DOI: 10.24352/UB.OVGU-2019-011 TECHNISCHE MECHANIK, 39, 1, (2019), 113 - 124
submitted: July 6, 2018

Numerical and Experimental Investigation of Flow in Partially Filled Sewer Pipes

M. Alihosseini, P.U. Thamsen

Complex phenomena in wastewater systems, such as flow pattern in sewers and sediment transport could be investigated in detail using computational fluid dynamics (CFD). However, it is not easy to find an appropriate CFD model for a specific problem. This paper aims to develop and validate a CFD model to correctly predict the free-surface turbulent flow passing through a circular pipe. In this study, the multiphase model Volume of Fluid (VOF) of the software Ansys-Fluent was used to capture the interface between air and water. Different variants of the k-ε turbulence model of the RANS group and meshing approaches were investigated. To validate the CFD model, a set-up of an acryl-glass pipe in a closed system was constructed under laboratory conditions. The centre-plane velocity profile was used to compare the CFD model results and Laser Doppler Velocimetry (LDV) measurements. Furthermore, the values of the average velocity and shear stress from the experiments were compared to the results of the CFD model. The best results were obtained using a Cutcell mesh combined with the RNG k-ε turbulence model. The validated model was used to investigate the influence of the bed roughness on the velocity and shear stress distribution in partially filled pipes. The velocity decreases while increasing the bed roughness, however the shear stress becomes greater over a rough bed than over a smooth bed.

1 Introduction

The complex process of flow and sediment transport in sewer systems has been investigated using in-situ measurements, laboratory experiments and numerical modelling. All of these methods have some shortcomings. In-situ measurements are mostly site-specific and are not predictive. The flow in the channel is non-stationary and unsteady, which prevents representing the measurements (Yan et al., 2014). In contrast, the laboratory experiments are more representative and precise. However, the results from the small-scale physical models are not always transferable to real complex systems. With increasing computational resources and advancements in numerical methods, computational fluid dynamics (CFD) provides a good alternative to investigate complicated phenomena in a less expensive and more flexible way. In the following, a review of the application of CFD concerning the simulation of sewer systems is briefly presented.

Schmitt et al. (1999) studied the bed load sediment traps in sewer systems using the Volume of Fluid (VOF) method of the software Ansys Fluent. The numerical results were in good agreement with previous empirical findings and led to a better design of sediment traps.

Berlamont et al. (2003) calculated boundary shear stresses for circular pipes with a flat-bed using CFD. They used experimental data from the literature to verify the CFD model of a package PHONICS. The k-ε turbulence model was used. The results were satisfying, although secondary currents were neglected, since they do not have significant influence on the shear stress distribution but rather on sediment transport.

He et al. (2004) studied the behaviour of flow and sediment in a combined sewer overflow (CSO) facility using CFD. They used the VOF and the Discrete Phase Method (DPM) of Fluent to investigate different flow fields and particle capture rates in order to assess flow conditioning baffles implemented in the CSO facility. They found CFD modelling as an attractive alternative for optimizing the existing CSO facilities.

Schaffner et al. (2004) studied the application of CFD for the calculation of flush waves in sewers using the software StarCD. VOF and the k-ε turbulence model were used. They concluded that the results of CFD lead to a better design and dimensioning of flushing devices. Kirchheim et al. (2005) used the same CFD package to calculate different scenarios for a flushing device used in combined sewers to avoid deposits and investigate the cleaning capacity. The parameters influencing the shear stress produced by flushing waves were numerically analysed.

Bardiaux et al. (2006) obtained the velocity profiles through a sewer channel using the software Fluent. They compared the k-ε and Reynolds Stress (RS) turbulence models as well as the monophasic approach obtained with the symmetry plane boundary condition and the VOF method. The comparison with the experimental data showed that RSM and VOF together could accurately simulate the water behaviour in open channels. The anisotropic RS model could represent the secondary currents better than the isotropic k-ε model.

Bares et al. (2006) verified a 3D CFD model of Fluent to obtain the flow pattern of a combined sewer overflow. The free surface was simulated as a wall without friction and the standard *k-ω* turbulence model was chosen as the best suited model. They used the Ultra Doppler Method (UDM) to visualize the 2D flow field and compared the results with numerical results. The time-averaged velocity flow field could be obtained in a good way.

Bardiaux et al. (2008) used VOF combined with the RS turbulence model to simulate turbulent flows to improve sewer net instrumentation. They modelled different flows in rectangular, circular and egg-shape sections. The method presented allows determining the local velocity of a flow in any point of a cross section.

Jarman et al. (2008) reviewed the application of CFD in modelling of urban drainage systems. In all of the reviewed studies, the flow was assumed to be turbulent and the RS or *k-ε* turbulence models were used. They found the VOF approach very accurate in simulating the free surface, but computationally expensive. The review showed that by growing the computational resources, CFD is becoming increasingly utilized as part of larger simulation schemes for multi-physical systems.

Dufresne et al. (2009) studied the flow, sedimentation and solids separation in a combined sewer detention tank using Fluent. They used the particle tracking facility of the software to study the location of the deposited particles and the pollutant load. Agreeable results showed that CFD is a good way to model sediment transport in sewer systems.

Chen et al. (2013) developed a 3D numerical model for optimizing design changes of a combined sewer system. The VOF method and the RNG *k-ε* turbulence model were used to simulate the turbulent free surface. A particle tracking approach was used to model the behaviour of suspended solids. They concluded that CFD is effective in designing a combined sewer system to reduce pollutant discharge into the receiving waters.

