Korean J. Chem. Eng., **29**(2), 201-208 (2012)
DOI: 10.1007/s11814-011-0163-9

Physical modifications to improve a channel's flow distribution

An-Kyu Lee*, No-Suk Park,†, Suhan Kim***, and Nam Kim******

*K-Water Institute, Korea Water Resources Corporation (K-Water), 462-1, Jeonmin-dong, Yusung-gu, Daejeon 305-730, Korea
**Water Research Center, K-Water Institute, Korea Water Resources Corporation (K-Water),
462-1, Jeonmin-dong, Yusung-gu, Daejeon 305-730, Korea
***Department of Civil Engineering, Pukyong National University, San 100, Yongdang-dong, Nam-gu, Busan 608-739, Korea
****Division of Electrical and Computer Engineering, Chungbuk National University, Cheongju 361-763, Korea
(*Received 19 July 2010 • accepted 21 June 2011*)

Abstract−A modified approach is suggested to achieve even flow distribution (*i.e.*, the equality of flow distribution) in an open channel in water treatment plant. The suggested approach includes installing a longitudinal baffle with orifices in the main flow direction. To evaluate the efficiency of suggested model, both computational fluid dynamics (CFD) model and pilot tests were carried out. The pilot tests, which were scaled down to 1/8 of full scale, showed that the equalities of the flow distributions in the modified channels were up to 30% higher compared to those in the conventional channel when the equalities were quantified using the standard deviation of the flow distributions. In addition, the velocity contours computed by numerical simulation in the conventional full scale channel were compared with the velocity contours in the remodeled channel in order to verify the more even flow distribution in the modified one than that in the conventional one.

Key words: Equality of Flow Distribution, Distribution Channel, CFD (Computational Fluid Dynamics), Pilot Test

INTRODUCTION

In water treatment plants the incoming flow must be evenly distributed to the process units, especially for a series of successive lateral basins. Distribution channels are commonly used to evenly distribute the flow from the rapid mixing basins into the flocculation basins. If the flow to each flocculation basins is not evenly distributed, the retention time will be considerably different from a designed value in the flocculation basin and other locations that are constructed as a package. For example in a basin with lower flow rate than the designed value, the lower velocity and longer retention time will make particle deposition accelerated in the sedimentation process. On the other hand, the higher horizontal velocity and shorter retention time in a basin with higher flow rate than the designed value could yield re-floating sludge [1]. Since the higher flow rate and velocity increase the surface loading rate, which is inadequate for particles to settle down within sedimentation basin.

To obtain an even flow distribution (i.e., the equality of flow distribution) in open channels, it is very important to determine the design factors, such as the inlet structure, sectional geometry, length, outlet size, and shape. However, since it is too difficult to understand the factual hydrodynamic behavior in channels accurately, it has been designed with trial-and-error methods that were based on experiments. Due to the complexity of most hydraulic constructions, it is impossible to establish a generalized design method. Particularly in the case of the distribution channel, fluid behaviors are so complicated that three-dimensional analysis should be performed in order to predict the actual flow phenomena. If this kind of hydrodynamic problem is analyzed with a one-dimensional time function or two-dimensional code based on shallow water theory, a serious error may occur [2]. It is because the theory lacks proper assumptions which help to simulate the actual flow behaviors in the distribution channels.

Fig. 1 shows the top view of a typical open type distribution channel with a constant width. The most important factor for designing an open type distribution channel is the Froude number, which represents the ratio of inertia force to gravity force, or the ratio of the average velocity to surface wave transfer velocity [3].

The 'step method' proposed by Chao and Trussel in 1980 has been widely used to design an open type distribution channel [3]. In this method, the flow distribution to each basin (flocculation or sedimentation basin) is determined by proceeding step by step from the downstream end of the channel to the upstream end where the flow enters. A number of modification approaches have been also proposed to correct the uneven flow distribution in the basic open type channel, including changing the weir elevation, tapering the channel to keep the Froude number constant, and slight adjustment of the elevation of each weir coupled with the tapered channel as the modification approaches [4]. However, it was proved that an even flow distribution could not be achieved in most channels which were designed by the 'step method' [1]. This uneven distribution occurred mainly from the abrupt turn of the flow direction due to inadequate inlet geometry. Ultimately, this phenomenon has significantly impaired the treatment efficiencies [2,5].

Therefore, the objectives of this study are to recognize and quantify the problems of the conventional distribution channel that are associated with the uneven flow distribution and to suggest a modified measure to improve the equality. To test the feasibility of the suggested measure, the computational fluid dynamics (CFD) tech-

