Examination of three-dimensional flow characteristics in the distribution channel to the flocculation basin using computational fluid dynamics simulation

Heung-Ki Baek, No-Suk Park, Jeong-Hyun Kim, Sun-Ju Lee and Hang-Sik Shin

ABSTRACT

This study was conducted to evaluate the equity of the flow distribution from the rapid mixing basins to the flocculation basins. Several types of inlet structure of the open channel affecting the flow pattern and distribution trend were simulated using the computational fluid dynamics (CFD) technique. In order to investigate the factual phenomena in the distribution channel, we selected a certain domestic water treatment plant with capacity of 361,000 $m^3 d^{-1}$. From the results of CFD simulation and measurements of flow discharge, it was investigated that the existing inlet geometry resulted in significant inequitable distribution. The largest deviations of flowrate in the basins and rows were both over 10%. In order to reduce these deviations, this study suggested installing a baffle against the influent, and the largest deviation was reduced to less than 3%. It was concluded that the existing design method of the open channel could be improved by optimizing the even flow with three-dimensional hydrodynamic analysis.

Key words | CFD (computational fluid dynamics), distribution channel, equity of flow, standard deviation

Heung-Ki Baek (corresponding author)
Southern Geum River Water Supply Headquarters,
Korea Water Resources Corporation,
Sungjae-ri 27, Gosan-myeon, Wanju-gun,
Jeollabuk-do 565-863,
Republic of Korea
Tel: +82-63-260-4300
Fax: +82-63-260-4300
E-mail: *pinetree@kowaco.or.kr*

No-Suk Park
Jeong-Hyun Kim
Sun-Ju Lee
International Water & Wastewater Research Center, Korea Institute of Water & Environment,
Korea Water Resources Corporation,
462-1, Jeonmin-Dong, Yusung-Gu,
Daejeon 305-730,
Republic of Korea

Hang-Sik Shin
Department of Environmental & Civil Engineering,
Korea Advanced Institute of Science and Technology (KAIST),
373-1, Kusong-dong, Yusong-Gu,
Daejon 305-701,
Republic of Korea

INTRODUCTION

In water treatment plants, it is essential that the incoming flow be evenly distributed to the process units, especially for the series of successive lateral basins. Distribution channels are commonly used to distribute the flow from the rapid mixing basins into the flocculation basins. If the flow to each flocculation basin is not even, the retention time is considerably different in the flocculation basin and in sedimentation basins which are constructed as a package. In the basin with the lower flow rate, lower velocity and longer retention could occur and sludge deposition could be accelerated in the sedimentation process. On the other hand in the basin with the higher flow rate, greater horizontal velocity and shorter retention time could yield breaking floc or re-floating sludge.

To obtain equitable flow in distribution channels, the design factors at the channels are very important such as inlet structure, sectional geometry, length, outlet size and shape. However, since the actual hydrodynamic behaviour in channels is too complex to understand accurately, it has been designed based on experiments and trial-and-error methods. Due to the complexity of most hydraulic constructions, it is impossible to establish a generalized design method. Especially in the case of the distribution channel, fluid behaviour is so complicated that three-dimensional analysis should be performed in order to predict the actual flow phenomena. If this problem is analysed with a one-dimensional time function or two-dimensional code based on shallow water theory, serious error can occur (Baek & Kim 2000). In addition, the differences between the assumptions of the theoretical model and the actual conditions of design can yield seriously inequitable distribution in the channels.

Until now, except for the 'step method', there are few researches or methods developed to achieve equitable flow distribution. The '(correction) step method' proposed by Chao and Trussel in 1980 has been widely used as a design method of distribution channels (Chao & Trussel 1980). In this method, flow distribution to each 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 modifications were also proposed to correct the inequitable flow distribution in the basic channel design, such as changing the weir elevation, tapering the channel to keep Froude number constant and slight adjustments in the elevation of each weir coupled with the tapered channel (Benefield *et al.* 1984).

However, it has been proved that an equitable flow distribution could not be achieved in most channels that were designed by the 'step method'. This misdistribution occurred mainly from the abrupt turn of the flow direction at inadequate inlet geometry. Ultimately this phenomenon impaired the treatment efficiencies seriously (Hudson 1981; Baek & Kim 2000).

