Transaction B: Mechanical Engineering
Vol. 17, No. 4, pp. 241–252

Experimental Investigation of Baffle Effect on the Flow in a Rectangular Primary Sedimentation Tank

H. Jamshidnia[1] and B. Firoozabadi[1,*]

Abstract. *In primary sedimentation tanks, short-circuiting enlargement of dead zones and high flow mixing problems are caused by circulation regions (dead zones), which can reduce the optimal sedimentation of particles. For proper design of such tanks, the formation of recirculation zones should be avoided. The provision of a baffle as a geometrical modification of a tank may influence the flow field for better sedimentation. Thus, in this study, velocity measurements were performed by a three-dimensional Acoustic Doppler Velocimeter (ADV) to investigate baffle effects on the velocity distribution in a primary rectangular sedimentation tank, quantitatively. Effects of baffle positioning were also determined. Mean flow analysis shows how a baffle can alter the hydrodynamics of the flow field. It was quantitatively found that the intermediate baffle not only influences the flow field in its downstream, but also affects the flow pattern in its upstream. It was found that the baffle setting and its position relative to the inlet and outlet influences the flow field and the development of flow. Baffled flow may provide better conditions for sedimentation by influencing velocity profiles. However, further detailed experimental study is necessary to fully capture the baffle effect and obtain further insight into the complex flow field in a sedimentation tank.*

Keywords: *ADV; Intermediate baffle; Sedimentation tank; Primary settling tank; Reverse flow region.*

INTRODUCTION

Sedimentation by gravity is a usual and important process in settling tanks to remove inorganic settleable solids from water and waste water in refinery plants [1].

Numerous studies show that in order to remove suspended solids with minimum cost, they should be removed as quickly and efficiently as possible from the water. In fact, if the removed solid concentrations from settling tanks are increased in order to increase treatment efficiency, the size of water treatment facilities, which are located downstream of the clarifiers, can be reduced [2]. According to the investigations of Camp [3] and Swamee and Tyagi [1], the investment costs of settling facilities contribute to a large portion (typically one-fourth to one-third) of the total cost of treatment plant construction. For that reason, significant savings in both capital and operational costs at various stages of treatment can be expected by increasing solid removal efficiency [2]. As a result, increasing removal efficiency is important.

Many factors can influence removal efficiency, including tank hydraulics, which are of great significance [4]. It is noticeable that the ability of a sedimentation tank to remove suspended solids depends on its flow field. Therefore, investigating the structure of the flow field is of great importance [5].

Settling tanks are divided into two main categories: primary and secondary (final) sedimentation tanks. It is worth making a distinction between these types of tank. Primary sedimentation tanks are designed to reduce the particulate flow velocity and provide for the settling of organic solids [6]. The sludge in these tanks is not activated [6], hence particle concentration is low and there is not a large difference between particle sizes [7,8]. As a result, the flow is not much influenced by concentration distribution [8]. But in a final sedimentation tank, the particle con-

1. *Center of Excellence in Energy Conversion, School of Mechanical Engineering, Sharif University of Technology, Tehran, P.O. Box 11155-9567, Iran.*
*. *Corresponding author. E-mail: firoozabadi@sharif.edu*

Received 14 February 2009; received in revised form 12 February 2010; accepted 18 May 2010

centration in settled sludge is relatively high resulting in significant density effects [7,8]. Also, a wide range of particles with various sizes can be found [8]. In a prototype test, Anderson compared the flow fields in primary and final settling tanks of similar geometries and hydraulic loadings. In the primary clarifiers, flow was observed to be along the surface from inlet to outlet [9]. On the contrary, sludge concentration in the secondary-clarifier inlet resulted in a density current along the bottom, causing reverse flow at the surface [7]. Furthermore, in a secondary clarifier, particles flocculate, which produces larger particles, causing an increase in particle settling velocity [10].

Attempts have been made to improve flow conditions using two approaches: manipulating the inlet structure, and installation of solid or perforated walls or baffles in sedimentation tanks [7]. The construction of baffled channels in water treatment plants with no mechanical parts is considered to be more reliable, requires less maintenance and hence reduces cost [11].

Various studies have been done on the effects of baffles on the flow and hydraulics of settling tanks and the major findings of these studies are presented here. Lyn and Rodi [12] conducted turbulence measurements by Laser Doppler Anemometry (LDA) in a rectangular laboratory settling tank and studied the flow fields from a plane jet impinging on two types of deflector. Experimental studies of Bretscher et al. [13] on velocity and concentration fields in a rectangular settling tank showed the effectiveness of a central barrier wall. Krebs [14] investigated the effect of the inlet and intermediate baffles on the flow field in final clarifiers. His research was based on experiments, numerical modeling and analytical relations. Other experimental investigations performed by Taebi-Harandy and Schroeder [15] on primary sedimentation tanks showed that the placement of an intermediate baffle, installed close to the middle of the tank and extended from the bed upward to one-third depth, had no significant effect on efficiency. They believed that the discrepancy between the results of their study and other works is likely due to the difference in flow patterns. This means that if the dominant current is a surface current, a baffle extending from the top upward may improve the velocity field and, accordingly, the solid removal efficiency [15]. Ahmed et al. [16] studied the effects of the position and height of the baffle in a secondary sedimentation tank by placing the baffle at three different positions and various heights, qualitatively. The best result was for the case in which an inlet baffle of the height of 67% of the total depth was placed in the first 5% of the channel [16]. In primary sedimentation tanks, short-circuiting, enlargement of dead zones and high flow mixing problems are caused by circulation regions known as dead zones, which in turn can reduce the optimal sedimentation of particles. Thus, for proper design of such tanks great care should be taken to avoid the formation of circulation zones. Tamayol et al. [14] simulated the effect of a simple baffle at various positions. Their results showed that when the baffle is located in the recirculation region, the dead zone (recirculation regions) would be reduced in size. Moreover, if the baffle is located in an improper position or if it has an improper height, the particle removal performance in the primary sedimentation tank would be decreased drastically [17,18]. From a review of previous research investigations, it is clear that there is no universal agreement between researchers over the existence and placing of the intermediate baffle. The lack of detailed measurements of flow field characteristics in sedimentation tanks was revealed by Lyn and Rodi [12]. Therefore, there is a need to perform an experimental study to determine the proper position and height of an intermediate baffle in a settling tank.

Considering the complex features of a flow field and the lack of detailed experimental data, this study is undertaken to obtain a better understanding of the flow field and to evaluate the influence of intermediate baffle on the velocity distribution in a rectangular primary sedimentation tank in a quantitative manner. It was realized that by using an Acoustic Doppler Velocimeter (ADV) the velocity field could be obtained non-intrusively and in a quantitative manner.

EXPERIMENTATION

Experimental Apparatus

An open channel loop at the Fluid Mechanics Laboratory of the Mechanical Engineering Department of Sharif University of Technology (Iran) that was originally designed to demonstrate the hydraulic characteristics and performance of an open rectangular sedimentation channel, was utilized for the experiments. A schematic sketch of the experimental set up is shown in Figure 1. As shown, inflow water was taken from the laboratory main supply to the cylindrical supply tank and, then, was pumped by the recirculation pump to a constant head tank. The role of the constant head tank is to keep the flow rate unchanged during experiments. Water moves from this tank to the main channel while passing through a flow meter. A multi-layer damping screen was installed at the inlet to distribute the inflow as uniformly as possible, to achieve uniform flow distribution along the inlet width and avoid entrance disturbances.