†To whom correspondence should be addressed.
E-mail: nspark@kwater.or.kr

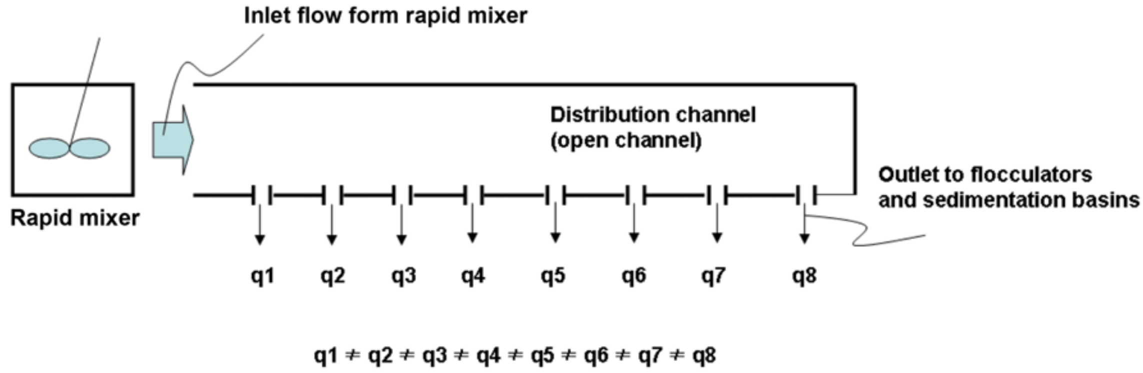


Fig. 1. The top view of an open type distribution channel between rapid mixer and flocculation basins.

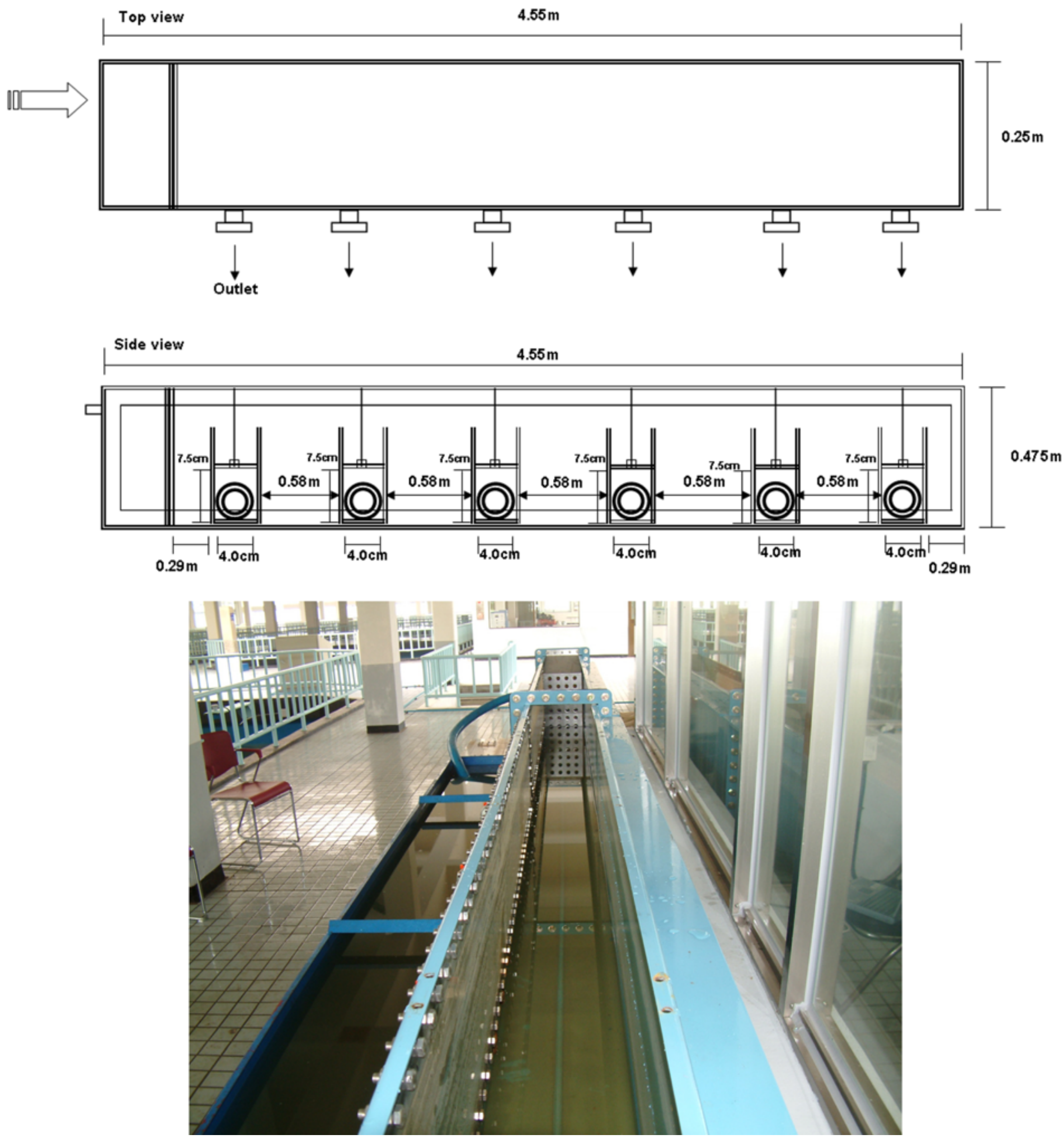


Fig. 2. The pilot scale channel to represent the conventional distribution channel.

nique was used, and to verify the CFD simulation results, pilot tests were carried out for the pilot scale channel, which was scaled down to 1/8 of a full-scale distribution channel being operated in the Kosan Water Treatment Plant, Republic of Korea. Furthermore, to establish the dynamic similarity between the full-scale and pilot scale, the Froude numbers were set to be identical.