Therefore, recognition of the significance of equitable flow distribution in channels is needed to thoroughly investigate the hydrodynamic behaviour in a full-scale channel, using computational fluid dynamics (CFD) simulations and acoustic Doppler velocimetry (ADV) techniques. This study suggested installing a baffle against the influent and showed its effectiveness in reducing the deviation to less than 3%.

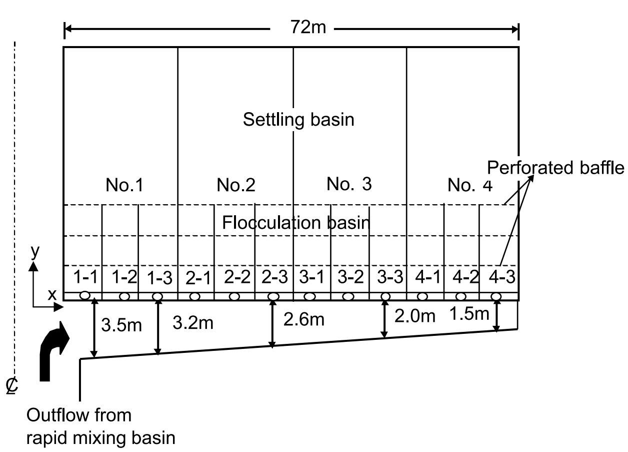

Figure 1 | Layout of distribution channel and flocculation basin.

MATERIAL AND METHODS

For the investigation of the actual phenomena in the distribution channel, we selected a certain domestic water treatment plant (WTP), S_WTP, with capacity of 361,000 $m^3 d^{-1}$. Figure 1 shows the plan view of the distribution channel and flocculation basin at S_WTP. The outflow from the rapid mixing basin enters the centre of the distribution channel through four pipes, each with diameter 1,100 mm horizontally. The distribution channel of 144 m in length is tapered from the centre towards the downstream end. Flow is distributed to eight flocculation basins through submerged valves. Each basin is divided into three rows. As shown in Figure 1, it is a symmetrical structure; therefore only half of the distribution channel (L = 72 m) and flocculation basins (no. 1 ~ 4) on the right-hand side were investigated.

In order to verify the result of CFD simulation and measure the actual flow discharge and velocity, acoustic Doppler velocimetry (ADV) using supersonic wave theory was used. The total flow discharge in each basin was 7,900 $m^3 h^{-1}$. The sampling points were located at the inlet of each basin (refer to Figure 1). Inlet flow rate in each basin was calculated from perforation ratio at each baffle and velocity measured. ADV used in this research measured the x, y and z components of velocity every 5 min, and generated time-average values. Table 1 summarizes features of acoustic Doppler velocimetry used in this study.

Methodology of CFD simulation

Governing equations

The CFD simulation works by splitting the geometry of interest into a large number of elements, collectively known as 'grids' or 'cells'. Then, momentum and continuity equations were formulated for each grid together with given boundary conditions, and then repeatedly solved by using the finite volume method (FVM) (Sicilian *et al.* 1987; Park *et al.* 2003). In this simulation, we generated 315,000 cells for geometry of the distribution channel and flocculation basin. For simulating the hydrodynamic behaviour in the distribution channel, Flow-3D, commercial code was used. The Flow-

3D program used in this simulation contains the continuity equation, momentum equation (Navier-Stokes), energy equation and volume of fluid (VOF) method suggested by Hirt and Nichols (1981) for free surface dynamics. The time-averaged Navier-Stokes equations for momentum and continuity were solved in this study for steady, incompressible, turbulent and isothermal flow.

The continuity and momentum equations are, as follows, respectively:

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

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

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 $\underline{u}$ the fluctuating velocity.

Turbulence modelling

Since the Reynolds number in the distribution channel is high enough to guarantee turbulent fluid conditions, all cases were simulated by the turbulent model. Reynolds number is a ratio of inertia force to viscous force. The range of Reynolds numbers was estimated to be 854,000 ~ 8,540,000 in the distribution channel. We could assume that the turbulence in the channel is isotropic. Therefore, a RNG k-ε model was used for modelling the turbulence transport of momentum (Suhas 1980; Perry 1984). The transport equations were solved to obtain the turbulent stresses and the turbulence energy dissipation rate.