Experiments were conducted in a rectangular open channel 8 m $\times$ 0.2 m $\times$ 0.4 m in length, width and height (x, y and z with $x = 0$ at the upstream end, $y = 0$ at the center of the channel and $z = 0$ at the bed), respectively, with a smooth bottom. A rectangular

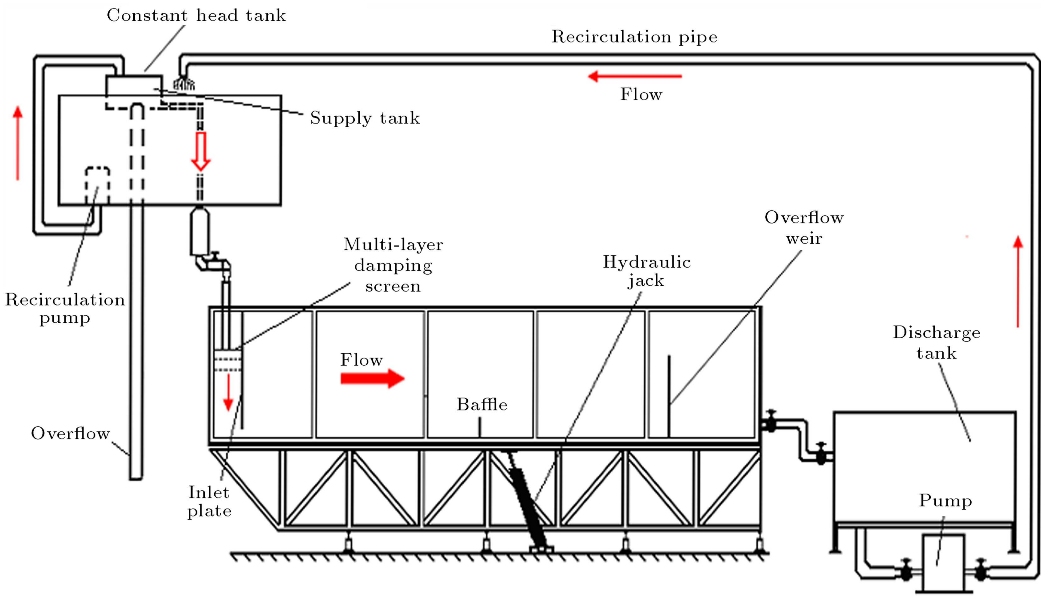


Figure 1. Schematic sketch of the experimental setup.

bottom feed slot with a height of $h_0 = 0.01$ m extending throughout the full width of the channel provided the inlet gate. The depth of water was controlled by a sharp-edged weir of a height of 32 cm located at the downstream end of the channel. A typical photograph of the installed baffle is shown in Figure 2.

Velocity Measurement Using Acoustic Doppler Velocimeter (ADV)

Flow velocity components in a 3D direction at each spatial point of the channel were measured by a three-dimensional Nortek Acoustic Doppler Velocimeter (ADV). It is noted that all the measurements have been performed on the vertical central plane and, as a result, the 2D flow can reasonably be assumed on this plane.

Figure 2. A photograph of installed baffle.

The ADV operates based on the principles of a Doppler shift of a sound wave reflected from moving particles suspended in the water [4] and is considered a proper choice for velocity measurements [19]. The ADV emits a burst of sound energy of a known frequency that is reflected back to the ADV probe by moving particles within the fluid stream. Consequently, a shift in frequency of the backscattered sound energy, known as a Doppler shift, occurs. Flow speed is proportional to the magnitude of this shift. The probe of the ADV consists of a central transmitter surrounded by three equally spaced receivers [20]. ADV is capable of measuring velocity accurately using a sampling volume as small as 0.254 cm^3 [20]. The sampling volume is located at the intersection of the transmit and receives beams from either 50 or 100 mm away from the probe (in this study 50 mm away from the sensor) to provide undisturbed measurements. As a result, the ADV is capable of measuring the velocity components reliably in a non-invasive way [20,21]. A schematic of ADV transducers and its operation method is illustrated in Figure 3.

The ADV is more advantageous compared with other forms of velocity measurement such as propeller, electromagnetic or hot-film sensors [20]. Furthermore, the ADV method is highly accurate and is not sensitive to water quality, which allows for a wide range of applications [21].

In our experiments, data were collected for about 20 to 30 s at each point at the maximum available sampling rate of 25 Hz. The 3-D velocity range is 2.5 m/s, and the velocity output has no zero-offset [21].

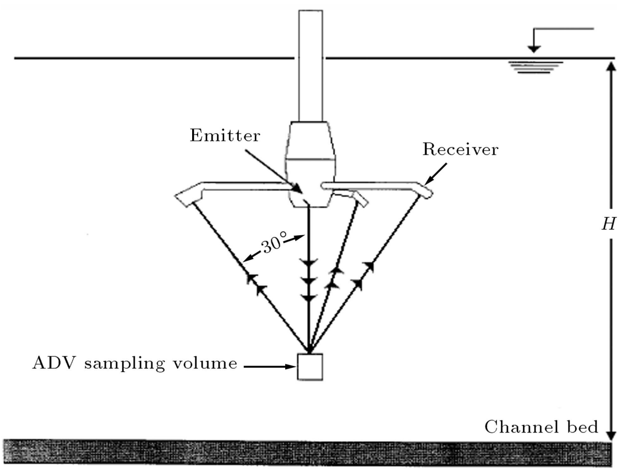


Figure 3. Schematic of ADV transducers and its operational method.

Since downward-looking ADV probes were used, velocity measurements were performed from the top of the channel and continued by dipping the probe to the channel bottom. About 14 positions were measured to get the representative velocity profile. These series of experiments were carried out to obtain instantaneous velocities and time-averaged velocity profiles.

Figure 4 shows the channel and the test sections. Two ADV probes were mounted separately on a carrier with 1.1 m distance. Thus, measurements at each pair of sections ((x = 1.5 m, 2.6 m), (x = 3.5 m, 4.6 m) and (x = 5.5 m, 6.6 m)) were performed simultaneously. The probes were easily transferable in vertical and horizontal directions. The ADV signal conditioning module was connected to a personal PC during measurements. Real-time observation of the velocity time series was made possible (in tabular and graphical forms) by data acquisition software supplied by the manufacturer.

It should be noted that due to a low concentration of scattered particles, ADV measurements are noisy when using clean tap water and measuring is more challenging. In order to overcome this problem, seeding particles provided by the Nortek Company were added as tracers to the tap water. The density of these particles was almost the same as water density and their average diameter was about 10 μm. For uniform distribution, seeding particles were mixed properly with an adequate amount of water and then were injected in the inlet region upstream of the damping screen (see Figure 1).

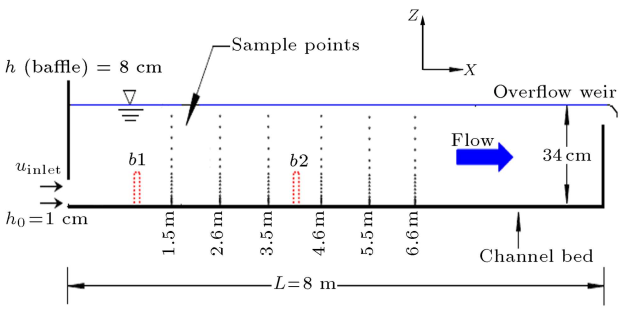


Figure 4. Schematic of open channel and measurement sections.

