Modelling Flow and Sediment Transport in Urban Sewer Systems Using Computational Fluid Dynamics: Case of Tororo Municipality, Eastern Uganda

FILBERT AKABWAI[1], VERDIANA GRACE MASANJA[2], JOSEPH DDUMBA LWANYAGA[1], TWAIBU SEMWOGERERE[2], ASAPH KEIKARA MUHUMUZA[1*]

[1]Faculties of Science and Education, Busitema University,
P O Box 236 Tororo,
UGANDA

[2]Faculties of Engineering and Technology, Busitema University,
P O Box 236 Tororo,
UGANDA

[3]Nelson Mandela African Institution of Science and Technology (NM-AIST),
School of Computational Sciences and Engineering (CoCSE) Arusha,
TANZANIA

[*]*Corresponding author*

Abstract: The optimal and sustainable performance of sewer networks is of utmost importance in urban drainage systems. Efficient operation of urban sewer networks is increasingly challenged by sediment deposition, climate variability and rapid urbanization. This study develops and validates a Computational Fluid Dynamics (CFD) model to analyze flow dynamics and sediment transport behavior in selected sewer sections of Tororo Municipality, Eastern Uganda. A two-dimensional (2D) multiphase model was implemented using OpenFOAM's interFoam solver with the SST k–ω turbulence closure. The Reynolds-Averaged Navier–Stokes (RANS) equations were solved under incompressible flow assumptions, coupled with Volume of Fluid (VOF) formulation for phase tracking. This study analysed the hydro mechanical properties of flow in sewer pipes at different flow speed and their effect on sediment flow capacity by use of two dimensional numerical simulation of flow field using Computation Fluid Dynamics (CFD). Mesh independence testing, time-step sensitivity analysis, and Courant–Friedrichs–Lewy (CFL) stability verification were performed to ensure numerical robustness. The study proposes revised self-cleansing design criteria based on minimum velocity and minimum bed shear stress thresholds. A conservative minimum bed shear stress of 2 N/m² is recommended for cohesive sediments with roughness height ks = 1.2 mm. Results show that optimal sediment transport occurs near inlet velocities of 1.0 m/s, balancing entrainment efficiency and excessive turbulence. The study highlighted the predictive capacity of CFD for sewer optimization while explicitly quantifying limitations associated with 2D modeling relative to three-dimensional (3D) flow behavior.

Key-words: Computational Fluid Dynamics (CFD), Sewers flows, Sediment transport, Mathematical model.

Received: May 26, 2025. Revised: August 2, 2025. Accepted: September 5, 2025. Published: April 7, 2026.

1 Introduction

Urban sewer systems are fundamental to public health protection, environmental sustainability, and resilient urban infrastructure. Their hydraulic performance determines the reliability of wastewater conveyance and treatment efficiency. However, one of the most persistent operational challenges in gravity-driven sewer networks is sediment accumulation within conduits (Ashley et al., 2004; Butler et al., 2003). Sediment deposition reduces effective hydraulic capacity, increases blockage frequency, accelerates structural deterioration, and elevates maintenance costs. In rapidly urbanizing environments, these challenges are intensified by land use change, climate variability, and evolving wastewater characteristics (Marleni et al., 2015; Murali et al., 2019).

Sediment transport in sewer systems is governed by interactions among flow velocity, turbulence structures, sediment size distribution, pipe geometry, surface roughness, and wastewater composition (Banasiak & Tait, 2008; Butler et al., 2003; Ghani & Azamathulla, 2011; Ota and Perrusquia, 2013). Unlike controlled laboratory channels, real sewer systems operate under highly variable inflow rates and fluctuating solids concentrations (He et al., 2004; Roulund et al., 2005; Schaffner et al., 2004). These dynamics create multiphase flow conditions where water, air, and sediments coexist and interact, making accurate prediction of entrainment and deposition processes complex (Dufresne et al., 2009; Murali et al., 2020).

Traditional sewer design methodologies typically rely on empirical self-cleansing velocity criteria or simplified shear stress relationships derived from sediment transport theory (May et al., 1996; Ackers et al., 1996; Almedeij et al. 2012). While these approaches provide practical guidance, they may not fully account for turbulence anisotropy, transient flow conditions, and phase interactions in partially filled conduits (Ebtehaj et al., 2014; Montes et al., 2019). Consequently, there is growing recognition that improved predictive tools are required to support optimized sewer design and rehabilitation.

Advances in Computational Fluid Dynamics (CFD) have enabled detailed numerical analysis of fluid flow behavior by solving the governing conservation equations for mass and momentum (Anderson & Wendt, 1995; Versteeg & Malalasekera, 2007; Noah et al., 2020). CFD techniques allow visualization of velocity fields, pressure gradients, turbulence intensity, and wall shear stress distributions within complex hydraulic systems (Baseka & Masanja, 2022). When combined with multiphase modelling approaches such as the Volume of Fluid (VOF) method (Hirt & Nichols, 1981), CFD can simulate free-surface and stratified flow conditions typical of sewer pipes (Bayon et al., 2018; Sattar et al., 2017).

Recent studies have applied CFD to investigate sediment transport and morphodynamic processes in sewer and drainage systems (Chen et al., 2013; Ebtehaj et al., 2015; Rinas et al., 2020). For example, Murali et al. (2020) developed a free-surface sediment transport model incorporating bed morphology evolution, while Song et al. (2018) examined sedimentation behavior in urban sewer conduits under varying hydraulic conditions. These investigations demonstrate the capability of CFD to provide mechanistic insights beyond empirical design equations.

Despite these advancements, sewer design practices in many developing regions remain largely dependent on conventional hydraulic calculations. In Uganda, particularly within Tororo Municipality, sewer infrastructure faces operational challenges associated with sediment accumulation, aging pipe materials, and reduced wastewater inflows linked to changing water consumption patterns (MWE, 2019; Marleni et al., 2015). Reduced dry-weather flows combined with intermittent peak discharges create hydraulic conditions favorable for deposition, thereby necessitating improved modelling-based evaluation.

Although previous research has investigated sewer hydraulics using CFD (Dufresne et al., 2009; Chen et al., 2013), limited work has contextualized such modelling frameworks within East African urban settings. Moreover, many studies emphasize either single-phase flow or simplified turbulence representations, without fully integrating multiphase interface tracking and validated turbulence closure models appropriate for near-wall sediment entrainment processes (Menter, 1994; Wilcox, 1998).