Bonakdari et al. (2015) numerically studied the minimum velocity required to prevent sediment deposition in sewer systems. They simulated a three phases flow (water, air, sediment) in a circular pipe using the *k-ε* turbulence model of Ansys-CFX. The modelled longitudinal velocity profile pattern of flow and volumetric sediment concentration were in good agreement with the experimental data.

Mohsin and Kaushal (2016) used VOF and DPM of Fluent to predict the efficiency of an invert trap. They commented that the realizable *k-ε* turbulence model is an appropriate choice for transferring turbulence among the phases.

Regueiro-Picallo et al. (2016) analysed the open channel flow in egg-shaped pipes for small combined sewer systems using the Ansys CFX 3D model. The VOF model was used to track the interface between water and air. The shear stress and velocity profiles obtained from the CFD model were in good agreement with the experimental results.

However, there are limited studies investigating the behaviour of flow in a partially filled circular pipe using CFD. Compared to the velocity distribution in the full-filled pipe flow, the velocity distribution in a partially filled pipe has been less researched (Jiang et al., 2016). Due to the fact that the free surface is always changing with time and space, it is much more difficult to model the flow in open channels. Most of the studies investigating the flow in open channels have been done in prismatic channels, which make the calculations simpler. It is very difficult to investigate the velocity distribution along the width of an open channel theoretically (Gandhi et al., 2010).

This study aims to develop a CFD model to predict the free-surface turbulent flow in a circular pipe. The investigations have examined the influence of grid size, structure and turbulence model. After obtaining the best suited CFD model, this model is used to investigate the influence of the bed roughness on flow behaviour in open channels.

2 Materials and Methods

2.1 Laboratory Experiment

To validate the CFD model, in the laboratory of the Chair of Fluid System Dynamics of the Technische Universität Berlin, a set-up of an acryl-glass-pipe in a closed system was constructed. The model consists of a 6000 *mm* long circular pipe with an internal diameter (d) of 342 *mm*. At the beginning of the pipe, an inlet tank and at the downstream, a collecting tank are placed. Figure 1 shows the experimental set-up. The bed slope S_0 is variable; it is 0.37% for this study.

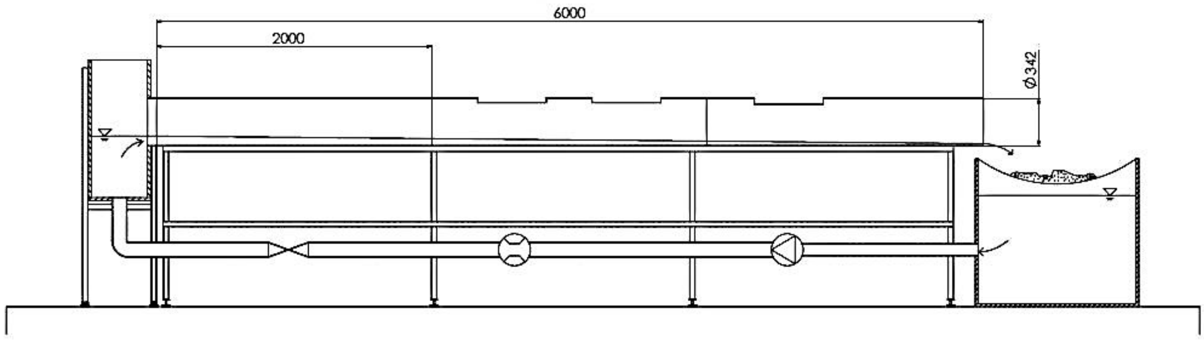

Figure 1. Construction of the experimental set-up (in *mm*)

The values of discharge (Q) were obtained using an MID flowmeter (Krohne Altometer SC 80 AS) and a compact echo sounder (NivuCompact Echolot) was used to determine the water height (h). The average flow velocity (V) and shear stress over the wetted perimeter (τ) can be expressed using the experimental data as follows:

$$V = \frac{Q}{A} \tag{1}$$

$$\tau = \rho g R_h S_f \tag{2}$$

where A is the cross-sectional area of flow which is dependent on the opening angle of the water (θ) and the radius of the pipe (r), ρ is the fluid density, g the gravitational acceleration and R_h is the hydraulic radius. In practice, the energy slope S_f is often replaced by the bed slope S_0. The parameters of the partially filled flow in a circular cross section are shown in Fig. 2.

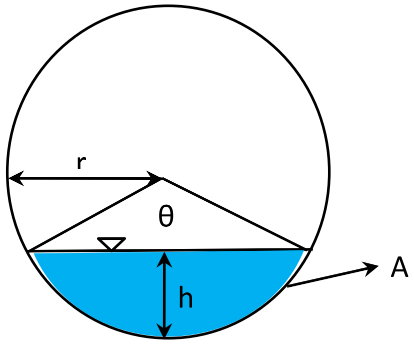

Figure 2. Partially filled flow in a circular cross section

Three experiments were conducted with different flow rates. All experiments were carried out over a smooth bed in turbulent and subcritical flow regimes. The Reynolds number (Re) was greater than 35000 and the Froude number (Fr) less than 1. The Reynolds number and the Froude number were calculated using the equations (3) and (4), respectively:

$$Re = \frac{VD_h}{\nu} \tag{3}$$

$$Fr = \frac{V}{\sqrt{hg}} \tag{4}$$

where D_h is the hydraulic diameter and ν is the kinematic viscosity.