Experimental Methods

Experiments were performed to investigate the effect of a baffle on the flow structure at different flow rates in a rectangular sedimentation open channel by measuring flow velocities. The experiments were conducted in two phases. In the first phase of the experiments, the velocity of the pure water flow was measured at the flow rate of $Q = 42$ l/min. This flow rate was the maximum established flow rate that could be achieved and, thus, was termed as full flow rate. In the second phase, the same procedure has been used for half of the full flow rate (e.g. $Q = 21$ l/min). This lower flow rate was selected to obtain a better understanding of the effect of flow rate and inlet Froude number.

In each phase of the experiments, two intermediate baffle configurations (one or none) were used. The height of the baffles was fixed to almost one fourth of the total water depth, $h_b = 8$ cm. To achieve a better investigation of the influence of the intermediate baffle on the hydraulics of the flow, its location, denoted by x_b which is a structural factor, was changed twice. They were placed individually at $x_b = 1.2$ m and $x_b = 4$ m from the inlet and are indicated in Figure 4 by $b1$ and $b2$, respectively. They are termed as inlet region baffle ($b1$) and intermediate baffle ($b2$) respectively. It should be noted that the placements were not done simultaneously.

The channel details, baffles and measurement sections have been schematically shown in Figure 4.

Table 1 summarizes flow conditions in the current experiments. The value of important non-dimensional quantities, such as Reynolds (Re) and Froude (Fr) numbers, at the inlet and in the channel has been illustrated for two flow rates, respectively. The Reynolds and Froude numbers are defined by the following relationships:

$$\mathrm{Re} = U_{ch} L_{ch} / \nu, \tag{1}$$

$$\mathrm{Fr} = U_{ch} / (g L_{ch})^{0.5}, \tag{2}$$

in which L_{ch} and U_{ch} are characteristic length and velocity, respectively. At the inlet, Re and Fr numbers are defined in terms of inlet height (h_O) and inlet bulk velocity. ν is the kinematic viscosity of the water at $T = 20^\circ$C. The Reynolds number inside the channel is calculated based on the average bulk velocity inside the channel and the hydraulic diameter of the channel.

Table 1. Experimental conditions.

Q (l/min)	Re (In the Channel)	Re (Inlet)	Fr (In the Channel)	Fr (Inlet)	Baffle Position (m) from Inlet
42	3168.2	3486	0.0056	1.117	No baffle
42	3168.2	3486	0.0056	1.117	1.2
42	3168.2	3486	0.0056	1.117	4
21	1584.1	1743	0.0028	0.558	No baffle
21	1584.1	1743	0.0028	0.558	1.2
21	1584.1	1743	0.0028	0.558	4

Although the inlet Froude numbers are greater than one (supercritical flow) and less than one (subcritical) for cases of full and half flow rate, respectively, they are very small inside the channel and, thus, the effect of free surface can be neglected. Despite this small number, the similitude of Froude was used, similar to Lyn and Rodi [12]. In the absence of sediment transport or density differences, the Froude and Hazen numbers have no significant effect.

The inlet Re number indicates that the inlet flow is turbulent. Similar to the work of Lyn and Rodi [12] the traditional hydraulic modeling criteria has been borrowed from pipe studies and Re $>$ 2000 was considered the criterion for turbulent flow. Thus, the values of Re number in the channel indicate that the operating flow remains turbulent, regardless of baffle position or value of Fr number.

RESULTS AND DISCUSSION

A typical time series of the streamwise velocity component, $u(t)$, measured by ADV at one point during the experiments has been shown in Figure 5.

It is noted that the ADV data acquisition program makes it possible to monitor the correlation values and Signal to Noise Ratio (SNR) for each ADV receivers to observe and make sure of the quality of data during data acquisition.

One of the disadvantages associated with ADVs is the appearance of spikes caused by the aliasing of the Doppler signal in the ADV time series [19], thus to obtain reliable results, the data should be checked before any analysis and interpretation. It is noticeable that these spikes can be reduced or eliminated by adjusting probe operational parameters but cannot be avoided totally [22]. Considering this constraint, an algorithm should be used to detect and eliminate these spikes.

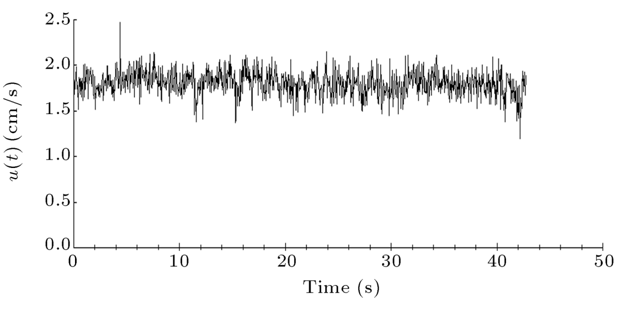


Figure 5. A typical one-point time series of streamwise velocity measurement.

All the spikes were removed by freeware WinADV computer program written by Tony L. Wahl for analysis and post processing of ADV datasets [23].

It should be noted that for calculating the time-averaged velocities at each point, first the time series of the velocity at that point has been filtered. Filtering can be done by removing low correlation or low SNR values. In general, if the instantaneous velocity values are not removed due to poor correlation (less than 0.7), a bias can be introduced into both velocity values and turbulent statistics. In this study, a high percentage of the measured data had a high correlation and the removal was achieved with a high degree of reliability.

In all velocity distribution graphs in this paper, the horizontal axis represents the ratio of time-averaged streamwise velocity (U) to average bulk velocity (U_{Ave}) in the channel, and the vertical axis is the ratio of height at each measurement point (h) to the total flow depth (H = 34 cm). It is noticeable that data acquisition near the free surface was not possible, because if the downward-looking probe of ADV were out of water, the change of sound velocity in the air and water would lead to poor quality data. Thus, the measurements could be performed only up to $h/H = 0.8$.

Furthermore, to obtain reliable results and to understand the accuracy of the results, each experiment was repeated several times under identical conditions, and the range of time-averaged velocity data at each point was studied. Typical results of the arithmetic average and the range of variation of time-averaged velocity values between their maximum and minimum is shown at each spatial point in Figure 6. This range of velocity variation shows the scattered velocity at each point. The amount of scatter near the bottom

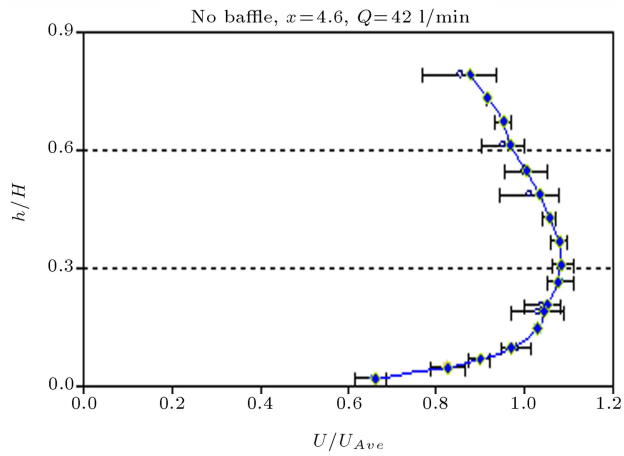


Figure 6. Range of velocity variation with respect to height.

is observed to be wider compared to the other points. This can be described as a result of the probable interference of the sampling volume with respect to the channel bed. In general, the scatter was higher near the channel bed and near the free surface.

