



## Numerical Analysis of Sediment Transport in Sewer Pipe

H. Bonakdari\*, I. Ebtehaj, H. Azimi

Department of Civil Engineering, Razi University, Kermanshah, Iran

### PAPER INFO

#### Paper history:

Received 12 October 2015

Received in revised form 14 November 2015

Accepted 19 November 2015

#### Keywords:

Sediment Transport

Non-deposition

Numerical Analysis

ANSYS-CFX

### ABSTRACT

The efficiency and economical performance of sewer systems is an essential issue in urban drainage. The need for sewers to carry sediment has been recognized for many years. One of the main problems in designing sewerage systems is the sediment deposition. Sedimentation in sewers occurs regularly according to the alternating natural flow. The present study investigates the hydraulic characteristics of flow in channels with a circular cross section with different bed slope and their effects on sediment transport capacity by use of 3D numerical simulation of flow field with ANSYS-CFX software. It studies hydraulic features of the flow passing through a circular channel in a two or three phase conditions. Self-cleansing velocity and volumetric sediment concentration in various Froude numbers were compared by lab outcomes to validate the results of numerical model. Results of numerical simulation indicate a proper adaptation of numerical and experimental models. Longitudinal velocity counters obtained by numerical simulation are compared in two or three phase flows.

doi: 10.5829/idosi.ije.2015.28.11b.03

## 1. INTRODUCTION

One of the most important issues to be considered in designing sewer channels is sedimentation of suspended particles. In wastewater channels, sedimentation occurs regularly based on normal flow conditions. Combined sewers, which create the main part of the wastewater conveyance channel, face variety of flow rates in different times of the operation periods. These different ranges and various sediments in many combined sewers results in the repetition of sedimentation, erosion, and transfer in different time durations. Generally, both storm and sanitary flows carry different kinds of particles into the sewer system. If the velocity of passing flow through the sewer channel was not enough or conduit slope was not much to prevent the transfer of suspended particles in flow, they settle down. If the sediments remain long in the wastewater channel, a change in sediments, especially during the absence of precipitation; permanent sedimentation on pipe bottom creates the changes in velocity and shear stress. Therefore, it affects sediment carrying capacity and hydraulic resistance of wastewater channel.

One of the easiest ways to prevent sediment deposition is using the constant velocity values, but due to the lack consideration of flow conditions and sediment properties, using this method, accompanies with no optimal and effective design [1]. Therefore, considering the efficient conditions on sediment transport, sewer system design using the self cleaning concept was performed. Self-cleansing is an important aspect of sanitary sewer design and is desired to minimize the deposition of silt, sediment, and debris. It is generally defined in two ways: First, limit of deposition (Ota and Nalluri [2], Almedej and Almohsen [3], Ota and Perrusquia [4]) and second is bed deposited (Nalluri et al. [5] Ota and Nalluri [6]). May [7] developed a design method on sediment movement in horizontal or nearly horizontal pipes. Enfinger et al. [8] used the tractive force method to design sewers with self-cleansing conditions based on a critical shear stress. This method can be extended from the design of new sewers to the evaluation of existing sewers under actual conditions. Almedej [9] suggested a self-cleansing design procedure for rectangular sewers based on the sediment transmit. Ebtehaj and Bonakdari [10] predicted sediment transport in sewer circular conduit by artificial neural network (ANN). Ebtehaj and Bonakdari [11] investigated performance evaluation of adaptive neural

\*Corresponding Author's Email: Bonakdari@gmail.com (H. Bonakdari)