Table 1 | Features of acoustic Doppler velocimetry

Acoustic frequency	10 MHz
Velocity range	0.002–0.1 m s^{-1}
Velocity resolution (accuracy)	0.1 mm s^{-1}
Velocity bias	± 0.5%
Operating temperature	0–40°C
Maximum depth	Up to 60 m
Response time	1 s

Boundary conditions

The liquid free surface at the top is considered flat and frictionless (i.e. a symmetry plane) even if there are some fluctuations at the free surface due to the turbulent flow. However, the fluctuations are usually small enough to be neglected for the purposes of this study. At the side and bottom wall surface, no-slip condition was assumed, and the well-known standard wall function method was used to bridge the viscous sublayer. Therefore, it is assumed that each component's velocity at each wall is zero. The wall shear stress was obtained from the logarithmic law of the wall (Currie 1993; Versteeg & Malalasekera 1995).

RESULTS AND DISCUSSION

Results of CFD simulation

Figures 2 and 3 show three-dimensional simulation results and the characteristics of flow distribution to each basin and row, respectively.

The discharge into basin no.1 near the inlet of the distribution channel was 57,500 m^3 d^{-1}, which corresponded to 31.9% of total discharge as shown in Figure 3. On the other hand, the smallest discharge was at basin no. 2: 21.8% (39,300 m^3 d^{-1}). In the case of basin nos. 3 and 4, the distribution rate was 22.7% and 23.6%, respectively. Therefore, discharge into basin no. 1 was larger than the average ($Q_{25\%}$ = 45,100 m^3 d^{-1}) by a factor of 1.27 and 1.46 times larger than that of basin no. 2. The maximum difference and standard deviation (σ) of flowrate in the basin were calculated to be 10.1% and 4.03, respectively.

It was also observed that significant deviation occurred among three rows in the basin: 59.8% of total discharge to basin no. 1 came into row no. 1-1. This discharge was 1.79 times larger than the average ($Q_{33.3\%}$ = 15,000 m^3 d^{-1}) in the three rows. Distribution rates of nos. 1–2 and 1–3 were 26.1% and 14.1% of the discharge into no. 1 basin, respectively. From row nos. 2–1 to 4–3, the distribution rate increased gradually towards downstream end owing to the dynamic pressure that affected the water level. Consequently, in basin no.1, there were two extreme rows with maximum (no. 1-1) and minimum discharge (no. 1–3) at the same time. The ratio of these two extreme discharges