Additionally, to verify the accuracy of the results, the arithmetic average of time-averaged streamwise velocities were compared with the arithmetic average of their maximum and minimum values at each point. A typical result is shown in Figure 7. As shown, the results of both types of averaging are very close to each other and proved to be reliable.

The time-averaged streamwise velocity profiles at 6 different sections from the inlet and in the absence of a baffle are illustrated in Figures 8 and 9 for cases of full and half flow rate, respectively.

Comparison of velocity profiles along the channel in the absence of a baffle (Figures 8 and 9) shows that the velocity is negative at the bottom part of the

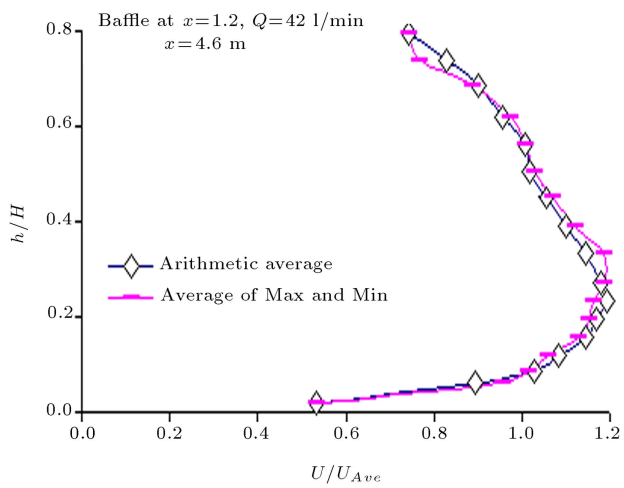


Figure 7. Comparison of arithmetic average and average of max and min velocity.

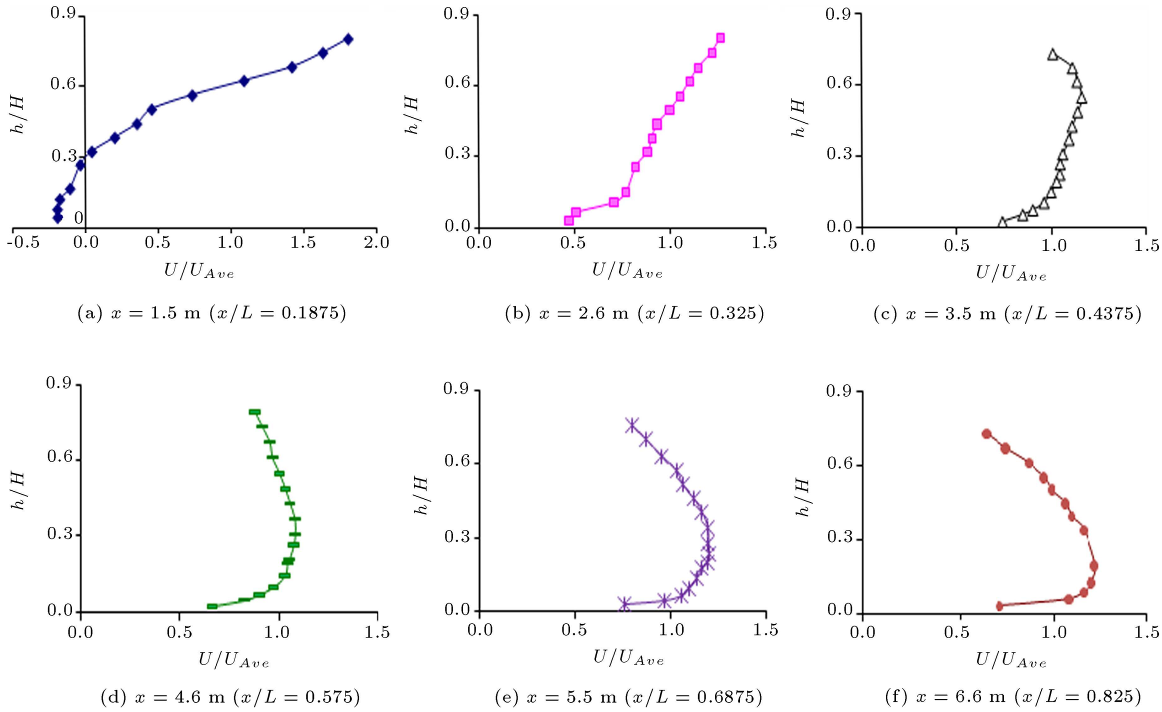
first section (immediately after the inlet, $x = 1.5$ m). Negative velocity values at the inlet region and near the bed indicate the existence of a vortex due to the shear layer between the inlet jet flow and the fluid inside the channel, which is produced as the result of a noticeable velocity gradient.

The difference in the vortex intensities and, consequently, vortex sizes at $x = 1.5$ m for different flow rates can be seen from Figures 8a and 9a. In the case of half flow rates, the inlet jet velocity is half that of the full flow rate and, accordingly, the intensity of the corresponding vortex due to the velocity gradient between the fluid layers is much smaller compared to that in a full flow rate. For the case of half flow rate, because the vortex size and its intensity are smaller, the velocity profiles achieve their developed shapes sooner, from $x = 2.6$ m (which may seem to be better for the purpose of sedimentation), whereas in the case of full flow rate this fact is observed from $x = 4.6$ m.

At a distance from the inlet, flow tends to be more developed. The first reason (which can be seen in Figures 8 and 9) is that although part of the velocity profiles at $x = 1.5$ m have negative values, the other velocity profiles in the five other measuring sections have positive values over the entire flow depth and, therefore, flow travels toward the outlet. Secondly, at a distance from the inlet, velocity profiles tend to become closer to their expected developed shape; this means that they tend to fit themselves to the boundary conditions of the channel, such as the effect of the bed's wall and free surface.

The results of time-averaged streamwise velocity profiles for channels with an inlet region baffle at x_b = 1.2 m (under full and half flow rates) are shown in Figures 10 and 11, respectively. As shown, part of the velocity profile at $x = 1.5$ m has negative values because this location ($x = 3.5, h_b = 1.5$ m) corresponds to the immediate downstream of the baffle. Specifically, the negative values in the case of full flow rate indicate a reverse flow region downstream of the baffle. This phenomena is similar to the flow downstream of a backward-facing step, which includes reverse flow region and reattachment point. In addition, the magnitude of velocities at the upper part of this section increased due to a reduction in cross sectional area and an increase in the kinetic energy of the fluid. After $x = 1.5$ m, similar to the case of no baffle, no negative velocity can be observed and velocity profiles tend to become more uniform at the downstream of the baffle. This uniformity may be more favorable towards having a tank with better flow conditions for sedimentation.