fuzzy inference system (ANFIS) for sediment transport in sewers pipe. Ebtehaj and Bonakdari [12] compared the genetic algorithm (GA) and imperialist competitive algorithms in predicting bed load transport in clean pipe. Bonakdari and Ebtehaj [13] compared two data-driven approaches in estimation of sediment transport in sewer circular channels. In recent years, the computational fluid dynamics (CFD) have been extensively used in the various engineering field [14-16]. By using Fluent software, Bardiaux et al. [17] simulated three dimensional flow turbulent in sewer networks using  $k-\varepsilon$  and Reynolds stress model. Bonakdari et al. [18] employed a numerical method to compute distribution of shear stress in sewers by ANSYS-CFX software. Zhang et al. [19] studied turbulent flow with settled particles on in curved channels using Fluent software and used RSM turbulent model to investigate the flow pattern of 180 degree bend and the effects of turbulent fluctuations on particle deposition. Chen et al. [20] developed a CFD approach to implement a 3D numerical model for optimizing design changes of a combined sewer system. The authors is modeled the behavior of suspended solids using a particle tracking approach. Azimi et al. [21] simulated flow field within a circular channel along a side weir using volume of fluid scheme (VOF) and RNG  $k-\varepsilon$  turbulence model. Azimi et al. [22] modeled the free surface flow and flow pattern along the side weir in a circular channel by RNG  $k-\varepsilon$  model and VOF method in subcritical flow regime. The 3D free surface flow and flow turbulence in triangular channels with side weir simulated by Azimi and Shabanlou [23]. They investigated the variations of free surface and flow behavior vicinity of side weir in subcritical flow condition. Sharifipour et al. [24] numerically studied impact of the confluence angle on flow field and flowmeter accuracy in open channel junctions.

In experimental studied, researchers have been estimated the minimum required velocity to prevent the sediment deposition in the channel bed which is expressed as densimetric particle Froude number ( $Fr=V/(g(s-I)d)^{0.5}$ ). However, these studies have no information about the minimum required velocity to prevent sediment deposition in sewer systems. Therefore, in this numerical study, ANSYS-CFX 14.0 software was used to simulate three dimensional and three phases flow (water+air+sediment) passing through a circular channel. Flow range turbulent was modeled using  $k-\varepsilon$  model. To validate the result of numerical model, flow velocity and sediments concentration in different Froude numbers were compared with experimental results. In the following, velocity field obtained from two and three phases modeling were compared. Also, changes in depth of sediment phase along the channel were studied.

## 2. MATERIALS AND METHODS

With the great development of computer capabilities in the last decades, the numerical computation of fluid flows, generally known as computational fluid dynamics (CFD), has become a common practice in engineering. The study of two and three dimensional flow in open channels has recently experienced a surge of interest in the application of CFD to practical problems in hydraulic and fluid mechanics. The interactions of the flow and sediment in the channel bed lead to complex flow characteristics in sewers (e.g. free surface variations, secondary flows and suspended solids transport). In this study, three-phase turbulent flow within a circular sewer channel using ANSYS CFX software is simulated. It solves the three-dimensional turbulent flow equations and utilizes a collocated and cell- centered storage scheme with a finite- volume discretization. In general, finite volume method uses conservation integral equations form as a starting point. In this method, shape functions are used to describe the variables changing from an element. Control volumes around each node are formed by connecting the geometric elements centers. Also, crossing fluxes from volume control borders and source statements are calculated based on elements as well. The equations for each time step are discrete and solved within a fully coupled system; answers are calculated during the replication process.

**2. 1. Governing Equations** The cornerstone of computational fluid dynamics is the fundamental governing equations of fluid dynamics. Partial differential equations describing the flow are written in a conservative form, to establish relations between the pressure, velocities and Reynolds stress [25]. The form of partial derivative equations for biphasic application is as follows: The continuity equation for each phase which is called q:

$$\left\{ \begin{array}{l} \frac{\partial \alpha_q}{\partial t} + U_i \frac{\partial \alpha_q}{\partial x_i} = 0 \\ 0 \leq \alpha_q \leq 1 \\ \sum_{q=1}^n \alpha_q = 1 \end{array} \right. \quad (1)$$

where,  $n$  is the number of phases,  $U_i$  the mean velocity components and  $\alpha_q$  is the volume fraction of phase  $q$ . In each cell, the overall volume mass  $\rho$  and viscosity  $\mu$  are computed using the volume fraction as follows:

$$\left\{ \begin{array}{l} \rho = \sum_{q=1}^n \alpha_q \rho_q \\ \mu = \sum_{q=1}^n \alpha_q \mu_q \end{array} \right. \quad (2)$$