Furthermore, the centre-plane velocity profiles were measured using Laser-Doppler-Velocimetry (LDV) at a distance of 3200 *mm* from the pipe inlet. The measurement section was selected at this point to ensure a uniform flow condition. LDV is a widely accepted and used tool for fluid dynamical investigations of gases and liquids. It is a well-established technique that gives information about the flow velocity.
Table 1 shows the characteristics of the three experiments.

Test	I	II	III
***Q* [*l/s*]**	2.5	3.5	4.5
***h* [*mm*]**	33	40	45
Re	35333	44764	54117
Fr	0.97	0.93	0.95

Table 1. Variable parameters from the experiment

2.2 CFD Simulation

For the numerical part of this study, the geometry consisted of the inlet tank and the pipe. The geometry should be discretized in meshes, in which the equations will be calculated. Here, a Cutcell mesh is compared to a mesh of tetrahedral elements. Using tetrahedral elements has the advantage that the mesh of the complex domain, consisting of cubes and cylinders, can be generated more easily. However, the same element size results in much more grid cells in a tetrahedral mesh compared to the Cutcell mesh. On the other hand, a Cutcell mesh can capture the interface between air and water more accurately. The CutCell mesher converts a volume mesh into a predominantly Cartesian mesh, which consists of mostly hexahedral elements with faces that are aligned with the coordinates axes. Fig. 3 shows the Tetrahedral and the Cutcell mesh. The mesh should be fine enough to capture the features of the flow. In general, the finer the mesh the more accurate are the results, but also the longer is the calculation time. Therefore, it is important to find a balance between the number of elements and the accuracy of the results. The refinement of the mesh should be done until the solutions from two meshes do not differ significantly. To obtain a sufficient mesh resolution at the wall, the values of y^+ are observed. Ansys Fluent user's guide (2016) suggests that the standard wall function should exhibit a y^+ value between 30 and 300, whereby a value close to the lower bound is most desirable. The here investigated meshes exhibit y^+ values between 40 and 195 near the wall in the measurement section. In addition, the skewness and the orthogonal quality of the mesh are considered to check its quality. The maximum skewness should be less than 0.97 and the minimum orthogonal quality more than 0.01 (Ansys, 2016). Table 2 summarizes the three investigated meshes in this study.

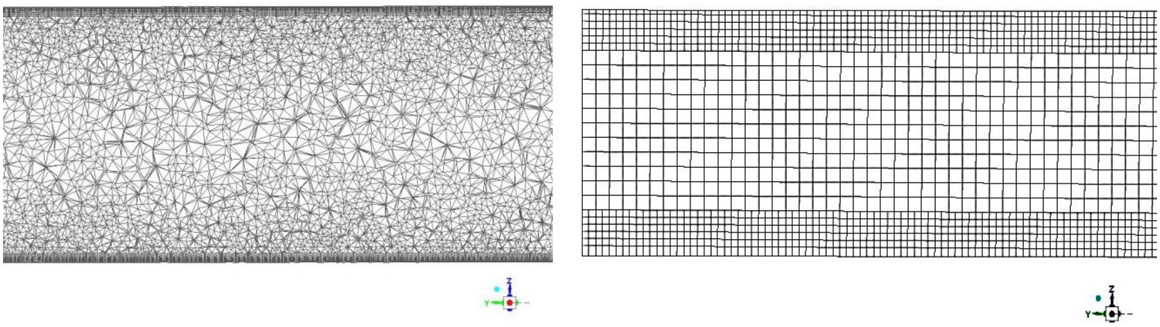

Figure 3. Tetrahedral mesh (left) and Cutcell mesh (right) in the middle cut plane

Mesh	Elements	Maximum skewness	Minimum orthogonal quality
Tetrahedral	2890604	0.9	0.09
fine Cutcell	1426914	0.95	0.28
coarse Cutcell	376097	0.78	0.33

Table 2: Summary of investigated meshes

The mostly used and validated multiphase solver Volume of Fluid (VOF) is used to simulate the interaction of air and water in partially filled pipes. VOF is able to describe the deformation of the free surface very well and therefore has been used successfully by a lot of researchers (Schmitt et al., 1999; He et al., 2004; Schaffner et al., 2004; Kirchheim et al., 2005; Bardiaux et al., 2006; Bardiaux et al., 2008; Chen et al., 2013; Mohsin and Kaushal, 2016; Regueiro-Picallo et al., 2016; Gandhi et al., 2010). VOF can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. In each computational cell, the volume fractions of all phases sum to unity. The volume-averaged values in each cell are either representative for one phase or for a mixture of the phases. The mixture density (ρ_{mix}) concept in VOF for a domain containing water and air is as follows:

$\rho_{mix} = \rho_{air}, \alpha_{air} = 1, \alpha_{wat} = 0$ (The cell is empty of water)

$\rho_{mix} = \rho_{wat}, \alpha_{wat} = 1, \alpha_{air} = 0$ (The cell is empty of air)

The density of the gas-liquid mixture fluid (at the interface of water and air) is:

$$\rho_{mix} = \alpha_{air}\rho_{air} + \alpha_{wat}\rho_{wat} \quad (5)$$

where ρ_{air} and ρ_{wat} are the density of the air and water, respectively. α_{air} and α_{wat} are the volume fraction of air and water, respectively.

The continuity equation for the volume fraction of the secondary phase (here air) neglecting the mass source has the following form:

$$\frac{\partial}{\partial t}(\alpha_{air}\rho_{air}) + \nabla(\alpha_{air}\rho_{air}V_{air}) = 0 \quad (6)$$

The volume fraction equation will not be solved for the primary phase (here water). The primary phase volume fraction is computed based on $\alpha_{air} + \alpha_{wat} = 1$.