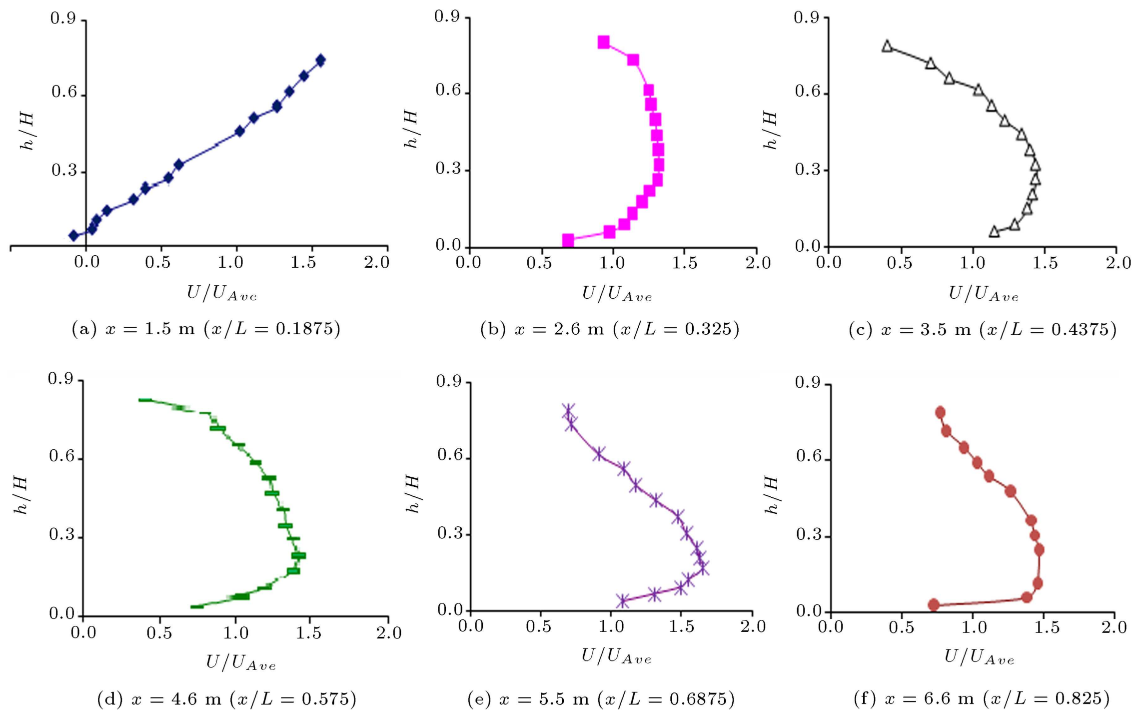
The velocity profiles for an intermediate baffle at x_{b2} = 4 m under full and half flow rates are shown in Figures 12 and 13, respectively. Again, after the inlet (x= 1.5 m), the velocity profile is partially



(a) $x = 1.5$ m ($x/L = 0.1875$) (b) $x = 2.6$ m ($x/L = 0.325$) (c) $x = 3.5$ m ($x/L = 0.4375$)

(d) $x = 4.6$ m ($x/L = 0.575$) (e) $x = 5.5$ m ($x/L = 0.6875$) (f) $x = 6.6$ m ($x/L = 0.825$)

Figure 8. Time-averaged streamwise velocity profiles with no baffle ($Q = 42$ l/min).

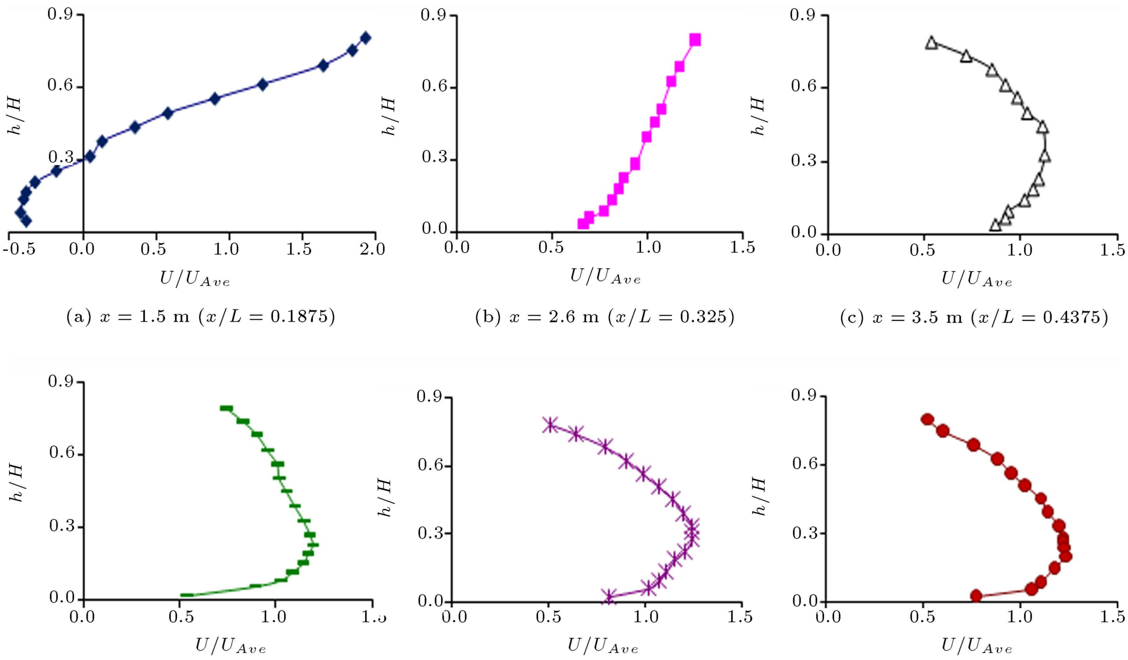


(a) $x = 1.5$ m ($x/L = 0.1875$) (b) $x = 2.6$ m ($x/L = 0.325$) (c) $x = 3.5$ m ($x/L = 0.4375$)

(d) $x = 4.6$ m ($x/L = 0.575$) (e) $x = 5.5$ m ($x/L = 0.6875$) (f) $x = 6.6$ m ($x/L = 0.825$)

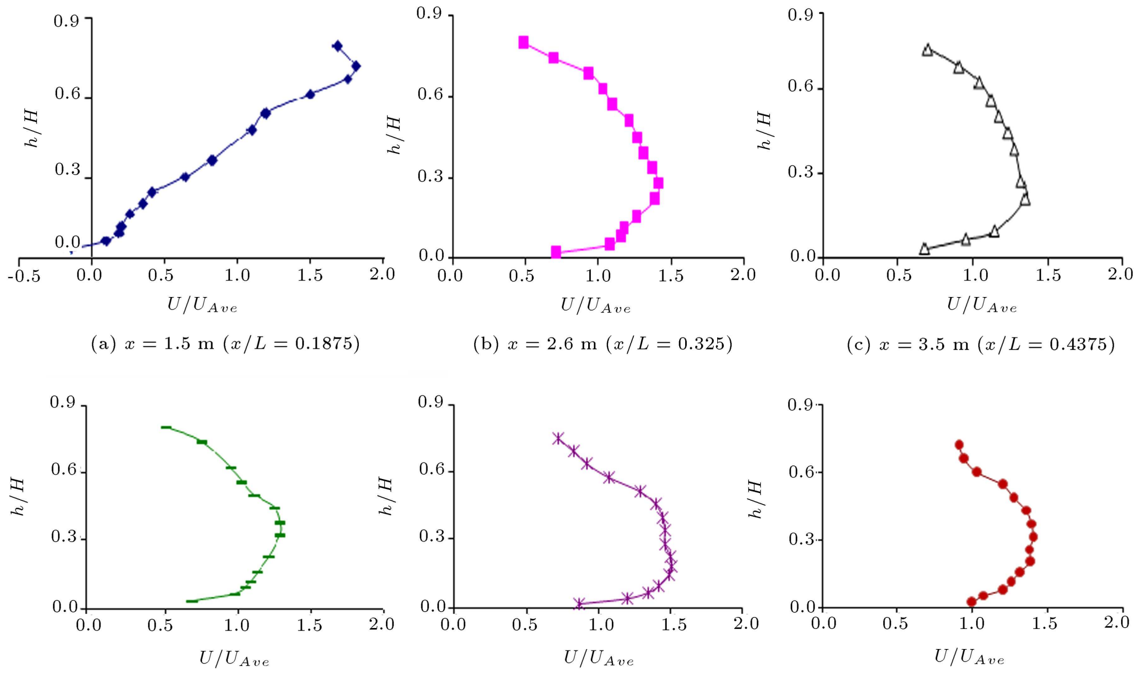
Figure 9. Time-averaged streamwise velocity profiles with no baffle ($Q = 21$ l/min).

