar Pradesh, India
Ravi Prakash Tripathi, Research Scholar, Department of Civil Engineering, IIT (BHU) Varanasi, 221005, Uttar Pradesh, India
Shambhu Dayal, M. Tech, Department of Civil Engineering, IIT (BHU) Varanasi, 221005, Uttar Pradesh, India

A gate was used in main channel at a distance 256 cm downstream from the junction point for controlling the depth and discharge (Fig. 1).

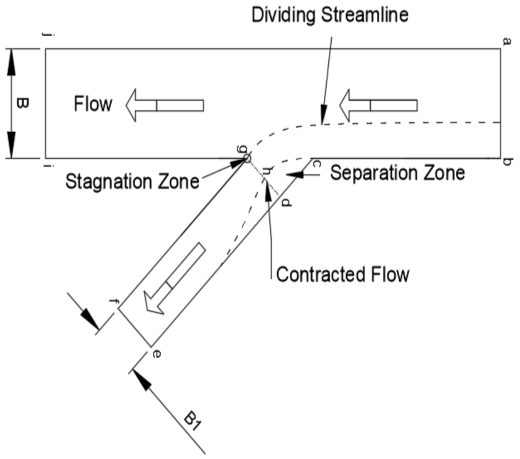


Fig. 1 Flow characteristics near the junction of 45-degree intake channel.

The flow entered the main channel quiescently from a water tank, being fed by pump. The water entered into the pump sump after leaving the main and branch channels. The point gauge was used to measure the water surface elevation and water depths in the channels. A 16-MHZ ADV (SonTek) was used to measure the mean velocity components of the flow fields. Upstream and Downstream Discharges in main channel are denoted by Q_t and Q_m respectively. Discharge in the branch channel is denoted byQ_b. In each run of the experiment, the discharge ratio was maintained at 0.38 (Table I).

Discussion of Experimental Observations

The experiments were conducted for three different discharges as shown in the Table I. Velocity profiles for each of these discharges were measured at three different vertical sections located symmetrically on each of three sections along main and branch channels as shown in the Fig.1.

Table I: Channel discharges flows

Q_t (m^3/s)	Q_b (m^3/s)	Q_m (m^3/s)	$Q_r = Q_m/Q_t$	$V_c = \left(\frac{Q_t^2}{gB^3}\right)^{1/3}$ (m/s)
0.045	0.028	0.017	0.38	0.166
0.054	0.033	0.021	0.38	0.067
0.060	0.037	0.023	0.38	0.072

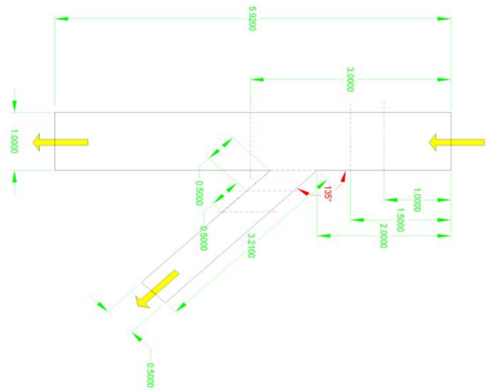


Fig. 2 Experimental set-up (plan view)

Cross-sections MS-1, MS-2 and MS-3 are located on main channel at 1.0m (upstream) ,0.5 m (upstream) and 1.0 m (downstream) respectively from the upstream corner point of the junction. Similarly, cross-sections BS-1, BS-2 and BS-3 are located along the branch channel at 0.0 m, 0.5 m and 1.0 m respectively from the junction in downstream directions. Vertical sections L and R are located at 15 cm from left and right bank respectively at each cross-sections of main and branch channel. The vertical section M is located at mid-point of each cross-section. ADV used in velocity measurement is accurate only for elevations of 5 mm and more from the channel bed. And therefore, experimentally observed velocity profiles have been interpolated linearly from bed to 5 mm stream-elevation from the bed, to compare the numerical results.