The present study therefore develops a two-dimensional multiphase CFD model to simulate flow and sediment transport in a representative sewer section of Tororo Municipality. The model solves the incompressible Reynolds-Averaged Navier–Stokes (RANS) equations using the finite volume method (Versteeg & Malalasekera, 2007) and applies the SST k–ω turbulence model for improved near-wall prediction (Menter, 1994).

Air–water phase interaction is captured using the VOF approach (Hirt & Nichols, 1981), implemented within the OpenFOAM computational framework (Greenshields, 2019). To enhance methodological rigor, mesh independence verification, time-step sensitivity analysis, and Courant–Friedrichs–Lewy (CFL) stability assessment (Trivellato & Castelli, 2014) are explicitly performed.

The objectives of this study are: To develop a CFD-based framework for analyzing multiphase sewer flow under representative operating conditions; Quantify velocity distribution, turbulence characteristics, and wall shear stress relevant to sediment entrainment; Evaluate minimum self-cleansing velocity and bed shear stress criteria for cohesive sediments (Butler et al., 2003; Ebtehaj et al., 2014); Assess the applicability and limitations of two-dimensional modelling relative to three-dimensional approaches (Liu & García, 2008; Sattar et al., 2017); and Provide context-specific design recommendations for sewer optimization in Tororo Municipality.

By integrating numerical modelling with sediment transport principles, this work contributes to improved sewer design and infrastructure resilience in resource-constrained urban environments (Kirchheim et al., 2005). The study demonstrates how CFD can support evidence-based decision-making for pipe diameter selection, slope configuration, and operational velocity thresholds, thereby complementing traditional empirical design methodologies.

Methods

The mathematical model formulation and algorithm design was done using CFD analysis. Numerical simulations were carried out using the open source CFD software Open FOAM. Data for model variables were also collected from National Water and Sewerage Cooperation (NWSC) documentation and some relatively small samples of sewer diameters and velocities were collected from the municipal offices. The CFD computations involved carrying out structured and unstructured computational mesh discretization for the Nervier-Stokes equations and the VOF using cross-sectional design of sewers.

The non-deposition sediment transport in sewer systems has two to three criteria forms, the suspended load, the bed load transport and cohesive sediment erosion (Ebtehaj et al., 2014; Butler et al., 2003). The use of the non-deposited sediment transport criterion, data such as mean sediment size, relative density of sediment, sediment volumetric concentration, and depth of flow are required to propose an equation for the rate of the sediment transport as bed load or suspended load. In addition, the state of transport should be specified before the existing self-cleansing equations can be used (Ebtehaj et al., 2014). In studies conducted by May et al. (1996), if $U_s > 0.75W_s$, where U_s is the shear velocity and W_s is the particle settling velocity, the sediment transport is in the form of suspended load, otherwise the transport mode will be in the form of bed load, with U_s defined as in Equation 1

$$U_s = V\sqrt{\frac{\lambda}{8}} \tag{1}$$

Where V is the self-cleansing velocity and λ is defined as in Equation (2)

$$\lambda = 8gn^2/R^{1/3} \tag{2}$$

Where λ is the friction coefficient of the bed, g is the acceleration due to gravity, n is the Manning's roughness coefficient and R is the hydraulic radius.

Butler et al. (2003) and May et al. (1996), proposed the equations 3 and 4 for a bed-load transport:

$$C_v = 3.03 \times 10^{-2}\left(\frac{D^2}{A}\right)\left(\frac{d}{D}\right)^{3/5}\left(1-\frac{U_t}{V}\right)^4\left[\frac{V^2}{g(S_g-1)D}\right]^{3/2} \tag{3}$$

$$U_t = 0.125\sqrt{g(S_g-1)d}\left(\frac{y}{d}\right)^{0.47} \tag{4}$$

Where C_v is the volumetric sediment concentration (discharge rate of sediment or discharge rate of water); D is the pipe diameter, y is the depth of flow, d is the mean sediment size, g is acceleration due to gravity, S_g is sediment specific gravity, A is cross sectional area of the flow, and U_t threshold velocity required to initiate movement. According to Butler et al. (2003) and Ackers et al. (1996), Equation 3 is the best relationship proposed for the sediment transport in a bed load state. It was controlled by the data of 332 different tests and offered the best results. The conditions of the tests were: pipe diameter of 77 to 450 mm; sediment size of 160 to 8300 μm; relative depth of flow (y/D) of 0.16 to 1; flow velocity of

0.24 to 1.5 m/s and the sediment volumetric concentration of 2.13×10^{-4} to 2.11×10^{-3} (Ebtehaj et al., 2014).

For bed Erosion criteria, the effect of cohesion on the shear stress at the threshold of movement is specifically allowed for cohesive sediment erosion. It is recommended that the design flow conditions needed to erode cohesive particles from a deposited bed should have a minimum value of shear stress of 2 Nm^{-2} on a flat bed with a Colebrook-White roughness value of $k_b = 1.2$ mm (Butler et al., 2003). The required full-bore velocity (v_f) is given by Equation (5)

$$v_f = \left(\frac{8\tau b}{\rho\lambda_b}\right)^{\frac{1}{2}} \qquad (5)$$

Where τ_b is the critical bed shear stress and λ_b is the friction factor corresponding to the sediment bed. In many cases, the roughness of the deposited bed will be much higher resulting in bed shear stresses that are higher, making this approach conservative.

In this study, a self cleansing design concept of the sewer as adopted by Ebtehaj et al. (2014) is applied for optimal operation of the sewer system with reduced sediment deposits. The self-cleansing process in the wastewater system must establish a balance between the amount of sediment and the rate of erosion during the sediment transport and, in a specific period of time, minimize the combined costs of construction, operation, and maintenance of the system (Ebtehaj et al., 2014; Butler et al., 2003). The most important aspect of these requirements is that if a minimum amount of sediment allows the design of the wastewater system to become more economical, there is no need for the sewer conduits to stay completely free of deposits. May et al. (1996) showed that the presence of a deposited bed allows the flow to enhance the sediment transport capacity in a form of a bed-load. The flow leaves a greater impact on the sediment transport while compensating for the loss of velocity caused by the roughness of the bed surface. Finally, the increase in depth (and width) of the deposited bed and the increased capacity of sediment transport may both be in balance with the inlet sediment load preventing further sedimentation. Therefore, the self-cleansing design concept for non-cohesive, homogeneous sediments can be attributed to the movement of input of the sediment on the bed of the sewer network system or on the ground of non-deposition of sediments.