negative for a full flow rate. The negative velocity at this section indicates the existence of a vortex near the bottom part of the channel due to the inlet jet flow resulting from a velocity gradient. This finding is similar to testing results with no baffle. In both figures, because of the presence of the intermediate baffle, a considerable increase in the velocity and kinetic energy is observed. Furthermore, downstream of the baffle, the velocity profiles tend to be much more uniform than the previous ones, since this section is located far downstream of the baffle (almost at a distance of $10h_{b2}$ from the baffle). With increasing



(a) $x = 1.5$ m ($x/L = 0.1875$)
(b) $x = 2.6$ m ($x/L = 0.325$)
(c) $x = 3.5$ m ($x/L = 0.4375$)
(d) $x = 4.6$ m ($x/L = 0.575$)
(e) $x = 5.5$ m ($x/L = 0.6875$)
(f) $x = 6.6$ m ($x/L = 0.825$)

Figure 10. Time-averaged streamwise velocity profiles with baffle at $x_{b1} = 1.2$ m ($Q = 42$ l/min).



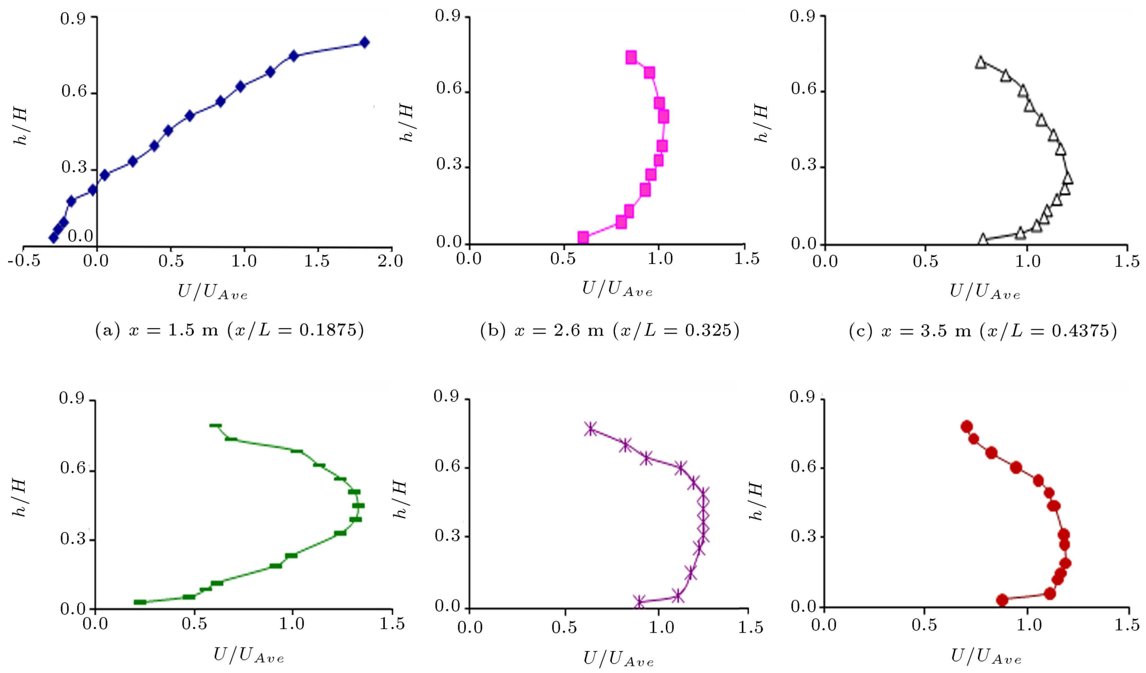
(a) $x = 1.5$ m ($x/L = 0.1875$)
(b) $x = 2.6$ m ($x/L = 0.325$)
(c) $x = 3.5$ m ($x/L = 0.4375$)
(d) $x = 4.6$ m ($x/L = 0.575$)
(e) $x = 5.5$ m ($x/L = 0.6875$)
(f) $x = 6.6$ m ($x/L = 0.825$)

Figure 11. Time-averaged streamwise velocity profiles with baffle at $x_{b1} = 1.2$ m ($Q = 21$ l/min).

distance from the baffle, water flows slower. As a result of the suction of the outlet weir, the shape of velocity profiles at the last two sections will not be so uniform.

It is noted that velocity profiles after $x = 2.6$ m tend to be more uniform for half flow rate, whereas the uniformity in velocity profiles for full flow rate occurs after $x = 3.5$ m. In general, results show that velocity profiles tend to develop faster in cases of half flow rate rather than those of full flow rate.

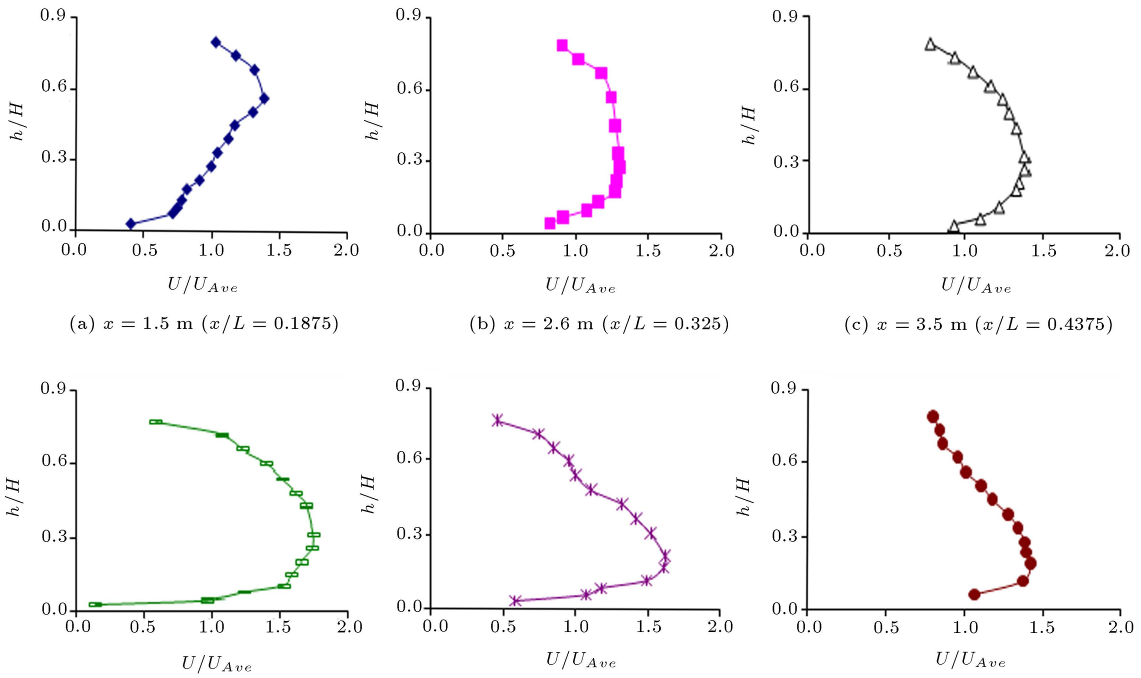
Figure 14 shows velocity profiles at $x = 1.5$ m for cases of full flow rate. The bottom part of all