The present investigation is conducted on the horizontal rectangular channel with lateral intake at 45^0 for subcritical flow to validate the numerical model. For each run of the experiment, water surface elevations, velocity profile and water depths at different vertical sections, location of surface wave if any, zone of return flow in the branch channel and location of surface and bed dividing stream lines were observed and recorded. Experimentally observed vertical-velocity profile at different sections and velocity-contours at 0.05 m, 0.10 m and 0.15 m elevations from the bed $Q_t = 0.045$ m^3/s discharge, have been shown and compared with the result of numerical simulation, in result and discussion section of this paper.

III. NUMERICAL SIMULATION

Governing Equations

Reynolds-averaged continuity and momentum equations are used for numerical modeling.

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0 \qquad (1)$$

$$\rho\left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij} + \bar{R}_{ij}\right) \qquad (2)$$

Where ρ = density of water; p = pressure; t =time; $\bar{u}_i$=mean velocity in x_i direction; $\bar{\tau}_{ij}$= mean viscous stress tensor; and $\bar{R}_{ij}$= Reynolds stress tensor (i, j=1, 2, 3) defined as

$$\bar{R}_{ij} = -\rho\overline{u_i' u_j'}$$
(3)

Using the Boussinesq's eddy-viscosity concept, the Reynolds stress tensor can be written as

$$\bar{R}_{ij} = \rho\varepsilon_m\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}\right) \quad (4)$$

The proportionality $\varepsilon_m$is called eddy viscosity and it has the dimension similar to kinematic viscosity ν used as proportionality constant for defining the mean viscous stress tensor

$$\bar{\tau}_{ij} = \rho\nu\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}\right) \quad (5)$$

The $k-\varepsilon$ model of Jones and Launder (1972) is most popular and widely used two-equation eddy viscosity model.

This model provide two equations as:

Turbulence kinetic energy equation

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_k}\left[\left(\nu+\frac{\varepsilon_m}{\sigma_k}\right)\frac{\partial k}{\partial x_k}\right]+\varepsilon_m\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}\right)-\varepsilon \quad (6)$$

And *dissipation equation*

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_k}\left[\left(\nu+\frac{\varepsilon_m}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_k}\right]+c_{\varepsilon 1}\frac{\varepsilon}{k}\varepsilon_m\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}\right)-c_{\varepsilon 2}\frac{\varepsilon^2}{k} \quad (7)$$

Here k and ε represent respectively rate of change of turbulence kinetic and dissipation energy. In Eq. (6) and (7), ε_m is modeled as

$$\varepsilon_m = c_\mu\frac{k^2}{\varepsilon} \quad (8)$$

The value of the parameters used in above equations are: $c_\mu = 0.09$, $c_{\varepsilon 1} = 1.44$,$c_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$.

Boundary condition and Initial Boundary conditions

The inlet boundary condition was set as 'velocity inlet' according to the discharge flowing in the main channel. Volume fraction at inlet was specified as water being 1 and air being zero. The boundary condition of the outlet of the channel was set as a 'pressure outlet' with zero gauge pressure and backflow volume fraction of air as 1 and zero for water. Initial inlet mean velocity is calculated after assuming initial flow depth for given total flow rate. The depth and velocity at the inlet changes with increasing time steps. Initial pressure head is assumed to be hydrostatic.

Numerical Algorithm

Numerical modeling is used to solve the standard 3D incompressible Reynolds-averaged Navier Stokes (RANS) equation. The turbulence model was selected as (k-ε) turbulence on FLUENT Computational Fluid Dynamics (CFD) platform. The RANS equations are solved with SIMPLE algorithm while discretizing using control volume approach..

Solution Procedure

The arrangement for numerical study is same as shown in the Fig.2. The main channel length upstream of the junction is 10 m (not shown in the Fig.2) which is taken as ten times the width of the channel to ensure uniform inlet flow conditions. The length of the branch channel from the junction is 3.21 m which is 6.42 time the width of the channel.

Grid geometry in ANSYS has been generated as shown in the Fig. 3. Meshing is done using power law function. The grid cells are populated in turbulent region near channel boundary. The variation or distinction between coarse and fine grid predictions is very minute or negligible. This detail confirms that the results are not dependent on the mesh size used in the refinement. The coarse mesh as such is enough to capture the important flow characteristics within permissible limits of error in the solutions.