EXPERIMENTAL METHODS

Pilot tests were carried out using two pilot scale channels (refer to Fig. 2 and 3) according to six different experimental conditions, as shown in Table 1 before being compared to the CFD simulation results.

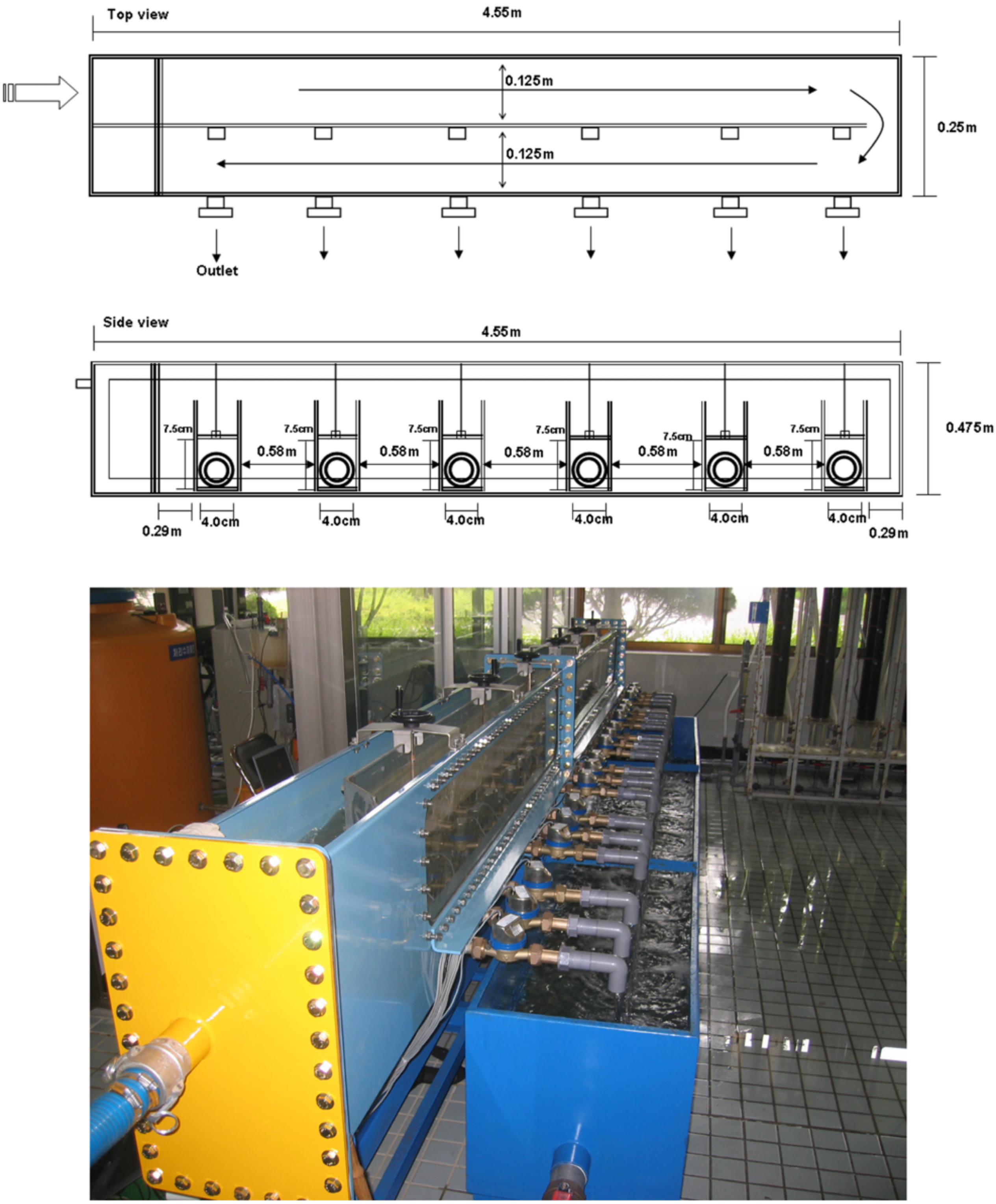


Fig. 3. The pilot scale channel with the modified baffle.

Table 1. Pilot test conditions matrix

Hydraulic characteristics		Average velocity (V, m/sec)	Froude No. (dimensionless)	Water level at inlet to channel (m)	Note
The conventional channel (without longitudinal baffle)	Test 1	0.0359	0.376	0.36	- The Froude number from the full-scale channel is 0.031 - Total inlet flow rate (Q) for pilot channels is 0.4 L/sec
	Test 2	0.0040	0.042	0.36	
The modified channel (setting longitudinal baffle with six orifices)	Test 3 (Opening ratio 0%)	0.008	0.010	0.4	
	Test 4 (Opening ratio 26.7%)	0.008	0.010	0.4	
	Test 5 (Opening ratio 66.7%)	0.008	0.010	0.4	
	Test 6 (Opening ratio 100%)	0.008	0.010	0.4	