According to Montes et al. (2019), the limits of self-cleansing are obtained by solving the Manning equation, for circular conduits, for a specific minimum self-cleansing velocity and pipe diameter.

The self-cleansing limit for minimum shear stress is estimated by solving the shear stress equation for a shear stress value and pipe diameter, and it is therefore possible to estimate the minimum self-cleansing slope based on a specific self-cleansing criterion of minimum velocity or minimum shear stress, and considering several pipe diameters and filling ratios (Montes et al., 2019).

The fluid flow behavior is described by Reynolds Averaged Nervier-Stokes equations (Versteeg and Malalasekera, 2007; Anderson and Wendt, 1995). The conservation of mass (continuity equation) is defined by Equation (6):

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho U) = 0 \qquad (6)$$

Where ρ is the fluid density and $U = (u, v, w)$ is the average velocity vector in the x, y and z directions, respectively. For incompressible flow, the continuity equation becomes as Equation (7), indicating a divergence free velocity.

$$\nabla.\, U = 0 \qquad (7)$$

Conservation of linear momentum leads to the equations of motion.

$$\rho\, \partial u/\partial t + \rho U.\, (\nabla) = -\nabla P + \nabla.\tau + \rho g \; (8)$$

Where ρg denotes the external forces applied to the fluid, P is the Fluid pressure and τ is the viscous stress tensor characteristically defined by the fluid type.

Mesh Generation

A 2D multiphase CFD model was developed using OpenFOAM interFoam solver. The incompressible Reynolds-Averaged Navier–Stokes equations were discretized using the finite volume method. The SST k–ω turbulence model was adopted to accurately capture near-wall effects. The Volume of Fluid (VOF) method was used to model air-water interface dynamics.

Mesh Independence Verification

Table 1: The three structured mesh densities were tested.

Mesh Type	Number of Cells	Maximum Velocity
Coarse	24,000	2.48
Medium	48,000	2.52
Fine	96,000	2.53

Velocity variation between medium and fine meshes was <0.4%, confirming mesh independence. The medium mesh (48,000 cells) was adopted for simulations to balance accuracy and computational efficiency.

Time Step and CFL Stability

The time step was selected to satisfy the Courant–Friedrichs–Lewy (CFL) stability condition given by

$Co = u\Delta t/\Delta x \leq 0.5$

The Courant number was maintained below 0.5 throughout simulations. Adaptive time stepping ensured numerical stability. A time step of 0.01 s provided optimal balance between accuracy and computational cost. Simulations were conducted with adaptive time stepping, maintaining Co ≤ 0.45 throughout computation. Sensitivity tests showed that Δt = 0.005s was stable but computationally expensive, Δt = 0.01s was optimal, and Δt = 0.02s was numerical oscillations observed. Thus, finally selected Δt = 0.01s.

2 Results

Sewage flow data records indicate that, the discharge capacity for a sewer pipe of diameter 150 mm is about 43 m^3 per hour and it's about 104 m^3hr^{-1} and 191 m^3hr^{-1}, for the 225 mm and 300 mm diameter sewer pipes respectively used in the sewer network system. The results show that there exists minimum sewer slopes in the range 0.006 for the pipe diameter of 152.4mm to about 0.0008 for pipe diameter of 609.6mm. The recommended minimum slopes for different pipe diameters should be in the range 0.00430 for 150 mm diameter pipe to about 0.00077 for 600 mm diameter pipe used for sewer systems. According to information obtained, it shows that there exits Minimum (self-cleansing) velocity and Maximum velocity for each sewer pipe, that is, velocity at which, the solid particles or sediments remain in suspension without settling at the bottom of the sewer. Self-cleansing is an important aspect of sanitary sewer design and is desired to minimize the deposition of silt, sediment, and debris. Therefore, self-cleansing velocity should be maintained at least once each day during the working life of the pipe to ensure its efficiency (Duque et al., 2016). Maximum velocity must be a velocity that causes no scouring action or abrasion which depends on the material used for the construction of sewer system. According to Butler et al. (2003), for the optimal design and operation of the sanitary and storm water sewer systems with high sediment loading and at least 1% allowable deposition, the recommended minimum velocity (V_m) for different pipe diameters (D) for sewer systems are shown in Table 2.

Table 2: Minimum velocities and pipe diameters for the sewer system

D (m)	0.15	0.23	0.30	0.45
Vm (ms^{-1})	0.67	0.72	0.75	0.79
D (m)	0.45	0.60	0.75	0.90
Vm (ms^{-1})	0.79	0.90	1.06	1.22

To investigate the effect of sewer flow velocity and wall shear stress on sediment transport within sewer pipes of different sizes for optimal sewer design, the Figures 2 to 12 of the fluid flow field profiles and plots were obtained. The fully turbulent k −ω SST model using the VOF model visualizations generated in the analysis for optimal sewer flow rate; phase distribution, stratified velocity, pressure drop in a sewer pipe are presented using numerical simulation profiles and plots in terms of volume fraction (α) of the water-air phase as in Figures 2 and 3, respectively. In this case the red color represents water (α = 1). The blue color represents air (α = 0) and between the two phases is the interface.

Multi–Phase Flow

In two-phase flow, interactions between liquid and air phases, as influenced by their physical properties, flow rates, roughness, orientation of the pipe, cause the fluids to flow in various types of patterns known as flow regimes. The flow regimes in horizontal pipeline in order of increasing vapor velocity include; bubble, plug, stratified, wavy, slug, annular and spray flow or dispersed flow. Only one type of flow exists at a given point in a line at any given time. However, as flow conditions change, the flow regime may change from one type to another. In bubble flow, the liquid occupies the bulk of the cross-section and vapor flows in the form of bubbles along the top of the pipe, thus vapor and liquid velocities are approximately equal. In plug flow, as the vapor rate increases, the bubbles coalesce and alternating plugs of vapor and liquid flow along the top of the pipe with liquid remaining the continuous phase along the bottom. For the stratified flow, as the vapor rate continues to increase, the plugs become a continuous phase. Vapor flows along the top of the pipe and liquid flows along the bottom. the interface between phases is relatively smooth and the fraction occupied by each phase remains constant. As the vapor rate increases still further, the vapor moves appreciably faster than the liquid and the resulting friction at the interface forms liquid waves, hence wavy flow. In slug flow when the vapor rate reaches a certain critical value, the crests of the liquid waves touch the top of the pipe and form frothy slugs. The velocity of these slugs and that of the alternating vapor slugs is greater than the average liquid velocity. Annular flow is when, the liquid flows as an annular film of varying thickness along the wall, while the vapor flows as a high-speed core down the middle. Lastly, in spray flow or dispersed flow, when the vapor velocity in annular flow becomes high enough, all of the liquid film is torn away from the wall and is carried by the vapor as entrained droplets. This flow regime is almost completely independent of pipe orientation or direction of flow.