(a) $x = 1.5$ m ($x/L = 0.1875$) (b) $x = 2.6$ m ($x/L = 0.325$) (c) $x = 3.5$ m ($x/L = 0.4375$)

(d) $x = 4.6$ m ($x/L = 0.575$) (e) $x = 5.5$ m ($x/L = 0.6875$) (f) $x = 6.6$ m ($x/L = 0.825$)

Figure 12. Time-averaged streamwise velocity profiles with baffle at $x_{b2} = 4$ m ($Q = 42$ l/min).



(a) $x = 1.5$ m ($x/L = 0.1875$) (b) $x = 2.6$ m ($x/L = 0.325$) (c) $x = 3.5$ m ($x/L = 0.4375$)

(d) $x = 4.6$ m ($x/L = 0.575$) (e) $x = 5.5$ m ($x/L = 0.6875$) (f) $x = 6.6$ m ($x/L = 0.825$)

Figure 13. Time-averaged streamwise velocity profiles with baffle at $x_{b2} = 4$ m ($Q = 21$ l/min).

these profiles shows negative velocities. As discussed before, for the case of no baffle and intermediate baffle, the negative values indicate the existence of a vortex recirculation region. But negative velocities for the case of an inlet region baffle ($x_b = 1.2$ m), are due to the existence of the baffle and hence the reverse flow at its downstream has bigger size and intensity (Figure 15). It prevents the velocity profiles from achieving their fully developed shapes very quickly. As a result, the inlet region baffle seems to have a considerable influence on the flow field at $x = 1.5$ m, which is located at $3.5h_b$ downstream of the baffle.

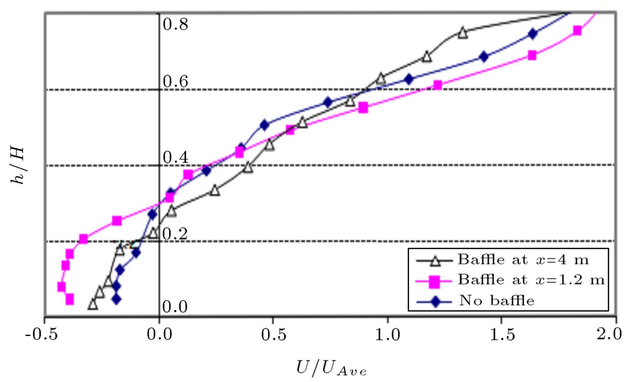


Figure 14. Time-averaged velocity profiles at $x = 1.5$ m ($Q = 42$ l/min).

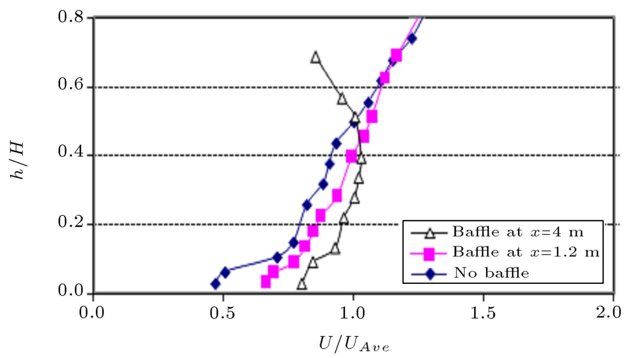


Figure 15. Time-averaged streamwise velocity profiles at $x = 2.6$ m ($Q = 42$ l/min).

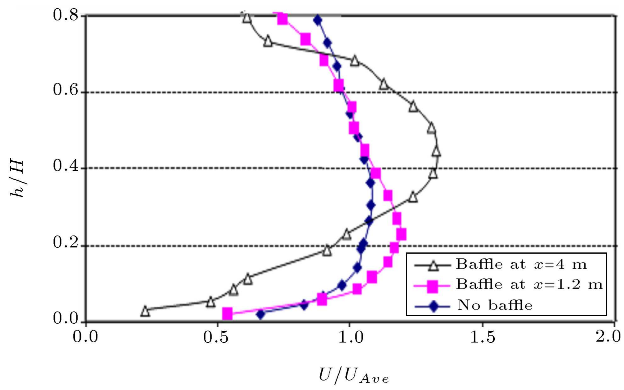


Figure 16. Time-averaged velocity profiles at $x = 4.6$ m ($Q = 42$ l/min).

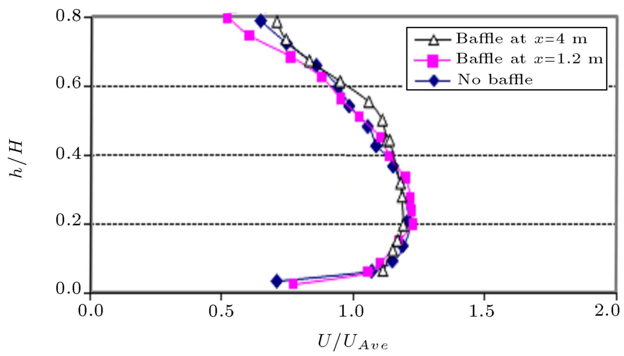


Figure 17. Time-averaged velocity profiles at $x = 6.6$ m ($Q = 42$ l/min).

In Figure 15, the velocity profile, in the presence of an intermediate baffle, has overtaken the velocity profile of a case of no-baffle in the bottom part of the channel. This increase in velocity can be attributed to a fast uprising flow at the intermediate baffle. Thus, an intermediate baffle not only influences flow field in its downstream but also affects the flow pattern in its upstream.

Therefore, in general, it is concluded quantitatively that baffle position and operating flow rate influence the flow field in various ways and attention should be paid to their proper design criteria.

In Figure 16, just behind the intermediate baffle, the velocity profile achieves its maximum in the upper part of the channel. In this section, velocity profiles that are related to the full flow rate, in cases of no baffle and inlet region baffle, achieve their maximum in the bottom part of the channel. Also, an obvious reduction of velocity gradient near the channel bed is observed behind intermediate baffle. Consequently, in the presence of this baffle, a decrease in bed friction occurs, which might be favorable (at this section) for sedimentation behind the baffle.

As shown in Figure 17, in the last section, the pattern of flow is almost the same in all cases. This section is located near the outlet weir and due to the suction of flow toward the outlet, velocity profiles have a similar pattern and the flow seems to be under the influence of the outlet.

Thus, flow pattern can be influenced by baffle setting and its position relative to inlet and outlet. Consequently, care should be taken in their proper settings.