* The opening ratio means the ratio of open area to total area with a gate

Fig. 3 shows the modified hydraulic structure suggested in this study to solve problems related to the uneven distribution within the conventional channels. The low average velocity within the channels may let micro-floc settle down before entering the flocculation basins and grow. The phrase "low average velocity" used in the previous sentence means the velocity which makes the Froude number less than 0.1 because the numerator term for expressing the Froude number is the average velocity and the denominator is the surface wave celerity. As can be seen in Fig. 3, installing a longitudinal baffle with orifices within the conventional channel would cause the inlet water to turn at the end of the right side. As the inlet water flow runs from the left side to the right side, the water level at the left side would be higher than that at the right side due to wall friction. After the water turns at the end of the right side, as it flows from the right side to the left side, the water level gradient is reversed. Since water passes through the orifices on the longitudinal baffle, the water level above each outlet orifice can be kept constant. Although both of the channels have the same theoretical retention time, the average velocity within the modified channel can be twice as high as that within the conventional channel because the cross-sectional area decreases to half. It is expected that this increased average velocity can diminish the micro-floc settling within the channel.

In addition, the results of pilot tests would be used to investigate the effect of various hydraulic characteristics and operating conditions on the equality of flow distribution. Under each condition, the steady state flow rates from six outlet orifices were obtained separately using a stop watch and cylinder.

Two pilot scale channels (both conventional and modified channels) were scaled down to 1/8th of the full-scale distribution channels. Figs. 4 and 5 show the geometries of the full scale channel.

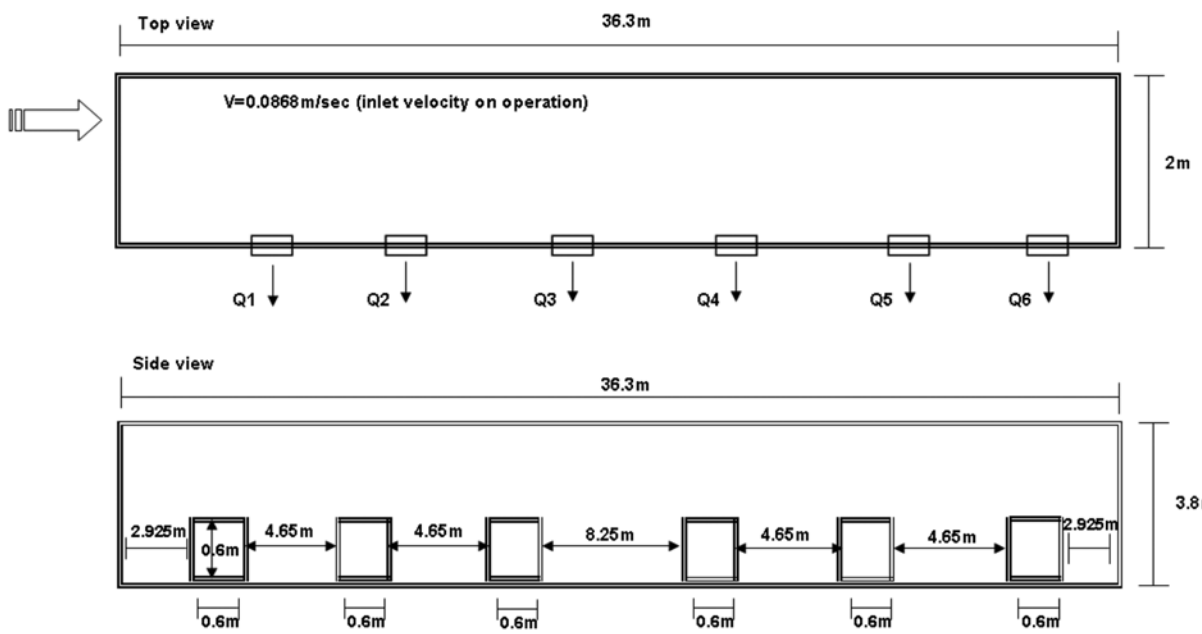


Fig. 4. Full scale channel without modification baffle (inlet velocity: 0.087 m/sec).