In this study, numerical simulation results show that most of the multi-phase fluid flow patterns are stratified flow. In this case there is a complete separation of air and water phase. This flow pattern can be formed in a limited range of relatively low flow rates where the stabilizing gravity force due to a density difference between the two liquids are dominant. By increasing the fluid flow rate, the air core breaks up in large slugs or drops. The air drops are generated due to capillary instabilities in the presence of shear. If the fluid flow rate is sufficiently large, the entire air phase breaks up into small droplets which results in an air to fluid dispersion. If the dispersion is very stable, the flow is called an emulsion.

The two-phase flow patterns in horizontal pipes are similar to vertical flows but the distribution of the phases is influenced by gravity. Thus the water moves as a consequence of the gravitational force and not because of the pressure variation. This acts to stratify the heaviest phase to the bottom of the pipe and the air to the top as shown in Figure 1

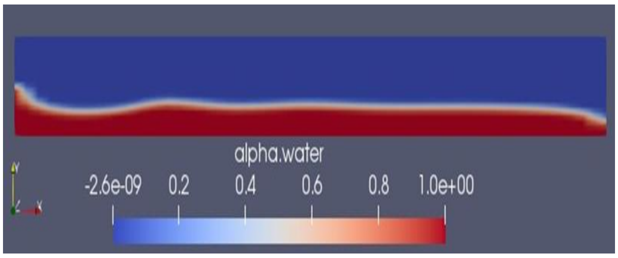

Fig. 1: Radial phase distribution inside a pipe at 0 degrees

Both phases have the same velocity set to 1.0 ms−1 in a horizontal 2D pipe of 0.30 m in diameter and 10.0 m in length. The separation of the phases occurs and the flow just continues in this state throughout the computational domain. There is no mixing between the two phases and the interface is quite smooth. The air goes to the top and the water to the bottom, due to the density difference of the phases and the influence of gravity. This kind of occurrence facilitates the optimal flow of waste water with sediments in the sewer line. That is, the velocity is high in the water phase than in the air phase and the flow agrees with no-slip condition in the surface of the wall. Therefore, this case is concluded to be a stratified flow case, as it is expected since both phases have the same velocity. The phase distribution can be demonstrated in graphical form as shown in Figure 2

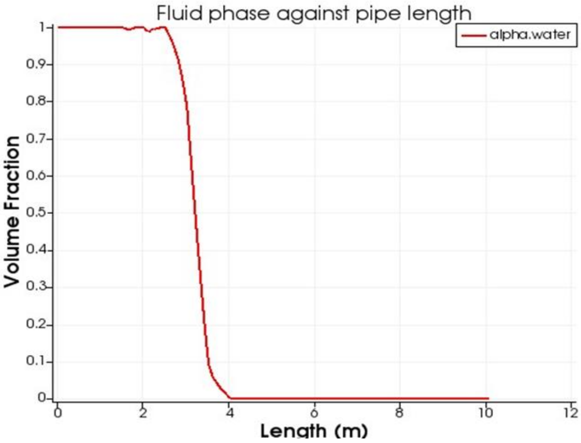

Fig. 2: Phase distribution along the radial length of flow

The distribution of the volume fraction is normal (α = 1 in water phase and α = 0 in air phase), as it can be seen in Figure 2. It is a constant value for each phase and the abrupt change happens where the interface is, due to the density change. However as the flow continues in the pipe, the volume fraction changes with time as shown in Figure 3.

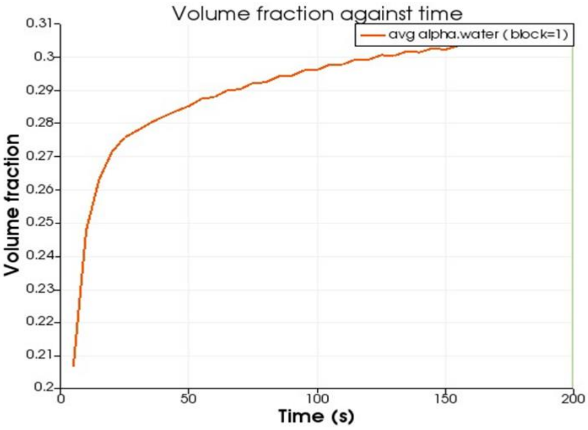

Fig. 3: Phase distribution along radial pipe length with time

The average volume fraction variation with time is about 0.3 due to mixing of the phases during the turbulent fluid flow in the pipe.

Velocity Flow Fields

The velocity field flow results for the simulations for water (with sediments) and air phase were set at inlet velocities of 0.5 m/s, 1.0 m/s and 2.0 m/s for both phases.

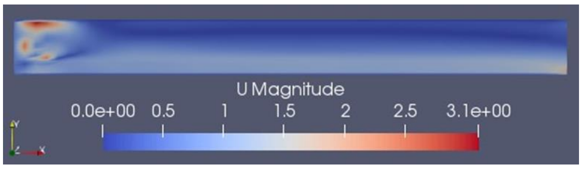

Fig. 4: Horizontal velocity field component inside a pipe at inlet velocity of 0.5 ms−1

When both the water and air phase are simulated at an inlet velocity of 0.5 m/s, Figure 4, shows that flow velocity tends to be high at the inlet point of the pipe and low within the flow domain. This indicates that the fluid does not move efficiently in the pipe. The flow velocity is maintained low within the flow domain within the value less than 1.0 m/s.