The momentum depends on the volume fractions of the phases through ρ and μ and is calculated as follows:

$$\rho\frac{\partial V}{\partial t} + \nabla.(\rho VV) = \nabla.[\mu(\nabla V + \nabla V^T)] - \nabla p + \rho g \quad (7)$$

where ρ is the fluid density, V is the fluid velocity, μ is the dynamic viscosity, p is the pressure and ρg is the gravitational body force. Only one set of momentum equations will be solved for the mixture fluid and the resulting velocity is shared among the two phases. Turbulence is also the same as in single phase flows. A single set of transport equations will be solved and the turbulence variables (here *k* and *ε*) are shared by the phases throughout the field.

The choice of the turbulence model is dependent on several factors such as the physics of the flow, the available computational resources and time for the simulation, the required accuracy, etc. Therefore, different turbulence models have been investigated. The Reynolds-averaged Navier-Stokes (RANS) approach reduces the required computational effort and resources and is widely adopted for practical engineering applications. The *k-ε* model of the RANS group is the most widely used and validated turbulence model (Stovin et al., 2002). The model is based on the Boussinesq assumption that Reynolds Stresses can be linked to the mean rates of fluid deformation. The *k–ε* model uses two quantities, the turbulent kinetic energy, *k*, and its rate of dissipation per unit mass, *ε*, to calculate the eddy viscosity. There are three variants of the *k-ε* model available in Fluent: standard, realizable and RNG, which require more computational effort respectively. They differ in:

- the method of calculating turbulent viscosity
- the turbulent Prandtl numbers governing the turbulent diffusion of *k* and *ε*
- the generation and destruction terms in the *ε* equation

The theory behind these models is well descried in Ansys Fluent Theory Guide (2016). In this study, the three variants are investigated and compared with each other. In all cases the standard wall function is selected. Table 3 summarizes different CFD models used in this study. Other turbulence models available in the Fluent package like the Reynold stress model (RSM) could give more accurate predictions for complex flows and particularly for the effects of the secondary flow. However, they are not as well validated as the k-ε model. Furthermore, in circular channels where the secondary flow is more of a result of the geometrical shape than of the turbulence, the k-ε model could more accurately predict the flow than the RS model (Knight et al., 2005).

Model	**Mesh**	**Turbulence model**
1	Tetrahedral	*k-ε* standard
2	fine Cutcell	*k-ε* standard
3	coarse Cutcell	*k-ε* standard
4	coarse Cutcell	*k-ε* RNG
5	coarse Cutcell	*k-ε* realizable

Table 3. Summary of investigated CFD models

The boundary conditions and the solution methods are set based on experiences and suggestions of the ANSYS User's Guide. At the inlet, a mass flow inlet is selected for the liquid phase with mass flow rates from the experimental data and zero for the gas phase. The turbulent intensity and viscosity ratio have been set to remain at 5% and 10, respectively at the inlet as well as at the outlet. The open top of the inlet tank and the outlet of the pipe have a pressure outlet boundary condition. The backflow value of the volume fraction is 1 for the gas phase. For the wall boundary conditions, the no-slip wall condition is used. Fig. 4 shows the computational domain and the boundary conditions.

Based on the implicit scheme for VOF, the following solution methods are selected: The PISO algorithm for the pressure-velocity coupling, the PRESTO discretization for pressure and Second Order Upwind for momentum. The interface between fluids is represented with the Modified HRIC scheme, because when calculating implicitly, all other discretization schemes will normally lead to numerical diffusion. The First Order Upwind scheme is selected for turbulent kinetic energy and turbulent dissipation rate.