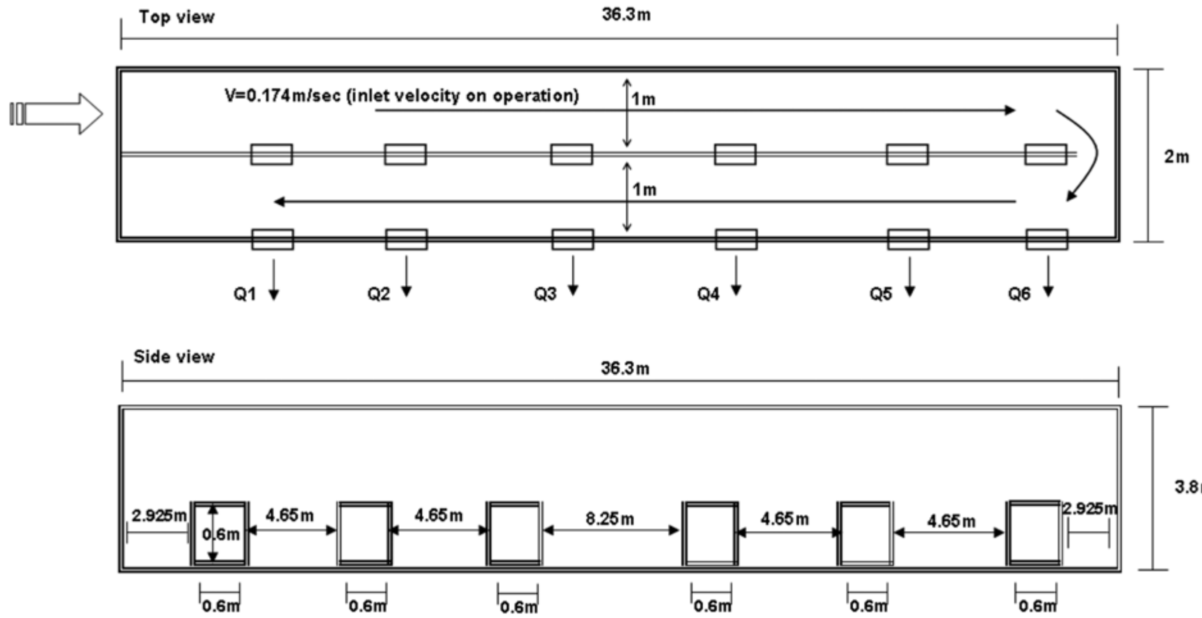


Fig. 5. The full scale channel with modification baffle (inlet velocity: 0.174 m/sec).

To achieve the dynamic similarity to the operation condition of the full scale channel, the Froude number, as a scale parameter, was selected to be identical for both the pilot and the full scale channels (refer to Table 1). Even though the Froude numbers vary from 0.01 to 0.049, it can be thought that those are similar enough to achieve dynamic similarity from the viewpoint of orders of magnitude. Generally, in the case that Froude number is selected as a scale parameter, it can be acceptable to make orders of magnitude the same for two channels with different scales. The opening ratio of orifices on the centrally installed longitudinal baffle could be changed by controlling each gate. The opening ratio means the ratio of open area to total area with a gate. The size of each orifice was designed to be 30 cm^2 (width 4 cm×height 7.5 cm).

Except for test 1 condition (Reynolds number is 2540), flow in the open channels could be considered as laminar flow since Reynolds number less than 2100 represents laminar flow condition. In the case of test 2 in Table 1, the Froude number was calculated as 0.042, which is the closer to the number (0.031) from the full-scale channel compared to the other cases. On the other hand, test 1, where the Froude number is relatively high, was conducted to investigate the effect of the average velocity and Froude number on the flow distribution within an open channel. The ratios of the open area (i.e., opening ratio in Table 1) for six orifices on the longitudinal baffle were uniformly set to be 0% (closed), 26.7%, 66.7%, and 100% in the conditions of tests 3, 4, 5, and 6, respectively. Tests 1 and 2 represent the conventional full-scale channel, and test 3, 4, 5, and 6 conditions represent the modified full-scale channel. Since the hydraulic radius increases in tests 3 to 6, in which the longitudinal baffle with orifices were installed within the channel, the Froude number tends to decrease.

CFD SIMULATION METHODS

In this study, fluid behavior in both pilot (Fig. 2 and 3) and full-scale (Fig. 4 and 5) distribution channels was simulated by the computational fluid dynamics (CFD) technique to evaluate the equality in each channel. Figs. 4 and 5 depict both geometries of the conventional full-scale distribution channel being operated in the Kosan Water Treatment Plant, Republic of Korea and the modified distribution channel for improving the equality of the flow distribution based on this concept. The Froude numbers calculated from the conventional and the modified measure approaches are 0.031 and 0.049 for Q=2.375 m^3/hr, respectively.

The velocity and water head distributions within each basin, which are associated with the equality, were simulated using 3-D commercial the CFD code, CFX 10.0 developed by ANSYS [6]. For the simulations, 880,000 cells (tetrahedron shape) were generated for both the geometries of the conventional and the modified channels. The time-averaged Navier-Stokes equations for momentum and continuity were solved for steady, incompressible, turbulent, and isothermal flow.

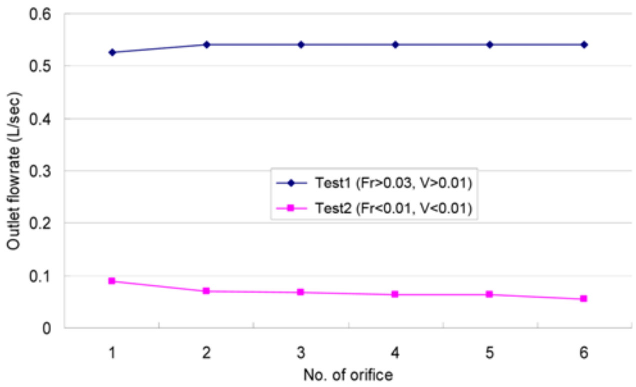
The equations below represent the continuity (Eq. (1)) and momentum (Eq. (2)), respectively:

$$\underline{\nabla}\cdot(\underline{U})=0 \tag{1}$$

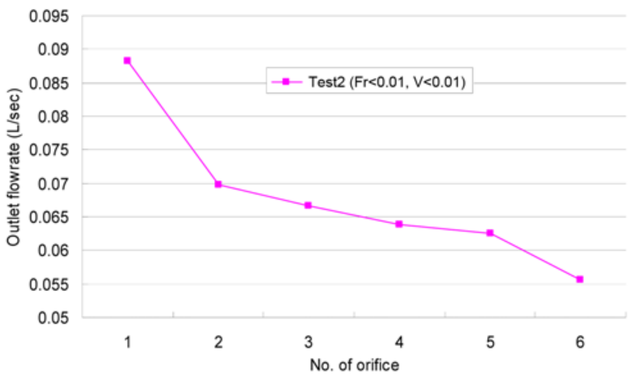
$$\underline{\nabla}\cdot(\rho\underline{U}\otimes\underline{U}-\mu\underline{\nabla}\underline{U})=\underline{B}+\underline{\nabla}P-\underline{\nabla}\cdot(\overline{\rho\underline{u}\otimes\underline{u}}), \tag{2}$$

where ρ and μ are the fluid density and dynamic viscosity, respectively; P the pressure; $\underline{U}$ the fluid mean velocity; $\underline{B}$ a body force; and u the fluctuating velocity.

Mean flow and turbulence characteristics of open-channel flows are studied either by experimental measurements or numerical computations [7]. In many engineering problems in fluid mechanics and hydraulics, the k-ε model has been most widely employed due to well-established empirical coefficients of the model [7]. All cases were simulated by using a turbulence modeling method in order to investigate the eddy flow and energy dissipation in detail. Therefore, a standard k-ε model was selected for modeling the turbulence



(a) The results of test 1 and 2



(b) The result of test 2

Fig. 6. The results of pilot tests 1 and 2.

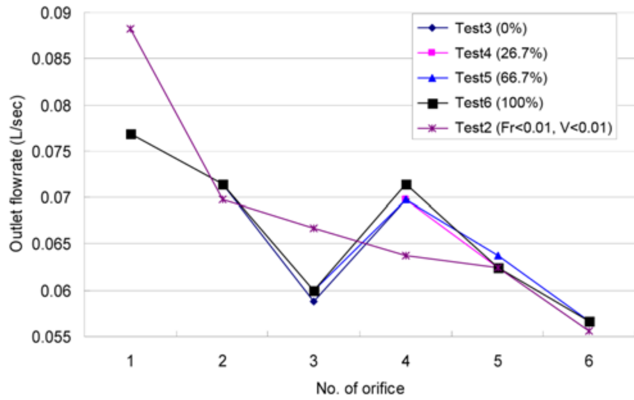


Fig. 7. The results of pilot tests 2, 3, 4, 5 and 6.

transport of momentum. The empirical coefficients and weighting constants suggested by Lopez and Garcia were used in the CFD simulation [8]. In addition, no-slip condition was assumed on the surfaces of sides, bottom, and longitudinal baffle, and the well-known standard wall function was applied to bridge the viscous sub-layer. To model the free surface, which is contact face between air and water, VOF (Volume of Fraction) method was utilized [9].

RESULTS AND DISCUSSION

1. Pilot Test Results

Fig. 6 shows the results of pilot tests conducted under the test 1 and test 2 conditions for the pilot scale channel. Fig. 6(a) shows the comparison between two conditions; one is the case where the Froude number is higher (0.376) than 0.1 and the other is the case where the Froude number is lower (0.042) than 0.1. Fig. 6(b) shows the results of test 2 only, whose condition is relatively more similar to the current operating condition for the full-scale channel than test 1 (refer to the fourth and the last column in Table 1). As can be seen in Fig. 6(b) compared to Fig. 6(a), the scale on the Y-axis decreases to emphasize maldistribution along with the channel length.

From the comparison between tests 1 and 2 in the case of relatively higher Froude number, the further the orifice is located from the inlet, the more outlet flow-rate through orifice is. On the other hand, in the case of relatively lower Froude number, the further the orifice is located from the inlet, the less outlet flow-rate through orifice is. Froude numbers from the full-scale and the pilot channel (test 2) are 0.031 and 0.042, respectively. Considering Froude number as a dynamic similarity parameter, it could be assumed that hydraulic characteristics of each flow within two channels are similar. From the results of test 2, the standard deviations of six outlet flow rates were calculated at 0.011 L/sec.

Fig. 7 shows the flow distribution dependent on the orifice opening ratio from the pilot tests conducted under test 2, 3, 4, 5 and 6 conditions. Among them, only test 2 represents the channel without baffle, and the others do the modified channel with baffle. It can be observed from Fig. 7 that the flow distribution patterns on test 3, 4, 5 and 6 are more equitable than that on test 2. To quantify the equality of flow distribution in detail, the standard deviation of six outlet flow rates for each condition was calculated as shown in Table 2. The smaller standard deviation means the more evenly distributed flow rates. The equalities of the flow distributions in the case of the modified channels (tests 3, 4, 5 and 6) were up to 30% higher (i.e., standard deviations for the cases of the modified channels were up to 30% smaller) than those in the conventional channels. In more detail, the highest and smallest standard deviations for the conventional and modified channel are 0.011 and 0.0076, respectively. In addition, pilot test results revealed that the opening ratios of orifices had little influence on improvement of equality as shown in Table 2.

2. CFD Simulation Results

Figs. 8 and 9 show the simulation results for the conventional full and pilot scale distribution channel and the modified channels, respectively.