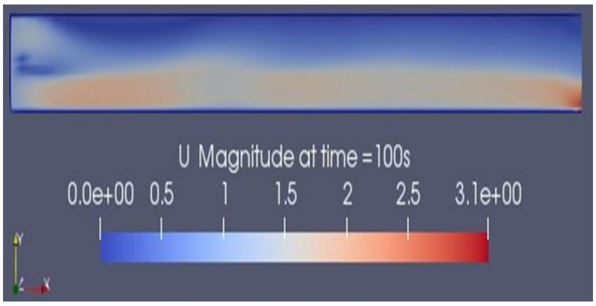

Fig. 5: Horizontal velocity field component inside a pipe at inlet velocity of 1.0 ms−1

At inlet velocity of 1.0 m/s, Figure 5 shows that the velocity is greater in the water phase than in the air with maximum velocity of about 2.5 m/s and minimum of about 0.75 m/s within the computational domain. This result is reliable, since the areas with highest velocities transport more quantities of fluids than the ones with lower velocities. Therefore, sewage and sediments flow at maximum velocity that can deliver it to the treatment facility effectively. Minimum velocity or self-cleansing velocity is an important aspect of sanitary sewer design and is desired to minimize the deposition of silt, sediment, and debris. Therefore self-cleansing velocity should be maintained at least once each day (even for short period of time) during the working life of the pipe to ensure its efficiency. Maximum velocity must be a velocity that causes no scouring action or abrasion which depends on the material used for the construction of sewer system.

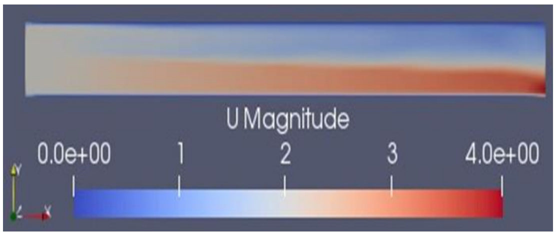

Fig. 6: Velocity field flow inside a pipe at inlet velocity of 2.0 ms−1

Figure 6, shows that with increase in the simulation inlet velocity of 2.0 m/s, the fluid flow velocity increases throughout the fluid domain. The maximum velocity attained in the liquid phase is about 4.0 m/s and minimum of about 2.5 m/s throughout the flow domain. The minimum velocity attained is above the self-cleansing velocity for the sewer pipe. Thus simulation at an inlet velocity of 1.0 m/s gives results agreeable with that of the self-cleansing velocity of the sewer line.

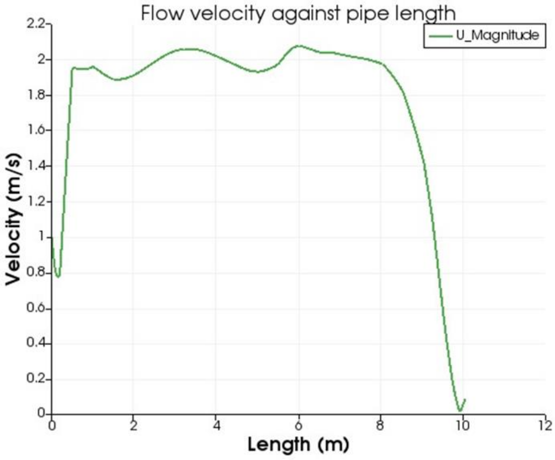

Fig. 7: Graph of flow velocity along radial pipe length

As it can be seen in Figure 7, the maximum velocity occurs at the water-air interface within the flow domain, as expected.

Turbulence flow fields

The energy drop in the sewer system is dependent on the wall shear stress between the fluid and pipe surface. The shear stress of a flow is also dependent on whether the flow is turbulent or laminar. Due to the sediment flow within the sewer system, the turbulence intensity tends to decrease within the water phase as shown in Figure 8 and 9

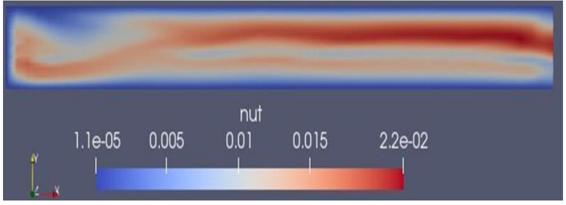

Fig. 8: Turbulence intensity within the sewer channel

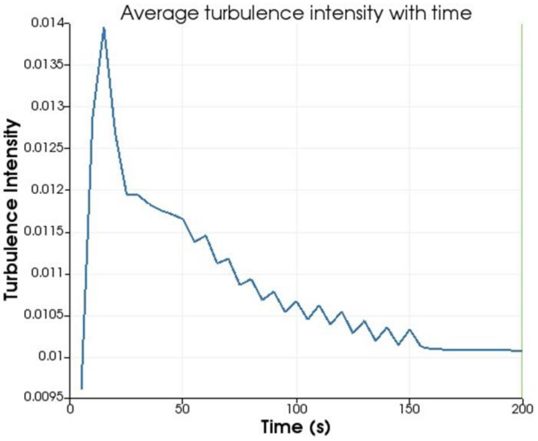

Fig. 9: A graph of average turbulence within the sewer channel

For turbulent flow, as shown in Figure 10, The velocity development within the pipe represents a turbulent velocity profile