The momentum equation:

$$\frac{\partial \rho U_i}{\partial t} + U_i \frac{\partial \rho U_i}{\partial x_i} = -\frac{\partial P}{\partial x_i} + \rho g_i + \mu \frac{\partial^2 U_i}{\partial x_j \partial x_j} - \frac{\partial \rho \overline{u_i' u_j'}}{\partial x_j} \quad (3)$$

where,  $P$  is the pressure term and  $g$  is the gravitational acceleration. Equation (3) represents the Reynolds Averaged Navier-Stokes (RANS) equations system (for  $i$  and  $j$  equal to 1, 2 and 3). The terms called Reynolds tensors can be estimated by means of closing equations such as  $k-\epsilon$  turbulent model. This software uses the finite volume method for solving partial derivative equations presented above. Therefore, the computational meshes as volumes of control must be constructed.

**2. 2. Geometry and Computational Cells** Lab geometry is set in the ICEM CFD code as the pre-processing step of the CFD simulation. To investigate the exactness of numerical model, Equations (4) and (5) were used, respectively, to estimate Root Mean Square Error ( $RMSE$ ) and Mean Absolute relative Error ( $MARE$ ):

$$RMSE = \sqrt{\frac{\sum_{i=1}^N (R_{EXPi} - R_{CFXi})^2}{N}} \quad (4)$$

$$MARE = \left(\frac{1}{n}\right) \sum_{i=1}^n \left(\frac{|R_{EXPi} - R_{CFXi}|}{x_i}\right) \quad (5)$$

where,  $R_{EXP}$  and  $R_{CFX}$  are results of experimental study and numerical simulation, respectively. The solving field was dismissed by a block mesh consisted of the cubic elements. Used mesh characteristics in numerical modeling viz. Root mean square error percent ( $RMSE$ ) and Mean Absolute relative Error ( $MARE$ ) percent for sediment volume density values can be seen from Table 1. It can be concluded from this Table that regarding to the computational cell numbers and low descendants between the grid 4 and 5 results, grid 4 for separation computational field is selected as well.

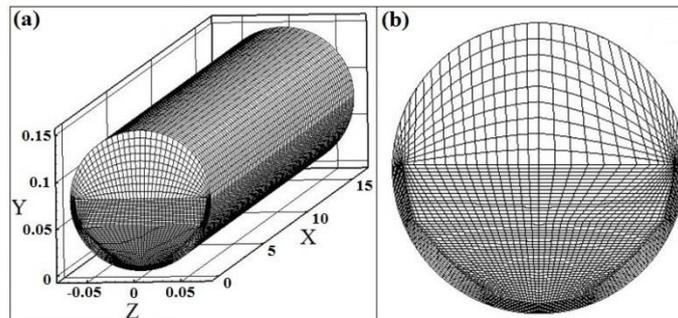
Figure 1 shows computational mesh number 4 in a circular channel in directions of X, Y, and Z by number of elements of 3200, 90, 100, respectively which used for this study.

**2. 3. Boundary Conditions** In this study, to verify the CFD model, the results of the experimental model of Vongvisessomjai et al. [25] is used. Also, applied boundary conditions to numerical models were chosen in a way that is in agreement with the physical conditions in vitro model. As all boundary conditions of numerical model were selected using a trial and error process, thus, the difference between experimental and numerical results reached to minimum value. Therefore, the first boundary condition ‘velocity-inlet’ is an imposed value of the velocity. Thus, the flow is injected through a wet section to obtain the expected inlet flow rate. In this case, the length of the inlet pipe must be sufficient to enable the velocity profile to be developed. The length required is considered 16 m as sets in experimental condition.

At the outlet, opening boundary condition is used which equals to the given flow depth. In this boundary condition, fluid phase is always defined as opening boundary conditions with zero Pascal relative pressure. All channel solid conditions are set as wall boundary condition.