As can be seen in Fig. 8, the longer the distance between the orifice and inlet point is, the lower the velocity of the flow passing through the orifice becomes. On the other hand, in the case of modified channel, a relatively uniform distribution of the velocity could be observed within overall channel (refer to Fig. 9). The portion of the

Table 2. Standard deviation from the results of pilot tests

Test condition	Test 1	Test 2	Test 3	Test 4	Test 5	Test 6
Standard deviation of six outlet flow rates (L/sec)	0.0092	0.011	0.0079	0.0077	0.0076	0.0079

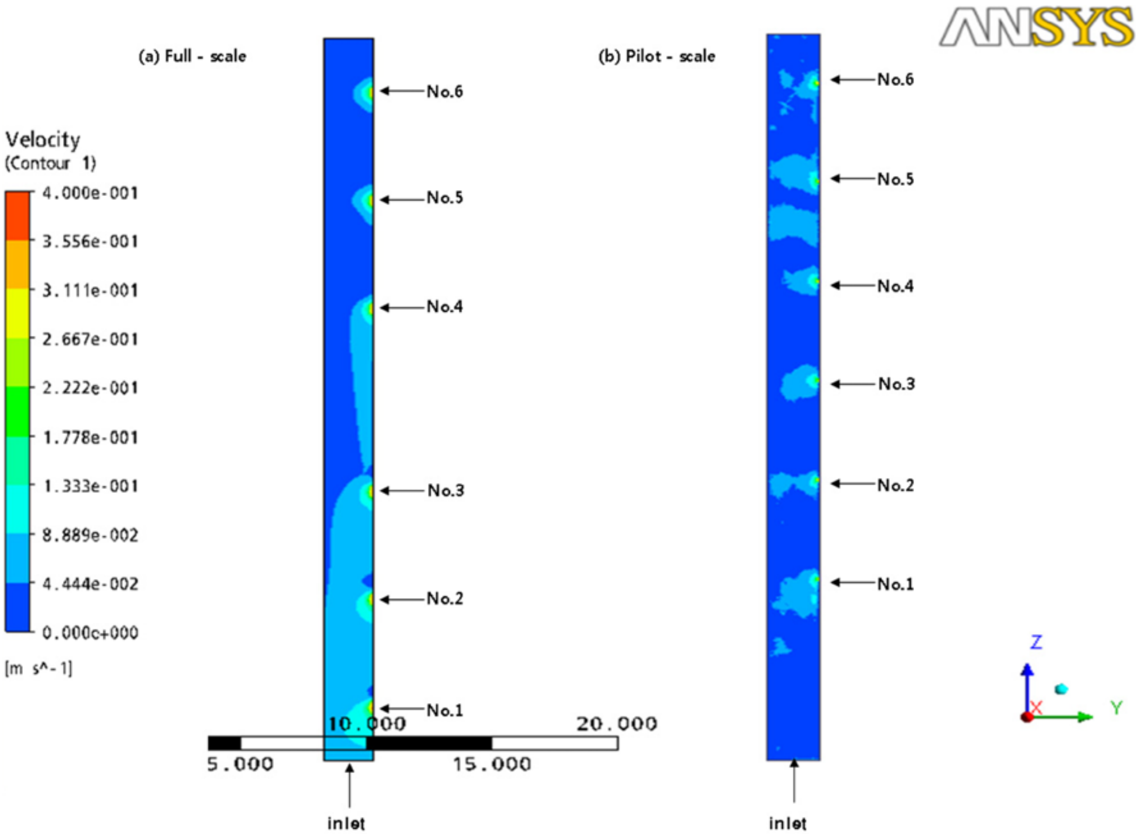


Fig. 8. CFD simulation results for the conventional distribution channels.