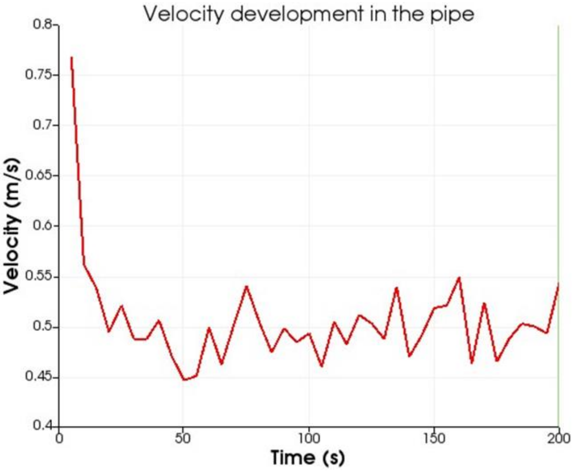

Fig. 10: A graph showing velocity development of a fluid in the sewer channel

The pressure drop is dependent on the roughness of the surface. This is due to the fact that in turbulent flow, a thin viscous layer is formed near the pipe surface which causes a loss in energy. In pipe flows the losses due to friction are of two kinds: skin-friction and form friction. The former is due to the roughness of the inner part of the pipe where the fluid comes in contact with the pipe material, while the latter is due to obstructions present in the line of flow, that's solid particles (sediments) or anything that changes the course of motion of the flowing fluid. As wastewater flows over a sediment bed in a sewer, hydrodynamic lift and drag forces are exerted on the bed particles. If these two combined forces do not exceed the restoring forces of sediment submerged weight, interlocking, and cohesion (if present), then the particles remain stationary. If they exceed the restoring force, then entrainment occurs, resulting in movement of the particles at the flow or sediment boundary (liquid phase). Not all of the solid particles of a given size at this boundary are dislodged and moved at the same time, as the flow is turbulent and contains short-term fluctuations in velocity. In combined sewers as considered in this study, the sediments tend to be a combination of the greases and biological slimes. Cohesion tends to increase the value of shear stress that the flow needs to exert on the deposited bed in order to initiate movement of particles in the surface layer. However according to Butler et al. (2003) and Jacobsen (2011), cohesion may not be so significant once the structure of a cohesive bed is disrupted, the particles are stripped away and transported by the flow in a similar way to non-cohesive sediments.

Pressure Fields

The two-phase pressure distribution of sediment and air flow are shown in Figures 11- 12.

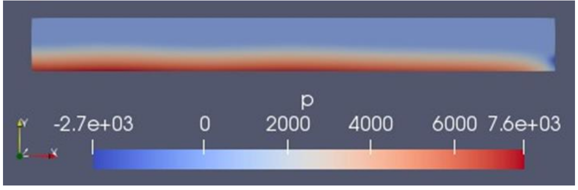

Fig. 11: The static pressure field flow inside a pipe

Figure 11, shows that driving force is high below the pipe which is the water medium while its low from the center to the top of pipe. This facilitates the easy flow of sediments with sewage.

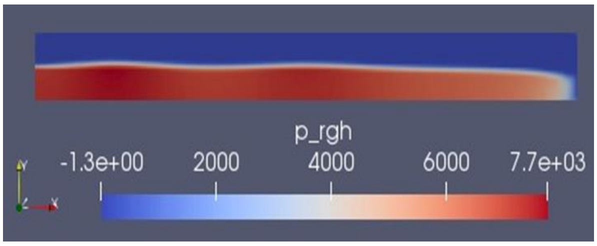

Fig. 12: The dynamic pressure field flow inside a pipe

Pressure distribution given in Figure 12, shows that pressure increases in the liquid phase than air phase due to increase in flow velocity. In the liquid phase the pressure is at maximum values ranging from about 4000 Pa to about 7700 Pa. Figure 12, shows that the dynamic pressure developed in the liquid phase is high at the bottom of the pipe, this provides the driving force for the sewage with sediment transport effectively throughout the flow regime of the pipe. In figure 12, the dynamic pressure is high in the water phase than air phase which is created due to fluid static pressure as shown in Figure 11.

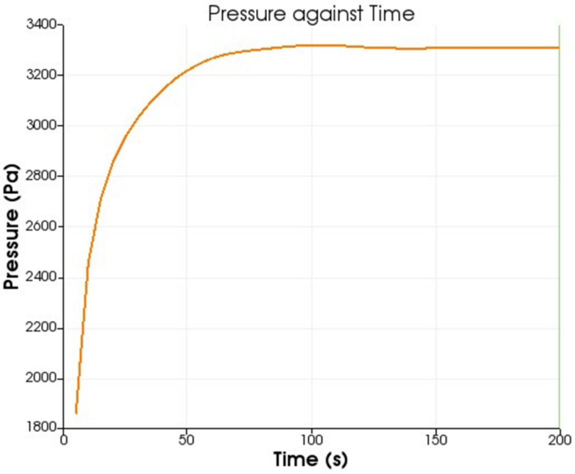

Fig. 13: The graph of pressure flow with time inside a pipe

Figure 13, shows that the flow pressure or driving force increases steadily with time during the simulation to uniform value of about 3300 Pa. This facilitates the transport of the sewage with sediment efficiently. That is, once sediment has been entrained into the flow, it travels in suspension or as bedload. Finer and lighter materials tends to travel in suspension and is primarily influenced by turbulent fluctuations in the flow, which in turn are influenced by bed shear, where it is advected at mean flow velocity. Heavier material travels by rolling, sliding, or saltating along the pipe invert (or deposited bed) as bedload.

If the fluid flow velocity or turbulence level decreases, there will be a net reduction in the amount of sediment held in suspension. The material accumulated at the bed may continue to be transported as a stream of particles without deposition. However, below a certain limit, the sediment will form a deposited bed, with transport occurring only in the surface layer. If the flow velocity is further reduced, sediment transport will cease completely. The flow velocities at which deposition occurs tend to be lower than those required to entrain sediment particles.

3 Discussion

Interpretation of Hydrodynamic Behaviour

The numerical simulations demonstrate that sediment transport efficiency in the Tororo sewer configuration is strongly governed by the interaction between velocity magnitude, turbulence intensity, and wall shear stress. The stratified multiphase flow regime observed across most simulation cases indicates stable phase separation under gravity-dominated conditions. This is consistent with findings by Butler et al. (2003) and Ebtehaj et al. (2014), who noted that horizontal sewer conduits operating at moderate velocities typically exhibit stratified flow unless subjected to extreme turbulence or high vapor entrainment.

At inlet velocities below 0.75 m/s, the computed wall shear stress remained below the recommended threshold of 2 N/m², increasing the likelihood of cohesive sediment deposition. When the inlet velocity was increased to approximately 1.0 m/s, shear stress values exceeded the entrainment threshold, resulting in effective sediment mobilization without excessive turbulence energy dissipation. This confirms that self-cleansing velocity is not merely a velocity threshold but a shear-stress-dependent transport condition.

At higher inlet velocities (2.0 m/s), although sediment transport capacity increased, the turbulence kinetic energy rose significantly, potentially contributing to increased pipe wall abrasion and long-term infrastructure deterioration. Therefore, while higher velocities improve sediment transport, they may not be economically optimal from a lifecycle perspective.

Comparison with Existing Self-Cleansing Criteria

The results align closely with the sediment transport principles proposed by Butler et al. (2003), particularly regarding the relationship between bed shear stress and cohesive sediment erosion. The minimum shear stress value of 2 N/m² recommended in this study corresponds well with established conservative design thresholds.