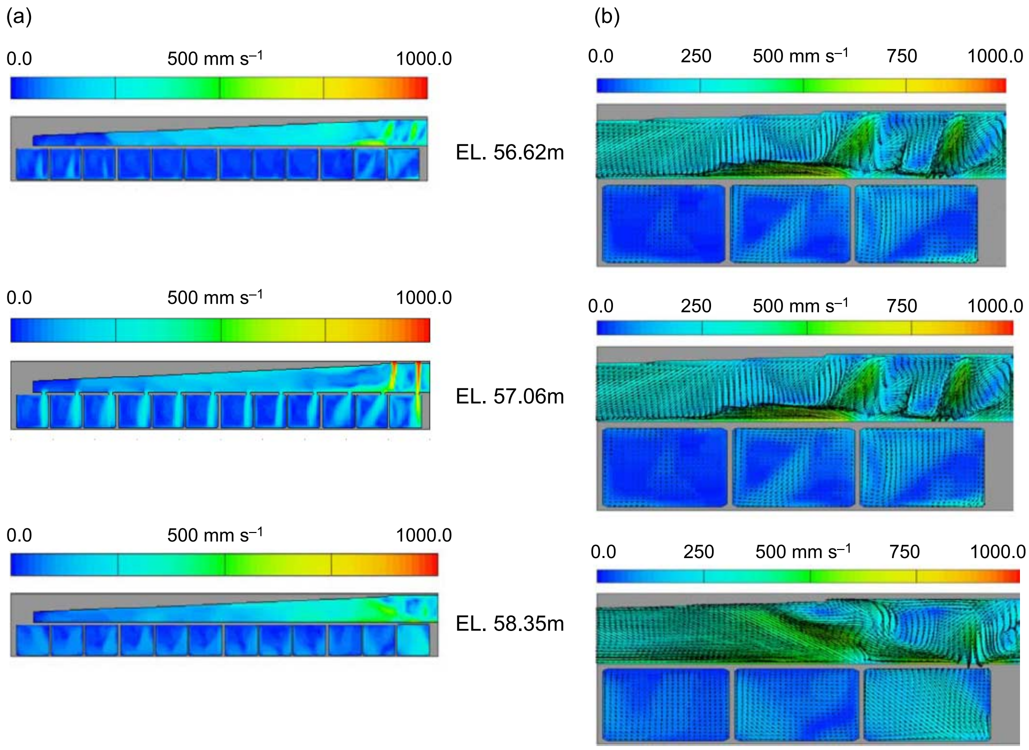

Figure 2 | Simulation results of flow fluid in the open channel; (a) right side channel (basin no. 1 ~ 4), (b) channel in front of basin no.1 (EL: elevation from bottom to water level).

was 4.2. A review of the hydraulic structure and fluid flow in the channel of the traditional method of introducing pipe flow to the centre of the open channel is needed for uniform distribution.

The y-directional (refer to coordinate in Figure 1) inflow velocities of over 1 m s^{-1} were gradually decreased to below 0.1 m s^{-1} at the downstream end. Also, it was observed that the inflow to the channel resulted in a sharp change of velocity in the channel. Flow patterns in the channel varied with width and section because of complicated flow such as eddy (see Figure 2). The velocity in the middle of the channel was 8–80 mm s^{-1}, while it was 500–600 mm s^{-1} alongside the wall with inlets. This rapid velocity hindered the even distribution to each row.

Verification of CFD simulation results

Total inflow in the actual plant during this study was 98,800 $m^3 d^{-1}$. As shown in Figure 4, the discharge into basin no. 1 from ADV measurement was 35,200 $m^3 d^{-1}$, 35.6% of total inflow, the largest inflow among the four basins. Flows to basin nos. 2–4 were 18.3%, 20.6% and 25.5%, respectively. The calculated flows to nos. 1–4 from the CFD simulation were 32.5%, 20.8%, 22.6% and 24.1%, respectively. Therefore, it could be concluded that CFD simulation was versatile enough to explain the trend of flow

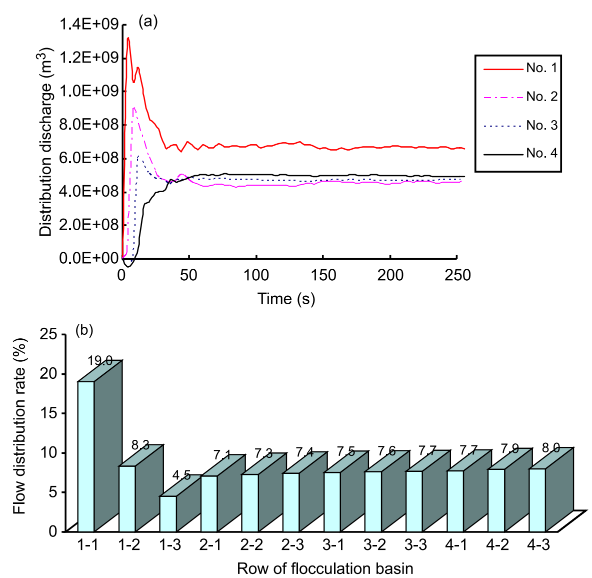

Figure 3 | Characteristics of flow distribution to basin and rows; (a) flocculation basin, (b) rows of flocculation basin.