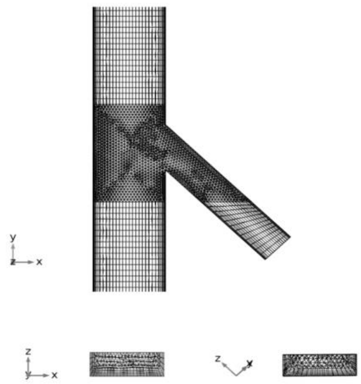


Fig.3. Grid geometry near the intake.

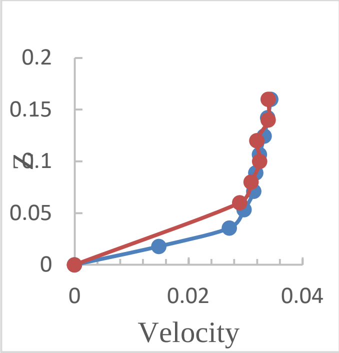
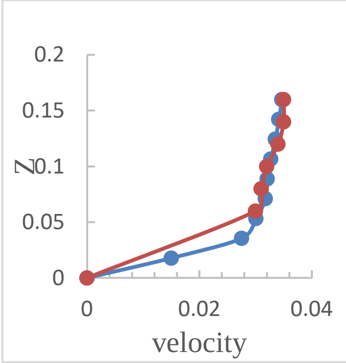
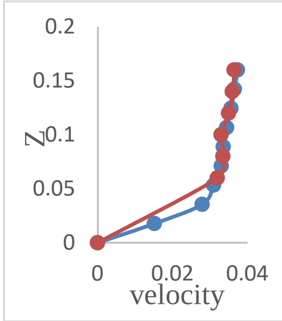
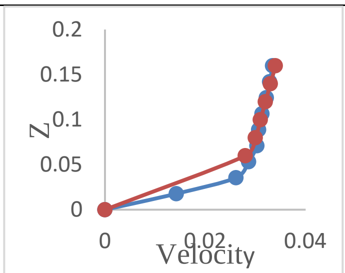
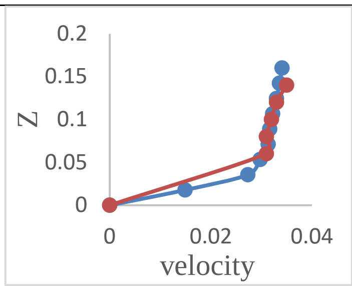
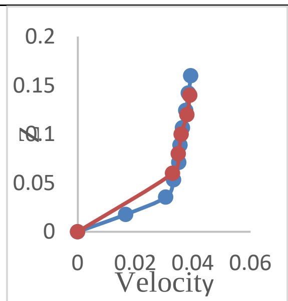
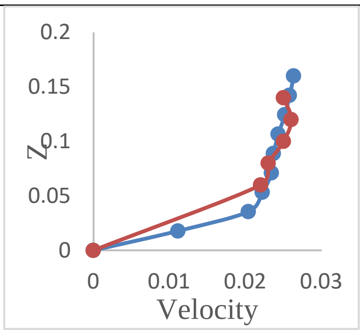
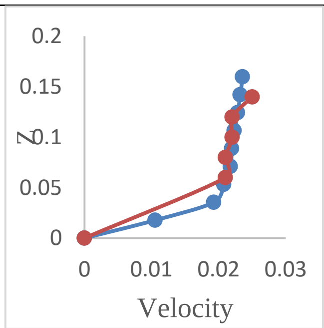
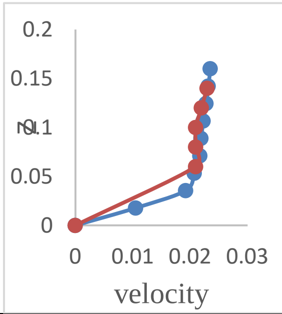
IV. RESULT AND DISCUSSION