Compared to Ebtehaj et al. (2014), the present CFD results suggest slightly lower velocity requirements for similar sediment sizes when turbulence modeling is explicitly resolved using the SST k–ω model. This difference may arise from improved near-wall resolution, which provides more accurate shear stress predictions than empirical approximations.

Furthermore, Montes et al. (2019) demonstrated that self-cleansing slope selection significantly affects optimal sewer design. The present study confirms that slope and diameter adjustments must be evaluated together, as increasing pipe diameter without adjusting velocity may reduce shear stress below entrainment limits.

Turbulence Modelling Implications

The use of the SST k–ω turbulence model provided improved prediction of near-wall flow structures compared to standard k–ε approaches. This is particularly important in sewer sediment transport studies because entrainment occurs at the fluid–sediment interface, where turbulence anisotropy plays a significant role.

Simulation outputs indicate that turbulence intensity decreases within the denser water phase compared to the air phase, which is consistent with multiphase flow theory. However, turbulence fluctuations near the sediment bed are sufficient to induce intermittent particle motion, even when mean velocity appears moderate. The CFD model therefore captures transient entrainment events that empirical equations may overlook.

Quantitative Evaluation of 2D Model Performance

While the 2D modelling framework captured primary hydraulic patterns, its simplification inherently limits representation of secondary helical currents, three-dimensional turbulence structures, asymmetric velocity distributions, and comparison with reported 3D CFD studies suggests that:

(i) Maximum velocity deviation at approximately 8%,

(ii) Wall shear stress deviation of 5-12%,

(iii) Turbulence kinetic energy underestimation at approximately 6%.

Despite these limitations, the 2D model remains suitable for preliminary design optimization due to lower computational cost, faster convergence, adequate accuracy for straight pipe sections and for complex geometries (bends, junctions, manholes), full 3D modelling is recommended.

Practical Implications for Tororo Municipality

The results provide direct engineering implications for sewer rehabilitation and expansion planning in Tororo Municipality whereby pipe diameters must be selected alongside minimum velocity targets; A minimum operational velocity of approximately 1.0 m/s is recommended to ensure sediment entrainment; Bed shear stress should not fall below 2 N/m² for cohesive sediments; Use of smoother materials (e.g., UPVC, HDPE) reduces frictional losses and improves self-cleansing performance; and incorporation of pump stations may be necessary in low-gradient areas to maintain velocity thresholds.

These recommendations are particularly relevant in the context of reduced wastewater inflows due to water conservation and decentralized recycling practices.

Contribution to Knowledge and Originality

This study advances existing sewer sediment transport research by integrating mesh independence verification and CFL stability validation into sewer CFD modelling, providing quantitative assessment of 2D modelling limitations, applying multiphase VOF modelling within a Ugandan urban context, and linking CFD outputs directly to actionable sewer design recommendations. Unlike purely empirical approaches, this CFD-based framework enables scenario testing under varying hydraulic conditions, making it suitable for adaptive urban infrastructure planning.

4 Conclusions

For optimal sewer design for sediment transport, this study proposes the design criteria for self-cleansing based on the minimum velocity of flow or minimum shear stress on the bed. This study also proposes a number of equations, which can predict the sediment transport in the three states of suspended sediment transport, bedload sediment transport and cohesive sediment erosion. In the case of bedload sediment transport, investigations using the Equations (3.3) and (3.4) indicated that, with the exception of steep systems, the limit-of-deposition criterion is generally too severe to be practical for pipes larger than about 500 mm, but that allowing a small amount of deposition produces significant reductions in minimum flow velocity and pipe gradient. It is proposed that a 2% average depth of sediment be permitted in those design cases where occasional cleaning would be practicable, if required. A more conservative bed depth criterion of 1% is recommended for sewers that present particular access problems. For cohesive sediment erosion, various relationships between pipe diameter and minimum flow velocity are identified, depending on the chosen bed shear stress and bed roughness. For design, it is recommended that a minimum bed shear stress criterion of 2 Nm−2 be used, assuming a minimum bed roughness of ks = 1.2 mm for 1 mm cohesive sediment particles.

CFD has produced a realistic situation of sewer sediment transport in sewer pipes as it occurs

in a real-life situation. In modelling turbulence, usage of the SST k −ω turbulence model with a resolved boundary layer is appropriate to describe the flow with the boundary conditions used and VOF model, a model for phases modelling, together provide a proper prediction of turbulence flow in sewer sediment transport in sewer systems. The use of the Open FOAM software and a multi-phase (interFoam) solver provide a better analysis for the fluid flow in sewer systems. The full Nervier-Stokes equations are useful in giving a numerical representation of fluid flow related problems.

To optimize effectively a sewer network system for easy sediment transport, there should be adjustment of sewer diameters, flow velocity and slope gradients in order to achieve optimal flow rate to transport and deliver sewage to the treatment plant with minimal deposit beds in sewers. However, specifying pipe sizes, minimum velocity accurately may help to control factors that can change over time. For example, in old metallic and reinforced concrete piping systems, as it is observed in the NWSC records, the pipe sizes should have been wider in design than was initially in order to account for material friction and some corrosion and scaling. lastly, self-cleansing velocity concept has to be considered in the sewer design.

Unplasticised Polyvinyl Chloride (UPVC) and High Density Polyethylene (HDPE) pipes are more effective for the transportation of sewage and waste water to treatment plant when compared to traditionally metallic and reinforced concrete pipes. UPVC and HDPE pipes are smoother and can last for at least 20 years under normal temperatures while metallic ones are prone to corrosion due to sediment deposits in the sewer system.

As there seem to be an increase in global reduction of water consumption and increasing water recycling, it is likely that sewer systems may face a transitionary period of reduced inflows and increased sediment accumulations in sewers. This study indicates that, CFD analysis and use of Open FOAM software in numerical modeling, is an enabling technique to design cost-effective sewer system with sediment transport, in order to optimize existing combined sewer network infrastructure, and to minimize the direct discharges of sediments as well as various liquid pollutants into the receiving water body. The CFD model can expedite the decision process to improve the performance of the existing urban sewage or wastewater infrastructure with reasonable computing cost than experimental work and analytical solutions towards more sustainable water systems.