**TABLE 1.** Mesh characteristics,  $RMSE$  and  $MARE$  for sediment volume density

Meshing	Number of cells	$RMSE$	$MARE$
1	337500	28.37	33.07
2	975000	19.74	25.60
3	1912500	12.08	17.93
4	2880000	7.31	13.40
5	3811500	6.53	12.16



**Figure 1.** Computational mesh of the circular channel (a) three dimensional view (b) Y-Z plane

In wall boundary conditions, no slip wall condition was used to introduce boundary condition at the section. Knowing that channel upper surface encounters atmosphere pressure in experimental condition, opening condition boundary with zero relative Pascal pressure and turbulence intensity of 0.05 are used.

**3. EXPERIMENTAL MODEL**

In order to validate, the present study makes use of numerical results driven from experimental data of Vongvisessomjai et al. [25]. Vongvisessomjai et al. [25] conducted their tests on pipes in two sizes 100 and 150mm in diameter and 16m in length. They employed two sections to measure the flow, one at a distance of 4.5m upstream and the other at the distance of 5.5m downstream. These two points were 6 m apart. In each section, the velocities were measured at flow surface, middle depth and near bottom and their mean average was taken as the average velocity. For the air/water phase of the flow, the Manning coefficient of roughness (*n*) was equal to 0.0125. To imply depositing phase, sand particle having average diameters of 0.2, 0.3 and 0.43mm were used. Vongvisessomjai et al. [25] tests were conducted in a non-deposited bed state. Vongvisessomjai et al. [25] used parameters which have the most influence for sediment transport in the limit of deposition and use of SPSS software to present the following equation that employs regression analysis:

$$\frac{V}{\sqrt{g(s-1)d}} = 4.31C_V^{0.226} \left(\frac{d}{R}\right)^{-0.616} \tag{6}$$

$$\frac{V}{\sqrt{g(s-1)d}} = 3.57C_V^{0.21} \left(\frac{d}{y}\right)^{-0.542} \tag{7}$$

Using nonlinear regression in MINITAB and wide range of dataset [2, 25, 26], Ebtehaj et al. [1] preented following equation:

$$\frac{V}{\sqrt{g(s-1)d}} = 4.49C_V^{0.21} \left(\frac{d}{R}\right)^{-0.54} \tag{8}$$

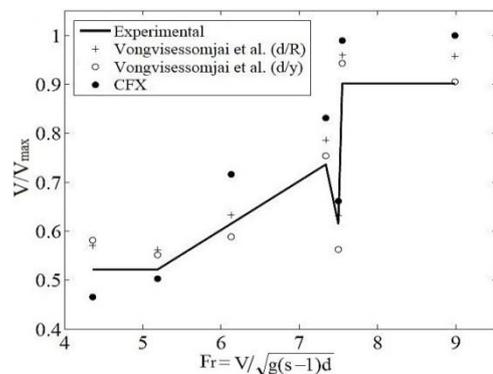
**4. RESULT AND DISCUSSION**

Figure 2 compares the numerical results of self-cleansing obtained from simulation, equations introduced by Vongvisessomjai et al. [25] and experimental results.

According to self-cleansing velocity resulted from numerical simulation, *RMSE* is equal 0.07 and *MARE* is equal 0.07. Besides validating numerical model, relative velocity resulted from given Equations (6), (7) and (8)

in order to transfer sediments up to depositing were compared with experimental results. Validating criteria from Equations (6) and (8) are *RSME*=0.044 and *MARE*=0.05 and *RSME*=0.13 and *MARE*=0.15 (respectively), where relative depth this *d/R*, and for Equation (7) *RMSE*=0.038 and *MARE*=0.05, where relative depth is *d/y*. As is seen, numerical model predicts the self-cleansing velocity with acceptable exactness. The least error by numerical model refers to descending numbers larger than 5 which is about 0.03, and the largest error in studying self-cleansing velocity by numerical model in comparison with experimental studies is about 0.16, whereas the largest error from equations introduced by Vongvisessomjai et al. [25] is 10 and 11 percent, for Equations (6) and (7), respectively and for the equation of Ebtehaj et al. [1] is 28%. It should be noticed that the exactness of the introduced equations is calculated by experiments used to estimate the equations. Therefore, one could say that the introduced numerical model suggests an exact average flow velocity estimation in comparison with experimental and given equations. The characteristics of experimental model that CFD model is defined based on it shwon in Table 2.