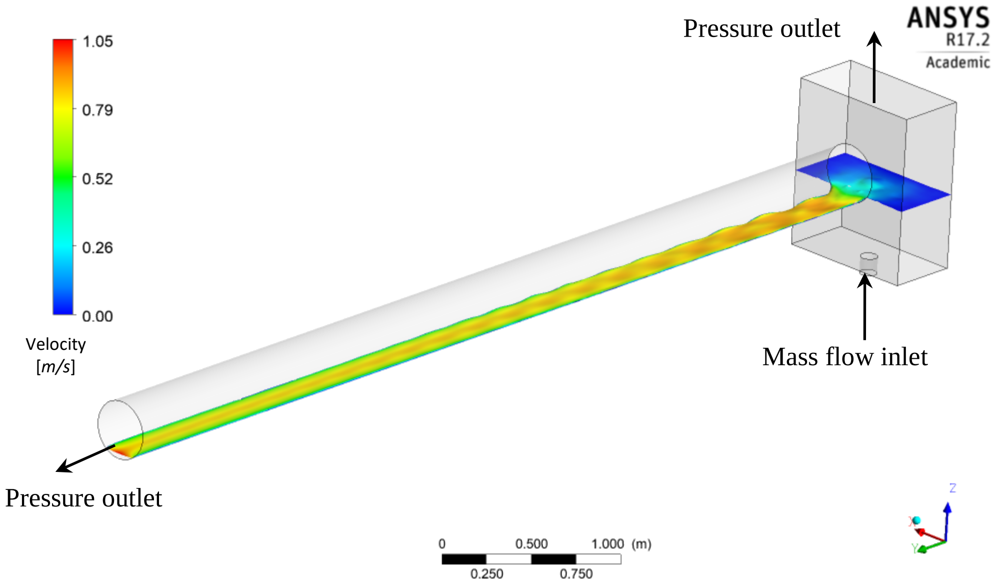

Figure 4. Sketch of the computational domain and the boundary conditions

3 Results and Discussion

3.1 Validation

First, the flow in the pipe with an inlet flow rate of 3.5 *l/s* is simulated. The CFD model centre-plane velocity profiles are compared with the LDV measurements at the middle-section of the pipe in a position of 3200 mm from the inlet. The comparison between the results of the first two models (Model 1 and 2) and the LDV measurements shows that the Cutcell mesh gives a better estimation of the centre-plane velocity profile. However, the mesh of tetrahedral cells was able to predict the values of the velocity in the near wall region. In the next step, the Cutcell mesh is coarsened (Model 3) and its velocity profile is compared to the model with a finer mesh (Model 2). Since there is no significant difference between the results and a coarser mesh leads to faster simulations, the coarser Cutcell mesh is selected as the best mesh. To choose the best turbulence model, three variants of the k-ε model are compared (Model 3, 4 and 5). The RNG model (Model 4) estimated the velocity profile with least error. Fig. 5 shows the comparison of the centre-plane velocity profile from the five CFD models and the LDV measurements.

Second, the Cutcell mesh and the RNG k-ε model (Model 4) are used to simulate the two other tests with flow rates of 2.5 and 4.5 *l/s*. Fig. 6 shows the comparison of the centre-plane velocity profiles simulated with Model 4 and LDV measurements of these two tests. As seen, the model was able to reasonably predict the velocity profile for different flow rates or rather different filling ratios in a circular pipe.

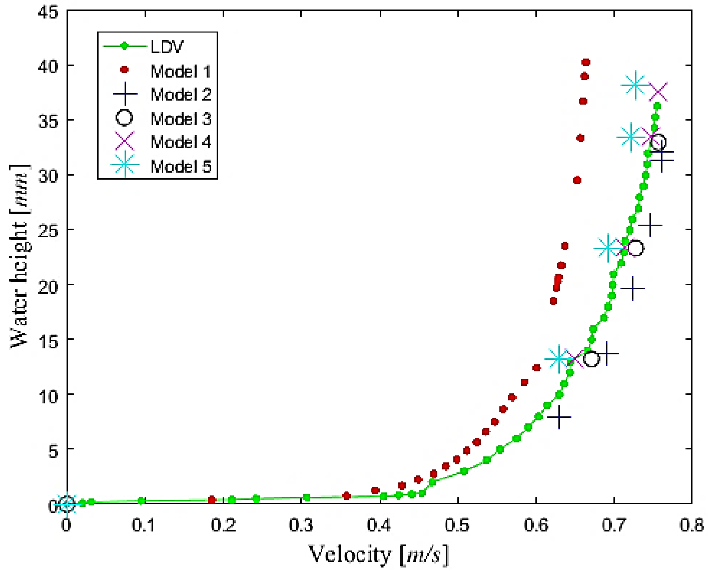

Figure 5. Experimental and numerical comparison of the centre-plane velocity profiles for an inlet flow rate of 3.5 *l/s* with different meshes and turbulence models

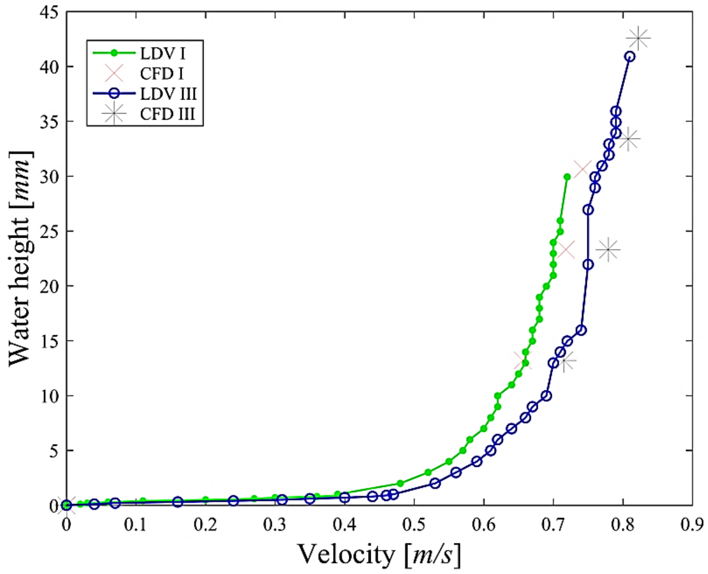

Figure 6. Experimental and numerical comparison of the centre-plane velocity profiles for Test I and III with an inlet flow rate of 2.5 *l/s* and 4.5 *l/s*, respectively using Model 4

Furthermore, the average velocity and shear stress over the wetted perimeter are used in order to validate the best model (Model 4) for three different flow rates. Table 4 shows the values of the experimental and numerical velocity and shear stress as well as the relative error between these two data sets. It is noticeable that the CFD model overestimates the values of the velocity. This could be explained by the fact that the k-ε model delivers better results for near free surface flows rather than flows close to the wall. Therefore, the small velocity values near the wall may not be very well estimated, which causes an overestimation of the mean velocity. Another alternative could be the Shear Stress Transport (SST) turbulence model which is a combination of the k-ε and k-ω turbulence models. The k-ω model is better suited for near wall regions. It was not possible within the frame of this work to test this model. On the other hand, this model underestimates the values of the shear stress. The reason could be that the experimental shear stress is an estimation of the average shear stress in the pipe. However, the value of the shear stress from the CFD model is the mean shear stress on a selected cross section. By the way, the relative error between experimental and numerical values is not more than 10% (except for V in test I).