Vertical Velocity Profiles

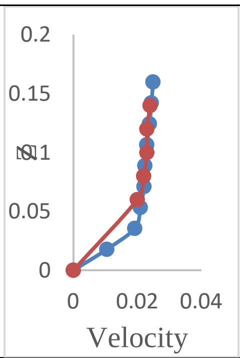
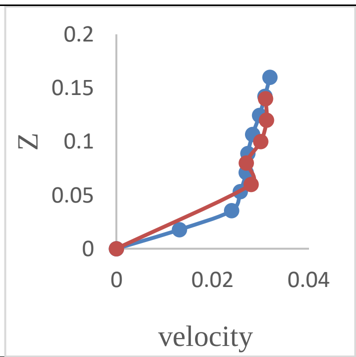
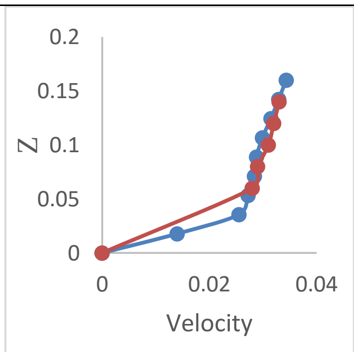
Experimentally observed velocity profiles at different vertical sections for $Q_t = 0.045$ m^3/s have been shown and compared with numerical result (Fig. 4). The result shows that measured velocity profile closely follows velocity profile obtained by proposed numerical solution.

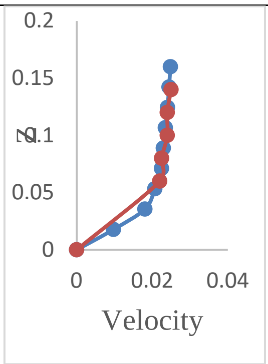
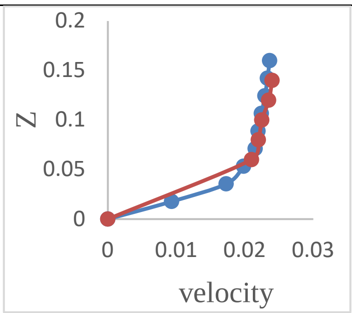
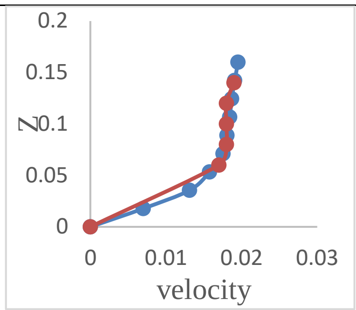
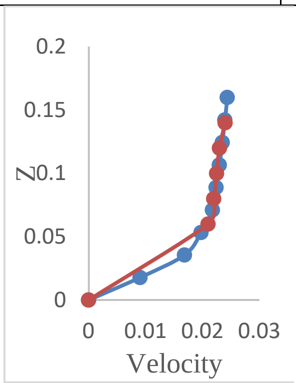
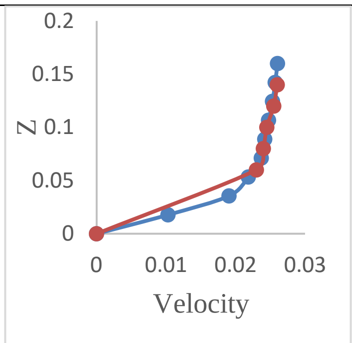
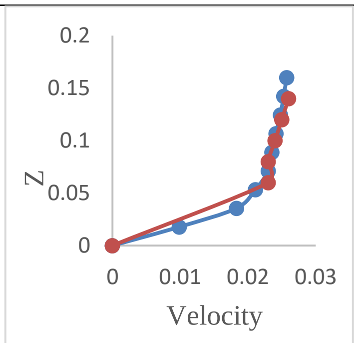
Velocity Contours

Figure 5 shows contour plot for velocity at depths of 0.05m, 0.10m and, 0.15m depth from bed surface for $Q_t = 0.045$ m^3/s. Experimental and numerical simulation studies shows that an oblique type separation zone, which cause contracted flow region, is formed in the branch channel immediately downstream of the junction. It is observed that as the stream-line elevation increases, width of separation zone is increasing. This phenomenon gives rise to secondary flow responsible for recirculation of water in this zone. Maximum velocity zone is observed just upstream of the junction instead of the branch channel as observed for 90^0 diversion channels. This phenomena is attributed to effect of separation zone in main channel. In Fig (5c) it can be observed that the peak of separation zone is pointing towards main channel and thus effect of maximum contraction is action in the main channel thus there is increase in velocity.

ain-Channel Sections	Vertical-Sections L	M	R
MS-1			
MS-2			
MS-3			

(a) Main Channel

Main-Channel Sections	Vertical-Sections L	M	R
BS-1			

BS-2			
BS-3			

(a) Branch Channel.