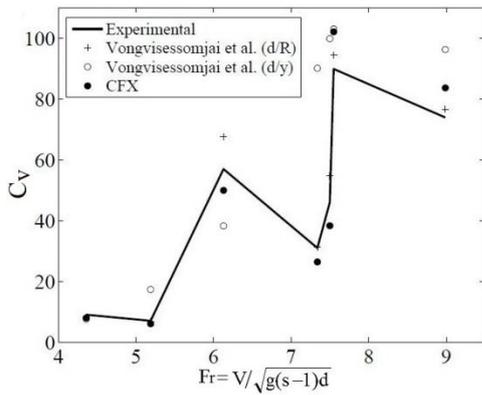
Figure 3 Compares sediment volume density from numerical simulation and experimental studies. Considering given validating criteria (Equations (4) and (5)) for volumetric sediment concentration, numerical results offer *RMSE*=7.31 and *MARE*=13.4. Besides numerical model validation, resulting relative velocity from suggested equations to transfer sediments at depositing limits (Equations (6), (7) and (8)) was compared to experimental results. Validation criteria for Equations (6) and (8) are *RMSE*=6.91 and *MARE*=0.08 and *RMSE*=135 and *MARE*=1.95 (respectively), where relative depth is *d/R*, and for Equation (7) equal to *RMSE*=63.54 and *MARE*=0.42 where relative depth is *d/y*.



**Figure 2.** Comparing self-cleansing velocity resulted from numerical and experimental results in different descending numbers

**TABLE 2.** Characteristics of the experimental model of Vongvisessomjai et al. [25]

$Q$ (L/s)	$y$ (m)	$V$ (m/s)	$d$ (mm)	$C_v$ (ppm)	$Fr$ (-)
1.08	0.03	0.43	0.20	46	7.50
1.29	0.06	0.51	0.30	31	7.33
2.81	0.03	0.43	0.30	57	6.13
4.13	0.06	0.63	0.30	74	8.98
2.39	0.06	0.36	0.43	9	4.34
3.37	0.06	0.51	0.43	40	6.13
1.08	0.03	0.43	0.43	69	5.12
4.13	0.06	0.63	0.43	90	7.50
0.96	0.06	0.33	0.43	7	3.93



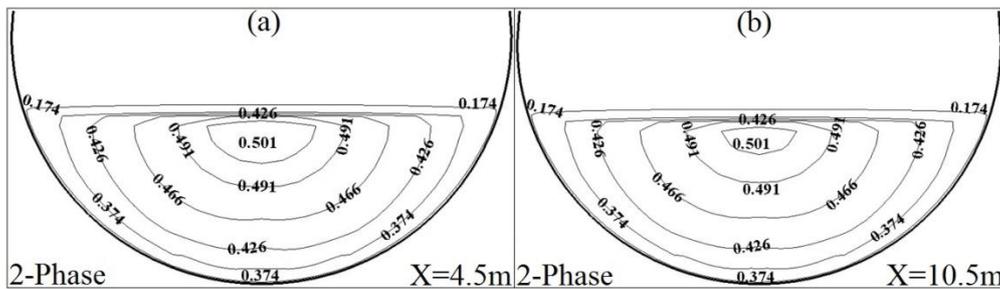
**Figure 3.** Comparing volumetric sediment concentration from numerical and experimental results for different descending numbers

As seen in the picture, numerical model shows proper results in comparison with Equation(6), where relative flow depth is  $d/R$ , and experimental results. Notice that an equation which considers relative flow depth as  $d/y$  introduces higher computed error in

comparison with results of numerical model. Considering estimated error, numerical model offers exactness in computing sediment volume density. Maximum and minimum errors introduced by numerical model are about 0.16 and 0.11, respectively. According to the picture, it is seen that an equation which defines relative flow depth as  $d/y$ , does not introduce exact results. One should consider that the exactness of the results from suggested equations is computed by experiments used to estimate the equations. Therefore, it could be concluded that suggested numerical model presents exact measurement of average flow velocity in comparison with experimental results and ones driven by suggested equations.