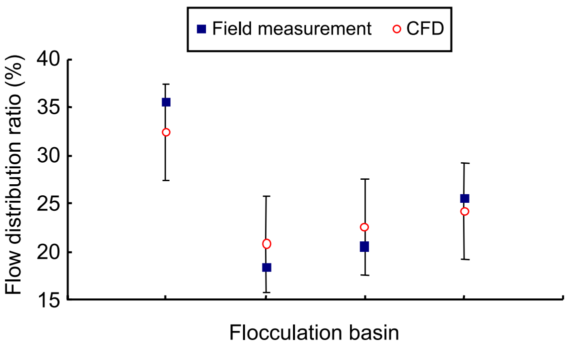

Figure 4 | Comparison of flow distribution between field measurement and CFD simulation.

distribution that was similar to the actual plant, although the largest simulation error was about 3%.

Distribution patterns by inlet type

Previous simulation results indicate that approaching velocity associated with the inlet structure of the distribution channel was an important factor affecting uniform flow distribution to flocculation basins. Various flow patterns simulated according to the different types of inlet structure are shown in Figure 5.

As was used previously, type A in which the flow comes into the open channel through a pipe at the perpendicular was not suitable as an inlet structure because of inequitable distribution. Another type with a bent pipe and vertical outlet (type B) showed a different distribution pattern from type A. Upflow from the pipe resulted in recirculation and back-water phenomena around the inlet of basin no. 1. The minimum discharge was observed in basin no. 1 while it was in basin no. 2 in type A. The other type of inlet structure with a weir against the influent (type C) could decrease the approaching velocity and result in a reduced deviation of distribution.

In type B, the maximum difference and standard deviation were 10.5%, 4.05 in the basin and 5.5%, 1.53 in

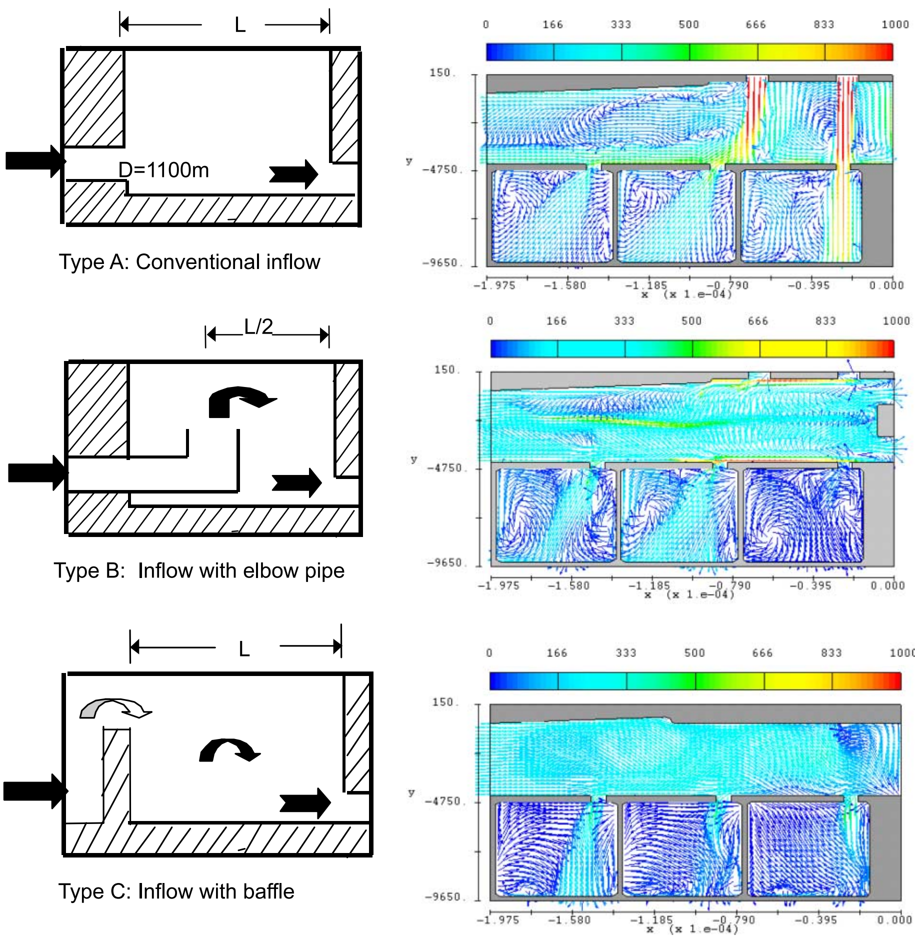

Figure 5 | Flow patterns in the channel according to the different types of inlet structure.

the rows, respectively. Although type B showed less deviation in the row than type A, the problem of inequitable distribution could not be solved satisfactorily. In type C, the maximum difference and standard deviation were 2.8%, 1.16 in the basin and 3.0%, 0.77 in the rows, respectively. Therefore, type C could improve equity of distribution compared with other types, although the deviation was over 2%.