Test	V_{EXP} **[m/s]**	V_{CFD} **[m/s]**	τ_{EXP} **[Pa]**	τ_{CFD} **[Pa]**
I	0.55	0.61	0.76	0.70
II	0.58	0.63	0.91	0.84
III	0.63	0.68	1.02	0.98

	Error V [%]	Error τ [%]
I	10.90	-7.89
II	8.62	-7.69
III	7.94	-3.92

Table 4. Comparison of the values of mean velocity and shear stress from experiment (EXP) and CFD

3.2 Velocity

As the bed roughness plays a major role in sediment transport in sewers, the validated CFD model is used to investigate its influence on the flow behaviour. Three different values for the roughness height are simulated and compared together for the same discharge (3.5 *l/s*) and bed slope (0.37 %); Smooth bed (k_s = 0), medium coarse bed (k_s = 0.5 *mm*) and rough bed (k_s = 1 *mm*) which represent plastic, concrete and stoneware as the sewer bed material, respectively. Fig. 7 shows the velocity contours on a wetted cross section with different roughness heights. The average velocity on the smooth bed is 0.63 *m/s*, on medium coarse bed 0.54 *m/s* and on the rough bed 0.49 *m/s*.

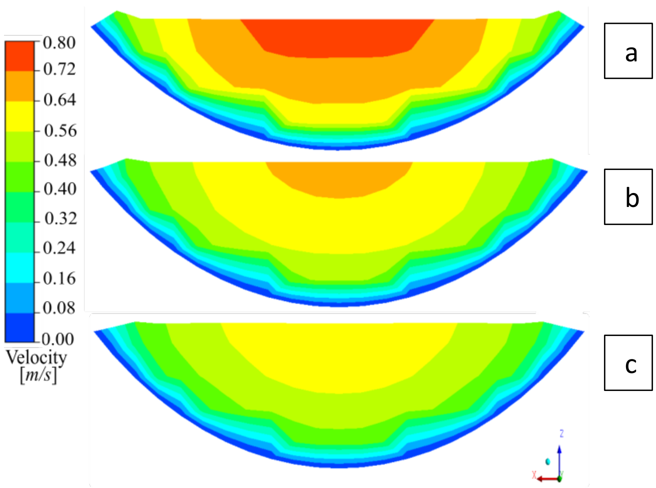

Figure 7. Longitudinal velocity contours on wetted cross section for (a) k_s = 0, (b) k_s = 0.5 *mm* and (c) k_s = 1 *mm*

In addition, the centre-plane velocity profiles are plotted against the water height. Fig. 8 shows the comparison for three different bed roughness heights. In the near wall region, the difference is not significant. However, the difference becomes greater with increasing water height. In the free-surface region, it is easy to see that the bed roughness has a great influence on the velocity profiles in the pipe. From the Figs. 7 and 8, it can be concluded that the velocity decreases while increasing the bed roughness, especially in the free surface region.

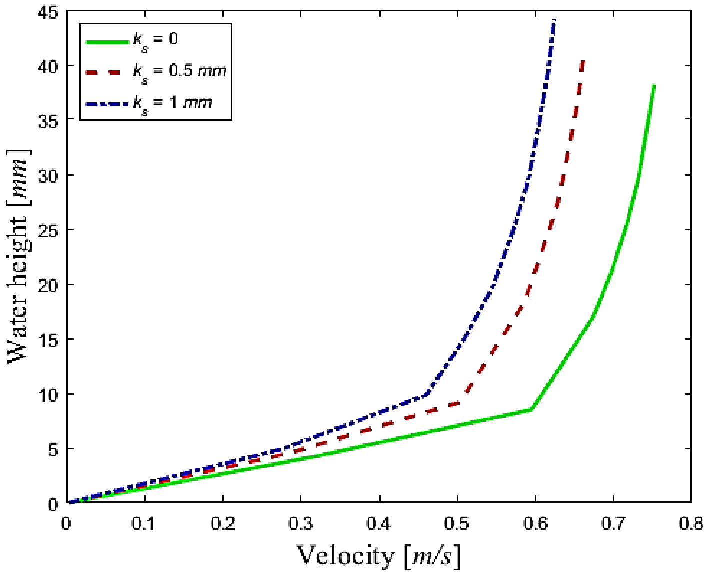

Figure 8. Numerical comparison of centre-plane velocity profiles for an inlet flow rate of 3.5 *l/s* and bed roughness of 0, 0.5 and 1 *mm*

3.3 Shear Stress

Simulated boundary shear stress distributions are plotted as a function of the lateral distance from the centre-plane for three different flow rates (Fig. 9) and three different roughness heights (Fig. 10). The *x*-axis represents the distance along the wetted perimeter of the pipe. It is visible in the figures and also concluded by (Knight and Sterling, 2000) that the shear stresses on the bottom of the channel are larger than the shear stresses on the walls. As shown in Fig. 9, increasing the flow rate leads to higher values for the shear stress. It can be observed that the distribution of the shear stress becomes more uniform by increasing the filling ratios. This is due the fact that, when the pipe is running full, the shear stress on any section of the boundary is constant. For higher flow depth ratios, the shear stress reaches its maximum value (1.2 *Pa* for 4.5 *l/s*) in the centre of the pipe; while for lower flow depth ratios, this maximum (1.05 *Pa* for 3.5 *l/s* and 0.83 *Pa* for 2.5 *l/s*) is reached in two symmetrical points of the wetted perimeter. The existence of two maxima on either side of the centreline in an open channel flow indicates the presence of secondary flows, which draws high momentum fluid away from the channel centre towards the corners. However, the strength of the secondary currents decreases with an increase in depth (Hoohlo, 1994). That’s why for higher flow rates, the maximum occurs in the centre. In addition, it is to be seen that the distribution becomes less uniform by increasing the roughness height. The shear stress on the bottom increases up to 20% while increasing the bottom roughness height by 1 *mm*.