Figures 4 and 5 show velocity longitudinal component counters for two and three phases flows in  $x=4.5$  m and  $x=10.5$ m sections, respectively. As is seen, velocity values decreased with the addition of sediment phase (Figure 5). The highest decrease in longitudinal velocity in three-dimensional flow relative to two-dimensional flow occurs near channel bed. Velocity decreases as the result of the third phase called sediment phase. Increasing the flow depth relative to channel bed, there is smaller decrease in longitudinal velocity in three phase flow in comparison with two phase flow. As the result, velocity decreases in the given section in three phase flow. According to simulation results, maximum longitudinal velocity happens near free surface, and the minimum is measured near channel bed. Simulation results show that changing pattern of velocity profile in permanent steady flows for three dimensional flow is similar to changing pattern of velocity profile in two dimensional flow.

Figure 6 shows the changes over channel bed plate in a three phase flow along the channel. Here, horizontal axis introduces channel length relative to entering flow depth, and the vertical axis offers the ratio of sediment depth to entering water depth as a dimensionless component. It could be seen that sediment phase balance along the channel is fluctuating which presents sediment transmit over channel bed in a three dimensional flow.



**Figure 4.** Longitudinal velocity component counters in two phase flow (a)  $x=4.5$  m (b)  $x=10.5$  m

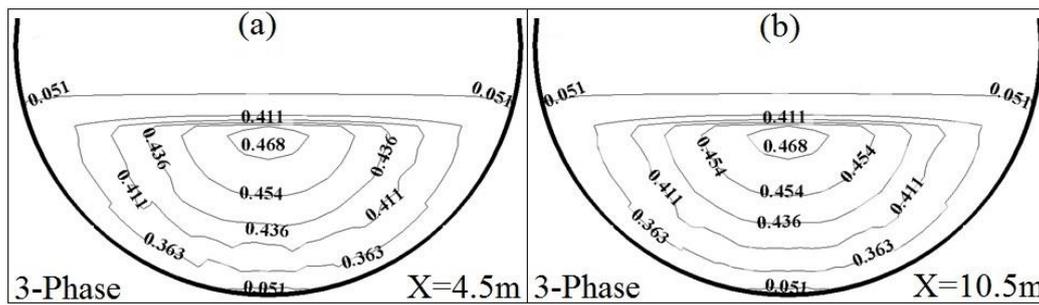


Figure 5. Longitudinal velocity component counters in three phase flow (a)  $x=4.5m$  (b)  $x=10.5m$

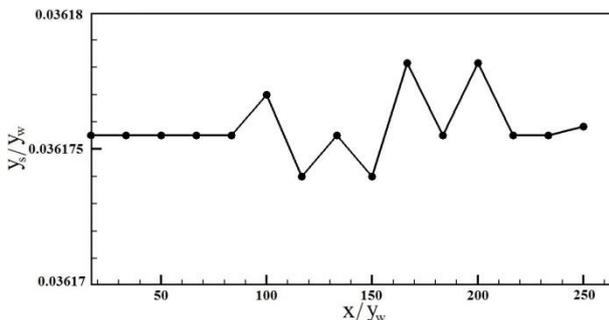


Figure 6. Changes of channel bed in three-dimensional flow along channel bed

## 5. CONCLUSION

Present numerical study used standard  $k-\epsilon$  turbulent model and ANSYS-CFX 14.0 to simulate turbulence and range of passing flow through a circular channel in two and three phase flows three dimensionally. The experimental results of Vongvisessomjai et al. [25] were used to validate numerical results. Comparing the results of numerical model to experimental results, root mean square error and mean average percent error were  $RMSE=0.07$  and  $MARE=0.07$  for self-cleansing velocity and  $RMSE=7.31$  and  $MARE=13.4$  for volumetric sediment concentration. In comparison with two phase flow, longitudinal velocity was smaller near channel bed in three phase flow as the result of sediment phase. Based on numerical simulation results, in both two and three dimensional flows, maximum and minimum velocity occurs at free surface and bed of circular channel, respectively. Longitudinal velocity profile pattern of flow in three dimensional conditions was simulated similar to velocity profile pattern of permanent two-phase steady flows by numerical model. Present numerical modeling studied the changes of channel bed balance in three phase flow condition where fluctuations on bed plate introduced sediment transfer near circular channel bed.