This study attempted to improve inequitable distribution in the traditional type of open channel focusing on the flow fluid associated with inlet structure. There would be other factors affecting even distribution such as inlet location at the channel and basin, length of channel, number of basins and discharge rates. Therefore, studies on these factors are necessary for more effective distribution.

SUMMARY

This study was conducted to evaluate the equity of the flow distribution from rapid mixing basins to the flocculation basins. Several types of inlet structure of the open channel affecting the flow pattern and distribution trend were simulated using the CFD technique. Generally, the CFD simulation scheme derives solutions in conjunction with head loss and level variation. We did not mention these factors as they are not controlling factors in this research. The findings of this study are:

(1) From the results of CFD simulation and ADV measurement, the existing inlet geometry resulted in seriously inequitable distribution (refer to Figures 2 and 3). The largest deviations in the basins and rows were over 10%. It was proved that the inadequate inlet geometry could not achieve an equitable flow distribution in channels that were designed by the step method. This misdistribution might ultimately impair the treatment performance of basins.

(2) This study was attempted to solve the inequitable distribution problem and suggested proper inlet structure design to minimize the approaching velocity required for even distribution. Three cases, horizontal pipe type, elbow type and baffle type, are considered. As the results of simulations, the elbow pipe type still showed an inequitable distribution, while installing a baffle against the influent could reduce the deviation of flow distribution by decreasing velocity.

(3) The inequitable distribution to the flocculation basins is a serious problem faced in many water treatment plants. This inequitable distribution is related to inlet structure and flow behaviour in the open channel. In addition, a three-dimensional hydrodynamic such as CFD could refine and improve the existing design method for optimizing even flow. If necessary, a new design tool based on hydrodynamic analyses is also recommended for better performance.

REFERENCES

Baek, H. K. & Kim, W. K. 2000 *Rapid Mixing and Flocculation Basins, Composite Correction Program for Seoung-nam Water Treatment Plant*. Korea Water Resources Corporation, Deajeon, pp. 48–105.

Benefield, L. D., Judkin, J. F. & Parr, A. D. 1984 *Flow in Open Channels, Treatment Plant Hydraulics for Environmental Engineers*. Prentice-Hall, New York, pp. 108–122.

Chao, J. L. & Trussel, R. R. 1980 Hydraulic design of flow distribution channels. *J. Environ. Engng, ASCE* **106**, 321–333.

Currie, I. G. 1993 *Fundamental Mechanics of Fluids*. McGraw-Hill, New York.

Hirt, C. W. & Nichols, B. D. 1981 Volume of fluid (VOF) methods for the dynamics of free boundaries. *J. Comp. Phys.* **39**, 201–210.

Hudson, E. H., Jr 1981 *Water Clarification Processes Practical Design and Evaluation*. Van Nostrand Inc., New York, pp. 258–275.

Park, N. S., Park, H. & Kim, J. S. 2003 Examining the effect of hydraulic turbulence in a rapid mixer on turbidity removal with CFD simulation and PIV analysis. *J. Wat. Suppl.: Res. & Technol.-AQUA* **52**(2), 95–108.

Perry, R. H. 1984 *Chemical Engineer's Handbook*, 7th edition. McGraw-Hill, New York.

Sicilian, J. M., Hirt, C. W. & Harper, R. P. 1987 FLOW-3D: Computational modeling power for scientists and engineers, Flow Science Report (FSI-87-00-1), Flow Science, Inc.

Suhas, V. P. 1980 *Numerical Heat Transfer and Fluid Flow*. McGraw-Hill, New York.

Versteeg, H. K. & Malalasekera, W. 1995 *An Introduction to Computational Fluid Dynamics*. Prentice-Hall, New York.

First received 23 August 2004; accepted in revised form 6 June 2005