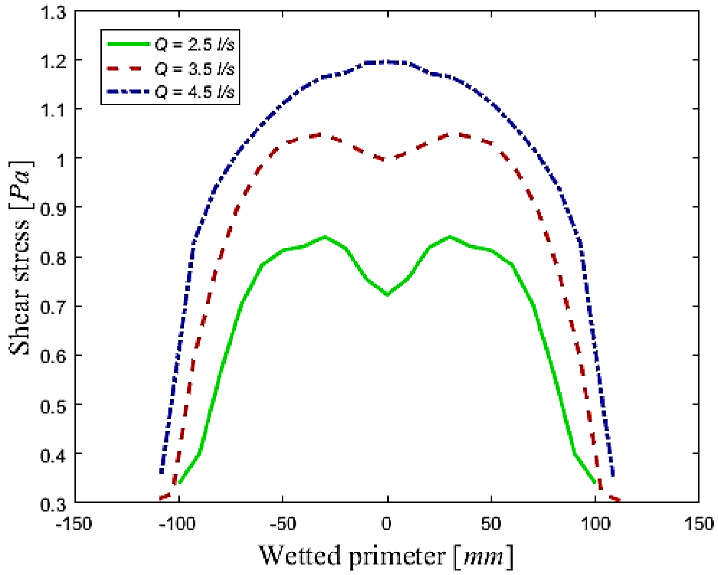

Figure 9. Boundary shear stress distribution in a cross section for different flow rates under smooth bed conditions

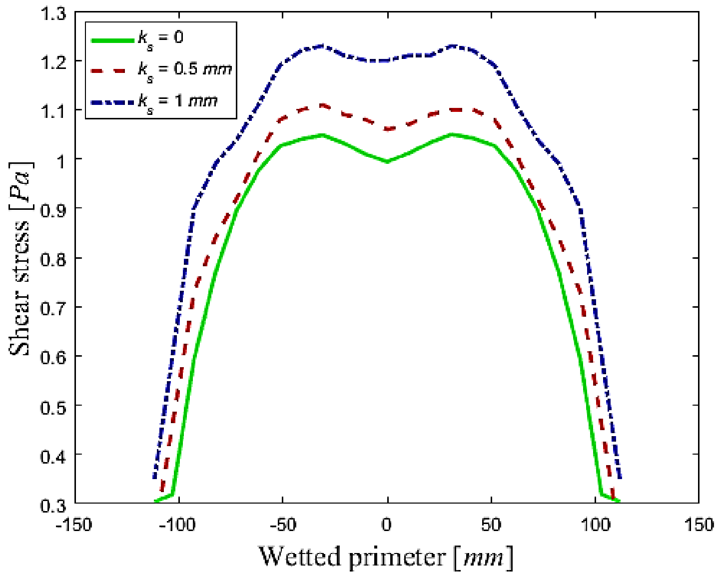

Figure 10. Boundary shear stress distribution in a cross section for different roughness heights with a flow rate of 3.5 *l/s*

4 Conclusions and Future Works

The aim of this study was to investigate various mesh approaches and turbulence models for the simulation of the flow in a partially filled pipe. A mesh of tetrahedral elements and two Cutcell meshes were compared. Additionally, three variants of the k-ε model were studied. The centre-plane velocity profiles obtained with different CFD models were compared to LDV measurements. The best results were obtained from a Cutcell mesh combined with the RNG k-ε turbulence model. Furthermore, the mean velocity and shear stress of experimental results were compared to the values of the best CFD model. The relative error was under 10%. The validated CFD model was used to analyse the influence of the bed roughness on the velocity distribution in pipes. The velocity decreases while increasing the bed roughness, particularly in the free surface region. In addition, the distribution of the boundary shear stress was investigated under different flow rates and bed roughness heights. The distribution becomes more uniform as the flow rate increases and less uniform while increasing the bed roughness. The shear stress over the rough bed is larger than that over the smooth bed. The results showed that the CFD model could reasonably simulate the flow in open channels under different boundary conditions. This paper can be seen as a preliminary study for working on more complicated problems. The results of this study have been used for investigations on the modelling of sediment transport in sewer systems using a coupled method of CFD and Discrete Element Method (DEM). The first results of this study are presented in (Alihosseini and Thamsen 2018).

5 Acknowledgments

This work has been funded and supported by Climate-KIC. Climate-KIC is supported by the European Institute of Innovation and Technology (EIT), a body of the European Union.

References

Alihosseini, M.; Thamsen, P.U.: Experimental and numerical investigation of sediment transport in sewers. *5th Joint US-European Fluids Engineering Division Summer Meeting (FEDSM 2018 ASME)*, Quebec, Canada (2018).

Ansys: *Ansys Fluent User's Guide, Release 17.2.* Southpointe, Canonsburg, ANSYS, Inc. (2016).

Ansys: *Ansys Fluent Theory Guide*, *Release 17.2.* Southpointe, Canonsburg, ANSYS, Inc. (2016).