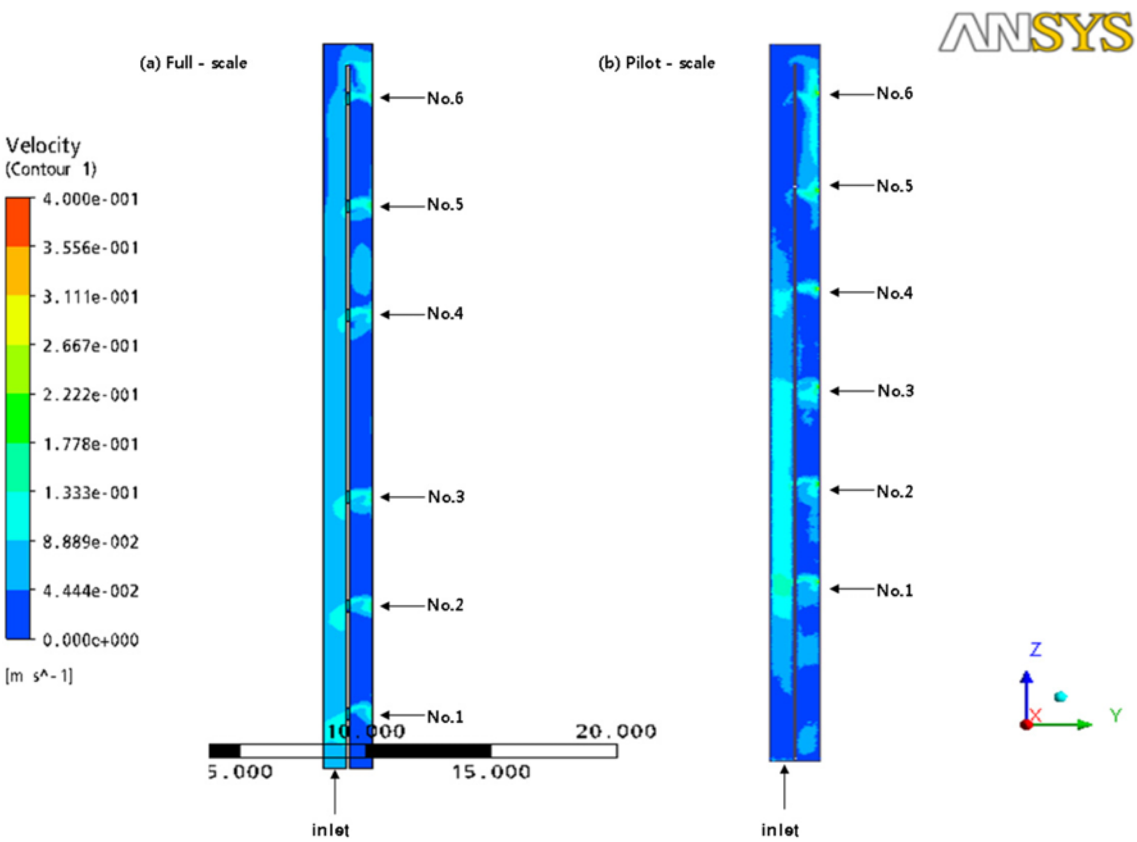


Fig. 9. CFD simulation results for the modified channels.

dark blue area in the key, representing lower velocity than that represented by sky-blue color, is concentrated on down stream. In the case of modified channel, sky blue color area is distributed more uniformly and widely.

From the results of pilot tests 3, 4, 5 and 6, the outlet flow rates through orifice No. 3 and 6 were relatively less than those through other orifices. As can be seen in Fig. 8(b) pilot scale (right hand side), the sky blue area in near No. 3 and 6 orifice is relatively smaller than other orifices. Although the reason why this phenomenon occurred has not been clear, these experimental results are observed from the results of CFD simulation where the local pressure near No. 3 and 6 orifices is relatively lower than those near to other orifices.

CONCLUSIONS

This study was conducted to quantify the problems of the conventional distribution channel, which are associated with uneven flow distribution, and to suggest a modified measure to improve the equality of flow distribution. To test the feasibility of the suggested measure, the computational fluid dynamics (CFD) technique was used and pilot tests were carried out for the pilot scale channel, which was scaled down to 1/8th of a full-scale distribution channel being operated in a water treatment plant in the Republic of Korea to verify the CFD simulation results. The key findings are as follows.

1) From the results of CFD simulation and pilot tests, the modification of the hydraulic structure in the distribution channel, which is to install the longitudinal baffle with orifices in the main flow direction, can increase the equality of flow distribution. It was quantified by calculating the standard deviation of the flow rates of all the outlets through each orifice. The equalities of flow distributions within the modified channels were up to 30% higher than those within the conventional channels.

2) In the case of a relatively low Froude number (<0.1) within the open distribution channel, the further the orifice is located from the inlet, the less the outlet flow-rate. On the other hand, if the Froude number is relatively higher (>0.1), since the dissipated energy due to the loss of flow mass and the friction with the walls may be less than the accumulated energy from the successive supplement, the trend of flow distribution is opposite to the former. The criterion of these controversial phenomena is the Froude number.

3) From the results of pilot tests, the opening ratio of the orifices on the longitudinal baffle did not influence the equality of flow distribution if the average flow velocity and Froude number were relatively low (Froude number $\approx$ 0.01).

REFERENCES

1. H. K. Baek, N. S. Park, J. H. Kim, S. J. Lee and H. S. Shin, *Journal of Water Supply: Research & Technology - AQUA-*, **54**(6), 349 (2005).
2. H. K. Baek and W. K. Kim, *Rapid mixing and Flocculation basins, Composite Correction Program for Seoung-nam Water Treatment Plant*, Korea Water Resources Corporation (2000).
3. J. L. Chao and R. R. Trussel, *J. Environ. Eng.*, *ASCE*, **106**, 321 (1980).
4. D. B. Larry, F. J. Joseph and P. A. David, *Flow in Open Channels in Treatment plant of Environmental Engineers*, Prentice-Hall (1984).
5. E. H. Jr. Herbert, *Water Clarification Process Practical Design and Evaluation*, Van Nostrand (1981).
6. ANSYS, *CFX Application manual*, United Kingdom: Oxfordshire (2006).
7. S.-U. Choi and H. Kang, *Journal of Hydraulic Research*, **42**(1), 3 (2004).
8. F. Lopez and M. Garcia, *Water Resources Research AGU*, **34**(9), 2341 (1998).
9. Athanassios A. Dimas, Nikolaos TH, Fourtiotis, Adreas P. Vouros and Alexander C. Demetracopoulos, *Journal of Hydraulic Research*, **46**(6), 802 (2008).