## 6. REFERENCES

1. Ebtehaj, I., Bonakdari, H. and Sharifi, A., "Design criteria for sediment transport in sewers based on self-cleansing concept", *Journal of Zhejiang University SCIENCE A*, Vol. 15, No. 11, (2014), 914-924.
2. Ota, J.J. and Nalluri, C., "Graded sediment transport at limit deposition in clean pipe channel. 28th international association for hydro-environment engineering and research", *Graz, Austria*, (1999).
3. Almedeij, J. and Almohsen, N., "Remarks on camp's criterion for self-cleansing storm sewers", *Journal of irrigation and drainage engineering*, Vol. 136, No. 2, (2009), 145-148.
4. Ota, J. and Perrusquia, G., "Particle velocity and sediment transport at the limit of deposition in sewers", *Water Science and Technology*, Vol. 67, No. 5, (2013), 959-967.
5. Nalluri, C., Ab Ghani, A. and El-Zaemey, A., "Sediment transport over deposited beds in sewers", *Water Science & Technology*, Vol. 29, No. 1-2, (1994), 125-133.
6. Ota, J.J. and Nalluri, C., "Urban storm sewer design: Approach in consideration of sediments", *Journal of Hydraulic Engineering*, Vol. 129, No. 4, (2003), 291-297.
7. May, R.W., "Preventing sediment deposition in inverted sewer siphons", *Journal of Hydraulic Engineering*, Vol. 129, No. 4, (2003), 283-290.
8. Einfinger, K.L. and Mitchell, P.S., "Scattergraph principles and practice: Evaluating self-cleansing in existing sewers using the tractive force method", in *World Environmental and Water Resources Congress*, Providence, Rhode Island, USA., (2010), 4458-4467.
9. Almedeij, J., "Rectangular storm sewer design under equal sediment mobility", *American Journal of Environmental Sciences*, Vol. 8, No. 4, (2012), 376-384.
10. Ebtehaj, I. and Bonakdari, H., "Evaluation of sediment transport in sewer using artificial neural network", *Engineering Applications of Computational Fluid Mechanics*, Vol. 7, No. 3, (2013), 382-392.
11. Ebtehaj, I. and Bonakdari, H., "Performance evaluation of adaptive neural fuzzy inference system for sediment transport in sewers", *Water Resources Management*, Vol. 28, No. 13, (2014), 4765-4779.
12. Ebtehaj, I. and Bonakdari, H., "Comparison of genetic algorithm and imperialist competitive algorithms in predicting bed load transport in clean pipe", *Water Science & Technology*, Vol. 70, No. 10, (2014), 1695-1701.
13. BONAKDARI, H. and EBTEHAJ, I., "Comparison of two data-driven approaches in estimation of sediment transport in sewer pipe".