Bardiaux, J.B.; Bonakdari, H.; Larrarte, F.; Mose, R.; Vazaquez, J.: Velocity Profiles Through a Sewer Channel: Using CFD to Obtain Velocity Fields. *Dresdner Wasserbauliche Mitteilungen*, 32, (2006), 327-335.

Bardiaux, J.B.; Mose, R.; Vazquez, J.; Wertel, J.: Two Turbulent Flow 3D-Modelings to Improve Sewer Net Instrumentation. *11th International Conference on Urban Drainage*, Edingburgh, Scotland, UK, (2008).

Bares, V.; Pollert, J.; Srnicek, P.: Flow Pattern Visualization of Combined Sewer Overflow. *5th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering*, Zürich, Switzerland, (2006), 25-28.

Berlamont, J.E.; Trouw, K.; Luyckx, G.: Shear Stress Distribution in Partially Filled Pipes. *Journal of Hydraulic Engineering*, 129, (2003), 697-705.

Bonakdari, H.; Ebtehaj, I.; Azimi, J.: Numerical Analysis of Sediment Transport in Sewer Pipe. *International Journal of Engineering (IJE)*, 28, (2015), 1564-1570.

Chen, Z.; Han, S.; Zhou, F.; Wang, K.; A CFD Modeling Approach for Municipal Sewer System Design Optimization to Minimize Emissions into Receiving Water Body. *Water Resources Management*, 27, (2013), 2053-2069.

Dufresne, M.; Vazquez, J.; Terfous; A.; Ghenaim, A.; Poulet, J.B. Experimental Investigation and CFD Modelling of Flow, Sedimentation, and Solids Separation in a Combined Sewer Detention Tank. *Computers & Fluids*, 38, (2009), 1042-1049.

Gandhi, B.K.; Verma, H.K.; Abraham, B.: Investigation of Flow Profile in Open Channels Using CFD. *IGHM.* IIT Roorkee, India, (2010), 243-251.

He, C.; Marsalek, J.; Rochfort, Q.: Numerical Modelling of Enhancing Suspended Solids Removal in a CSO Facility. *Water Qual. Res. J. Canada*, 39, (2004), 457-465.

Hoohlo, C.: *A Numerical and Experiemntal Study of Open-Channel Flow in a Pipe of Circular Cross-Section with a Flat Bed.* Doctoral Thesis: Newcastel University Library, (1994).

Jarman, D.S.; Faram, M.G.; Butler, D.; Tabor, G.; Stovin,V.R.; Burt, D.; Throp, E.: Computational Fluid Dynamics as a Tool for Urban Drainage System Analysis: A Review of Applications and Best Practice. *11th International Conference on Urban Drainage*, Edinburgh, Scotland, UK, (2008).

Jiang, Y.; Bin, L.; Chen, J.: Analysis of the Velocity Distribution in Partially-Filled Circular Pipe Employing the Principle of Maximum Entropy. *PloS ONE*, 11, (2016), 1-17.

Kirchheim, N.; Schaffner, J.; Oberlack, M.: Parameter study of a flush wave using numerical modeling. *10th Internationl Conference on Urban Drainage*, Copenhagen, Denmark, (2005).

Knight, D.W.; Sterling, M.: Boundary Shear in Circular Pipes Running Partially Full. *Journal of Hydraulic Engineering*, 126, (2000), 263-275.

Knight, D.W.; Wight, N.G.; Morvan, H.P.: *Guidelines for applying commercial CFD software to open channel flow*. http://www.nottingham.ac.uk/cfd/ocf/guidelines.pdf, (2005), 1-32.

Mohsin, M.; Kaushal, D.R.: 3D CFD Validation of Invert Trap Efficiency for Sewer Solid Management using VOF Model. *Water Science and Engineering*, 9, (2016), 106-114.

Regueiro-Picallo, M.; Naves, J.; Anta, J.; Puertas, J.; Suarez, J.: Experimental and Numerical Analysis of Egg-Shaped Sewer Pipes Flow Performance. *MDPI - Water*, 8, (2016), 1-9.

Schaffner, J.; Oberlack, M.; Kirchheim, N.: The Application of Numerical Modeling (3-D) for the Calculation of Flush Waves in Sewer Channels. *6th International Conference on Urban Drainage Modelling (UDM'04)*, Dresden, Germany, (2004), 1-8.

Schmitt, F.; Milisic, V.; Bertrand-Krajewski, J.-L.; Laplace, D., Chebbo, G.: Numerical Modelling of Bed Load Sediment Traps in Sewer Systems by Density Currents. *Water Science and Technology*, 39, (1999), 153-160.

Stovin, V.R.; Grimm, J.P.; Buxton, A.P.; Tait, S.J.: Parametric Studies on CFD Models of Sewerage Structures. *Proc. 9th International Conference on Urban Drainage*, Portland, Oregon, USA, (2002), 1-15.

Yan, H.; Lipeme Kouyi, G.; Gonzalez-Merchan, C.; Becouze-Lareure, C.; Sebastian, C.; Barraud, S.; Bertrand-Krajewski, J.-L.: Computational Fluid Dynamics Modelling of Flow and Particulate Contaminants Sedimentation in an Urban Stormwater Detention and Settling Basin. *Environmental Science and Pollution Research*, 21, (2014), 5347-5356.

Address: Department of Fluid Mechanics, Faculty of Mechanical Engineering, Berlin University of Technology. Straße des 17. Juni 135, 10623 Berlin, Germany
email: m.alihosseini@tu-berlin.de