Fig. 4 Velocity Plot at different sections for $Q_t = 0.045$ m^3/s

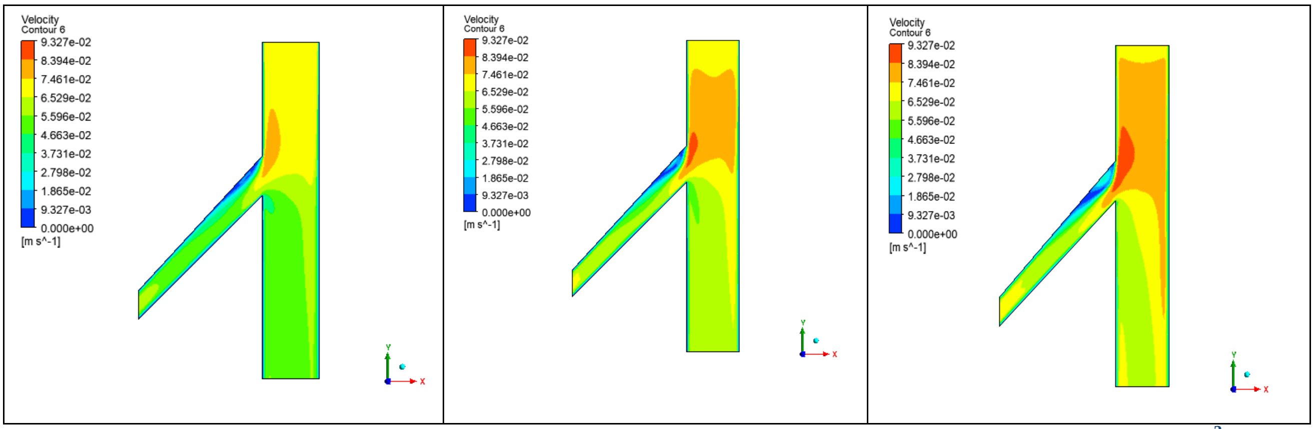
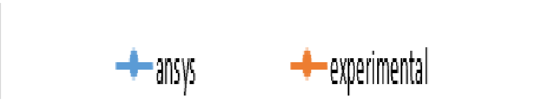


Fig.5 Velocity contour at 0.05m, 0.10m and, 0.15m depth from bed surface for $Q_t = 0.045$ m^3/s.



V. CONCLUSION

The study shows that increasing the discharge ratio lead to reduction in the length and breadth of the separation zone. Therefore, position and the extent of the flow separation zone depends on the discharge ratio. The separation zone near the surface is larger in comparison to its dimension closer to bed. The flow expansion occurring along the main channel at downstream portion of the intersection also results in creation of separation zone at the junction. For this kind of separation zone, the dimensions of the separation zone increases with increased discharge ratio. Momentum transfer takes place in between the streamlines because of the difference in velocities especially at places where streamlines are very close. This momentum transfer leads to generation of a series of eddies which travel downstream of the intake releasing the energy of the flow along the intake. This results in reduced flow rate in the branch channel.

In a water intake positioned at ninety degrees, the separation zone is formed at upstream of the water intake, whereas in a water intake placed at forty-five degrees, separation zone occurs at the downstream portion.

The extents of separation are very large at 90^0 and gradually reduces to half the width of water intake. At the position of intake having 45^0 angle, the length of separation zone is small as compared to the intake positioned at $90^{0.}$ The dimensions of separation zone is comparatively large along downstream, for the intake positioned at 45^0, and lesser in upstream. This implies, that the position of separation zone shifts from upstream to downstream portion when the intake changes from 90^0 to 45^0. For a water intake positioned at 90^0, the size of separation is huge upstream, whereas it reduces considerably for an intake positioned at 45^0. Formation of eddy is more at the junction of the channel. Due to increase in length ofseparation zone sedimentation is more in 90 degree intake in comparison with the 45 degree intakes.