14. Heidarinejad, G. and Delfani, S., "Numerical simulation of the wake flow behind an ellipse using random vortex method (research note)", *International Journal of Engineering-Transactions A: Basics*, Vol. 16, No. 2, (2003), 183-192.
15. Hadian, M.R., Zarrati, A. and Eftekhari, M., "Development of an implicit numerical model for calculation of sub-and super-critical flows", *International Journal of Engineering*, Vol. 18, No. 1, (2005), 775-784.
16. Nassab, S.G. and Atashafrooz, M., "Three dimensional laminar convection flow of radiating gas over a backward facing step in a duct", *International Journal of Engineering-Transactions A: Basics*, Vol. 25, No. 4, (2012), 399-410.
17. Bardiaux, J.B., Mose, R., Vazquez, j. and Wertel, J., "Tow turbulent flow 3d-modeling to improve sewer net instrumentation", in 11th International Conference on Urban Drainage, Edinburgh, Scotland, UK., (2008 of Conference).
18. Bonakdari, H., Larrarte, F. and Joannis, C., "Study of the shear stress in narrow channels: Application to sewers", *Urban Water Journal*, Vol. 5, No. 1, (2008), 15-20.
19. Zhang, P., Gros, Y., Roberts, R. and Bénard, A., "Modeling of turbulent flow with particle deposition in curved pipes", in 7th International Conference on Multiphase Flow, Tampa, FL., (2010).
20. Chen, Z., Han, S., Zhou, F.-Y. and Wang, K., "A cfd modeling approach for municipal sewer system design optimization to minimize emissions into receiving water body", *Water Resources Management*, Vol. 27, No. 7, (2013), 2053-2069.
21. Azimi, H., Shabanlou, S. and Salimi, M.S., "Free surface and velocity field in a circular channel along the side weir in supercritical flow conditions", *Flow Measurement and Instrumentation*, Vol. 38, (2014), 108-115.
22. Azimi, H., Hadad, H., Shokati, Z. and Salimi, M.S., "Discharge and flow field of the circular channel along the side weir", *Canadian Journal of Civil Engineering*, (2015).
23. Azimi, H. and Shabanlou, S., "The flow pattern in triangular channels along the side weir for subcritical flow regime", *Flow Measurement and Instrumentation*, (2015).
24. Sharifipour, M., Bonakdari, H. and Zaji, A., "Impact of the confluence angle on flow field and flowmeter accuracy in open channel junctions", *International Journal of Engineering-Transactions B: Applications*, Vol. 28, No. 8, (2015), 1145-1153.
25. Vongvisessomjai, N., Tingsanchali, T. and Babel, M.S., "Non-deposition design criteria for sewers with part-full flow", *Urban Water Journal*, Vol. 7, No. 1, (2010), 61-77.
26. Ghani, A.A., "Sediment transport in sewers", (1993).

## Numerical Analysis of Sediment Transport in Sewer Pipe

H. Bonakdari, I. Ebtehaj, H. Azimi

Department of Civil Engineering, Razi University, Kermanshah, Iran

### PAPER INFO

چکیده

#### Paper history:

Received 12 October 2015

Received in revised form 14 November 2015

Accepted 19 November 2015

#### Keywords:

Sediment Transport

Non-deposition

Numerical Analysis

ANSYS-CFX

عملکرد موثر و اقتصادی سیستم‌های فاضلاب یکی از مسائل مهم در زهکشی شهری به حساب می‌آید. استفاده از سیستم‌های فاضلاب به منظور انتقال رسوب از سال‌ها پیش مورد توجه بوده است. یکی از مشکلات اساسی در طراحی این سیستم‌ها، ته نشینی رسوب در بستر کانال است. رسوبگذاری با توجه به جریان متناوب طبیعی اتفاق می‌افتد. در این مطالعه مشخصات هیدرولیکی جریان در کانال دایره‌ای با شیب‌های مختلف و اثر آن بر روی ظرفیت انتقال رسوب با استفاده از شبیه‌سازی سه بعدی میدان جریان با استفاده از ANSYS-CFX بررسی می‌شود. در این شبیه‌سازی عددی از مدل آشفتگی k-ε استاندارد استفاده شده و مشخصات هیدرولیکی جریان عبوری از داخل یک کانال دایره‌ای در شرایط دو و سه فازی مورد بررسی قرار گرفته است. به منظور اعتبار سنجی نتایج مدل عددی سرعت خودشویی و غلظت حجمی رسوب در عدد فرودهای مختلف با نتایج آزمایشگاهی مقایسه شده است. نتایج حاصل شبیه‌سازی عددی نشان دهنده انطباق مناسب مدل عددی با مدل آزمایشگاهی است. در ادامه کانتورهای سرعت طولی شبیه‌سازی شده در جریان‌های دو و سه فازی با یکدیگر مقایسه شده است. همچنین پروفیل‌های سرعت در دو حالت جریان دو و سه فازی و تغییرات تراز بستر کانال مورد مطالعه قرار گرفته است.

doi:10.5829/idosi.ije.2015.28.11b.03