SUMMARY AND CONCLUSIONS

The conclusions drawn from this experimental study are summarized below:

1. ADV has been successfully applied to measure the flow velocity in a rectangular primary sedimentation open channel. The results were used to assess the effect of a geometrical modification of the channel on the flow in the presence or absence of an intermediate baffle at different streamwise sections. Two different flow conditions and baffle positions were also studied.
2. At the inlet region and in the absence of a baffle or in the presence of an intermediate baffle, the negative mean streamwise velocities near the channel bed indicate the existence of a vortex that is of a bigger size and intensity at higher Reynolds

numbers. But immediately downstream of the so called inlet region baffle ($x_b = 1.2$ m), the existence of a reverse flow region is observed with different size and intensity, depending on the flow rate. It seems to influence the development of the flow and consequently, the flow structure in the channel.

3. On the whole, experimental results show that baffle can alter the hydrodynamics of a flow field in different ways. Analysis of the mean flow shows how a flow structure changes from upstream to downstream of the baffle at various positions. Immediately downstream of the baffle, a reverse flow region was found based on the negative values of mean streamwise velocity profiles. It seems that this recirculation zone is of importance when the Reynolds number is high enough. It was also found quantitatively that the baffle can change the distribution of velocity at its downstream by increasing flow velocity and kinetic energy. Baffle also influences the velocity gradient near the bed of the channel.
4. Baffle provision may have the possibility of providing better conditions for sedimentation by influencing velocity profiles. Positioning of a baffle in the middle of the channel ($x_b/L = 0.5$) may improve the flow field at its downstream by modifying the velocity gradient near the channel bed. Positioning of an inlet region baffle might be beneficial at low Reynolds numbers, but further investigations are necessary.

ACKNOWLEDGMENTS

The authors of this paper are grateful to all who provided useful assistance. We especially offer our gratitude to the Center of Excellence in Energy Conversion (CEEC) at the School of Mechanical Engineering, and to the Research Deputy of Sharif University of Technology who supported this research. We also express our thanks to the respected reviewers of the paper for their valuable comments.

REFERENCES

1. Swamee, P.K. and Tyagi, A. "Design of class-I sedimentation tanks", *J. Env. Engng.*, **122**(1), pp. 71-73 (1996).
2. Cripps, S.J. and Bergheim, A. "Solids management and removal for intensive land-based aquaculture production systems", *J. Aqua-Cultural Engng.*, pp. 33-56 (2000).
3. Camp, T.R. "Sedimentation and the design of settling tanks", *Trans. ASCE*, **111**(6), pp. 895-952 (1946).
4. de Clercq, B., Kinnear, D.J. and Vanrolleghem, P.A. "On-line dynamic fluid velocity profiling in secondary clarifiers", *Biomath*, Ghent University, Coupure Links 653, B-9000 Belgium (2003).
5. Campbell, B.K. and Empie, H.J. "Improving fluid flow in clarifiers using a highly porous media", *J. Env. Engng.*, **132**(10), pp. 1249-1254 (2006).
6. American Water Work Association, *Water Treatment Plant Design*, McGraw-Hill, New York (1990).
7. Krebs, P., Vischer, D. and Gujer, W. "Inlet-structure design for final clarifiers", *J. Env. Engng.*, **121**(8), pp. 558-564 (1995).
8. Tamayol, A., Firoozabadi, B. and Ashjari, M.A. "Hydrodynamics of secondary settling tanks and increasing their performance using baffles", *J. Env. Engng.*, **136**(1), pp. 32-39 (2010).
9. Anderson, N.E. "Design of settling tanks for activated sludge", *Sewage Works J.*, **17**(1), pp. 50-63 (1945).
10. Overcamp, T.J. "Type II settling data analysis", *J. Env. Engng.*, **132**(1), pp. 137-139 (2006).
11. Swamee, P.K. "Design of flocculating baffled channel", *J. Env. Engng.*, **122**(11), pp. 1046-1048 (1996).
12. Lyn, A. and Rodi, W. "Turbulence measurements in model settling tank", *J. Hydraulic Engng.*, **116**(1), pp. 3-21 (1990).
13. Bretscher, U., Krebs, P. and Hager, W.H. "Improvement of flow in final settling tanks", *J. Env. Engng.*, **118**(3), pp. 307-321 (1992).
14. Krebs, P. "Success and shortcomings of clarifier modeling", *J. of Water Sci. and Tech.*, **31**(2), pp. 181-191 (1995).
15. Taebi-Harandi, A. and Schroeder, E.D. "Analysis of structural features on performance of secondary clarifiers", *J. Env. Engng.*, **121**(12), pp. 911-919 (1995).
16. Ahmed, F.H., Kamel, A. and Abdeljavad, S. "Experimental determination of the optimal location and contraction of sedimentation tank baffles", *Water, Air and Soil Pollution*, **92**(3-4), pp. 251-271 (1996).
17. Tamayol, A., Firoozabadi, B. and Ahmadi, G. "Effects of inlet position and baffle configuration on the hydraulic performance of primary settling tanks", *J. Hydraulic Engng.*, **134**(7), pp. 1004-1009 (2008).
18. Tamayol, A. and Firoozabadi, B. "Effects of turbulent models and baffle position on hydrodynamics of settling tanks", *Scientia Iranica J.*, **13**(3), pp. 255-260 (2006).
19. Goring, D.G. and Nikora, V.I. "Despiking acoustic Doppler velocimeter data", *J. Hydraulic Engng.*, **128**(1), pp. 117-126 (2002).
20. Finelli, C.M., Hart, D.D. and Fonseca, D.M. "Evaluating the spatial resolution of an acoustic Doppler velocimeter and the consequences for measuring near-bed flows", *Limnol. Oceanogr.*, **44**(7), pp. 1793-1801 (1999).
21. Nortek, *ADV Operation Manual*, Nortek AS, Vollen, Norway (2000).

22. Wahl, T.L. "Discussion of despiking acoustic Doppler velocimeter data", *J. Hydraulic Engng.*, **129**(6), pp. 484-487 (2003).

23. Wahl, T.L. "Analyzing ADV data using WinADV", *Joint Conference on Water Resources Engineering and Water Resources Planning & Management*, July 30-August 2, Minneapolis, Minnesota (2000).

BIOGRAPHIES

Hamidreza Jamshidnia was admitted to Azad University and graduated in March 2005 with a BS in Mechanical Engineering with a focus on Fluid Mechanics and Heat Transfer. He was then accepted at Sharif University of Technology and awarded a MS in Mechanical Engineering (Energy Conversion) in September 2007. His thesis title was "Experimental investigation of flow pattern in a rectangular sedimentation open channel" using an ADV (Acoustic Doppler Velocimeter). Subsequently, he was awarded a Japanese Government Scholarship in 2007 and joined Hokkaido University to conduct his PhD studies in flow measurement. His research has been particularly focused on the study of multidimensional characteristics of complex spatio-temporal flow structure around a standing baffle by application of a UVP (Ultrasonic Velocity Profiler) and is presently in the final stages of completion.

Bahar Firoozabadi is associate professor in the school of mechanical engineering at Sharif University of Technology, Tehran. Her research interests are Fluid Mechanics in Density Currents, presently focusing on Biofluid Mechanics, and Porous Media. She received her PhD in mechanical engineering also at Sharif University. She teaches Fluid Mechanics and Gas Dynamics for undergraduates, and Viscous Flow, Advanced Fluid Mechanics, Continuum Mechanics and Biofluid Mechanics for graduate students.