The two and three-dimensional CFD modeling on the numerical model results are employed to estimate the velocities profiles for downstream main channel in 45° junction with the fully turbulent flow and comparisons between numerical simulations and the measured experimental velocities with different discharge have been made.

Velocity profile changes in the main channel. There is sharp increase in turbulence at intake. Effective viscosity is highest at the center of the channel. As comparison with the 90 degree intake 45 degree intake is better due to lesser length of separation zone in 45 degree intake, due to this reason sedimentation is lesser than 90 degree intake.

REFERENCES

1. Chen H. And Lian G. (1992) "The numerical computation of turbulent flow in tee-junctions" Journal of Hydrodynamics, Series B, No.3, pp. 19-25.
2. Grace J. L., And Priest, M. S,(1958) "Division of flow in open channel junctions", Bulletin No. 31, Engineering Experimental Station, Alabama Polytechnic Institute.
3. Hager, W. H. (1989), "Transitional flow in channel junctions." Journal of Hydraulic Engineering, ASCE, Vol. 115(2), pp. 243-259.
4. Huang J., Weber L. J., Lai Y.G. (2002) "Three dimensional numerical study of flows in open channel junctions", Journal of Hydraulic Engineering, Vol. 128(3), pp. 268-280.
5. Law and Reynolds. (1966)."Division of flow in an open-channel". Journal of Hydraulic Engineering. ASCE, 192(2), pp. 207-231.wo
6. Maia, R. J., Costa, N. P., Proença, M. F. and Pinho, F. T., (2000). "Asymmetric diverging flow on 1 90° tee junction", Proceedings of the Tenth International Symposium on Applications of Laser Techniques to Fluid Mechanics, 10-13 July 2000, Lisboa, Portugal.
7. Ramamurthy A. S., Junying Qu And Diep Vo. (2007), "Numerical and Experimental study of Dividing open-channels flows", Journal of Hydraulic Engineering, Vol. 133(10), pp. 1135-1144
8. Taylor, E. H. (1944), "Flow characteristics at rectangular open-channel junctions." Trans. ASCE, 109, pp. 893–902
9. H. Shamloo and B. Pirzadeh, "Numerical investigation of velocity field in dividing open-channel flow," in *Proceedings of the 12th WSEAS International Conference on APPLIED*, 2007.
10. A. Keshavarzi and L. Habibi, "Optimizing water intake angle by flow separation analysis," *Irrigation and Drainage: The journal of the International Commission on Irrigation and Drainage*, vol. 54, pp. 543-552, 2005..

AUTHOR'S PROFILE

Dr, K.K.Pandey is Associate Professor in Civil Engineering Department, IIT (BHU) Varanasi. He has done his M. Tech from IIT Kanpur and PhD from IIT (BHU) Varanasi. His area of expertise involves Concrete fracture mechanics and Hydraulics & water resource engineering. His core academic teaching includes hydrology, open channel flows, dividing channels, irrigation engineering, dams, numerical simulations and experimental and mathematical modelling. He has numerous papers in national and international publications and conference proceedings.

Amiya Abhash is a currently pursuing his PhD in Hydraulics and water resource engineering from Department of Civil Engineering, Indian Institute of Technology (BHU), Varanasi, India. His doctoral thesis work is on Experimental and numerical study of flows around Piano Key Weirs. His area of research includes hydraulics of open channel flows, numerical and experimental modelling and weirs.

Ravi Prakash Tripathi is currently pursuing his PhD degree in Hydraulics and water resource engineering from Indian Institute of Technology (BHU), Varanasi, India. His area of research includes study of scouring and flow field around spur dike in River Meandering channel. His area of research interest include hydraulics of open channel flows, river meandering, turbulence and sediment transport.

Shambhu Dayal received his post graduate degree in Hydraulics and water resource engineering from Indian Institute of Technology (BHU), Varanasi, India in 2013.He is currently working in RDSO Lucknow, India.