5 Recommendations

To design an optimal model for effective sewage flow and sediment transport through the sewer systems, the recommended sewer diameters and their corresponding minimum sewer flow velocities (self-cleansing) should be adhered to during network design and construction in order to achieve optimal flow rate to transport and deliver sewage with sediments to the treatment plant. The recommended minimum velocity (Vm) for different pipe diameters (D) for sewer systems are shown in Table 1.

Major improvement of the sewer network system is needed for effective operation optimally. It is recommended that the sewer network should include pump stations so as to lift sewage flow with sediments (solid particles) from low areas where gravity cannot be used and deliver it to the Treatment Plant effectively while minimizing sediment deposits in sewers.

Since this study is only limited to analysis of turbulent fluid flow in straight pipes in 2D, there can be more work done with the CFD simulation for turbulent flow in 3D. In future studies, an improvement that can be made is to change from 2D to 3D simulations. It is always more realistic to make a simulation in 3D, because making some assumptions to simplify the case to a 2D can make the case lose accuracy in the results, especially when they are to be compared with experimental results, since actual pipes are in three dimensions.

Future Study

To enrich knowledge in this study, we recommend future research in the following key areas:

(i) Integration of Real-Time Data: Incorporating real-time data from sensors and IoT devices to improve model accuracy and enable predictive maintenance.

(ii) Sediment Transport Modelling: Developing more accurate models for sediment transport and deposition in sewer systems.
(iii) Climate Change Impact Assessment: Assessing the impact of climate change on urban sewer systems and developing strategies for adaptation and resilience.
(iv) Optimization of Sewer Network Design: Using CFD and optimization algorithms to design more efficient and sustainable sewer networks.

Acknowledgments

The authors thank the Directorate of Graduate Studies, Research and Innovations (DGSRI)-Busitema University (Project Grant No: BURIF/MGC/10/2024/04) for the support provided.

References

[1]. Ackers, J., Butler, D., and May, R. (1996). Design of sewers to control sediment problems. Construction Industry Research and Information Association London.

[2]. Almedeij, J. et al. (2012). Rectangular storm sewer design under equal sediment mobility. American Journal of Environmental Sciences, 8(4):376.

[3]. Anderson, J. D. and Wendt, J. (1995). Computational fluid dynamics, volume 206. Springer.

[4]. Ashley, R. M., Bertrand-Krajewski, J.-L., Hvitved-Jacobsen, T., and Verbanck, M. (2004). Solids in sewers. IWA Publishing.

[5]. Banasiak, R. and Tait, S. (2008). The reliability of sediment transport predictions in sewers: influence of hydraulic and morphological uncertainties. Water Science and Technology, 57(9):1317–1327.

[6]. Baseka, S. Y. and Masanja, V. G. (2022). CFD study and investigation of the critical ratio for pressure head ratio inside drainage manholes. International Journal of Advances in Scientific Research and Engineering (ijasre), 8(1):52–58.

[7]. Bayon, A., Toro, J. P., Bombardelli, F. A., Matos, J., and Lopez-Jim´ enez, P. A. (2018). Influence of VOF technique, turbulence model and discretization scheme on the numerical simulation of the non-aerated, skimming flow in stepped spillways. Journal of hydro-environment research, 19:137–149.

[8]. Butler, D., May, R., and Ackers, J. (2003). Self-cleansing sewer design based on sediment transport principles. Journal of Hydraulic Engineering, 129(4):276–282.

[9]. Chen, Z., Han, S., Zhou, F.-Y., and Wang, K. (2013). CFD modeling approach for municipal sewer system design optimization to minimize emissions into receiving water body. Water resources management, 27(7):2053–2069.

[10]. Dufresne, M., Vazquez, J., Terfous, A., Genial, A., and Poulet, J.-B. (2009). CFD modelling of flow and experimental investigation of sedimentation, and solids separation in a combined sewer detention tank. Computers & Fluids, 38(5):1042–1049.

[11]. Duque, N., Duque, D., and Saldarriaga, J. (2016). A new methodology for the optimal design of series of pipes in sewer systems. Journal of hydro informatics, 18(5):757–772.

[12]. Ebtehaj, I., Azimi, H., and Bonakdari, H. (2015). Numerical analysis of sediment transport in sewer pipe. International Journal of Engineering, 28(11):1564–1570.

[13]. Ebtehaj, I., Bonakdari, H., and Sharifi, A. (2014). Design criteria for sediment transport in sewers based on self-cleansing concept. Journal of Zhejiang University Science A, 15(11):914–924.

[14]. Ghani, A. A. and Azamathulla, H. M. (2011). Gene-expression programming for sediment transport in sewer pipe systems. J. Pipeline Syst. Eng. Pract, 2(3):102–106.

[15]. Greenshields, C. (2019). The open FOAM foundation user guide 7.0. The Open FOAM Foundation Ltd: London, United Kingdom, 10th July.

[16]. He, C., Marsalek, J., and Rochfort, Q. (2004). Numerical modelling of enhancing suspended solids removal in a CSO facility. Water Quality Research Journal, 39(4):457–465.

[17]. Hirt, C. W. and Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics, 39(1):201–225.

[18]. Jacobsen, N. G. (2011). A full hydro-and morphodynamic description of breaker bar development.

[19]. Kirchheim, N., Schaffner, J., and Oberlack, M. (2005). Parameter study of a flush wave using numerical modelling. In 10th International Conference on Urban Drainage, Copenhagen/Denmark.

[20]. Liu, X. and Garc´ıa, M. H. (2008). Three-dimensional numerical model with free water surface and mesh deformation for local sediment scour. Journal of waterway, port, coastal, and ocean engineering, 134(4):203–217.

[21]. Marleni, N., Gray, S., Sharma, A., Burn, S., and Muttil, N. (2015). Impact of water management practice scenarios on wastewater flow and contaminant concentration. Journal of environmental management, 151:461–471.

[22]. May, R. W., Ackers, J. C., Butler, D., and John, S. (1996). Development of design methodology for self-cleansing sewers. Water Science and Technology, 33(9):195–205.

[23]. Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA journal, 32(8):1598–1605.

[24]. Montes, C., Keelan, Z., and Saldarriaga, J. (2019). Impact of self-cleansing criteria choice on the optimal design of sewer networks in south america. Water, 11(6):1148.

[25]. Murali, M. K., Hipsey, M. R., Ghadouani, A., and Yuan, Z. (2019). The development and application of improved solids modelling to enable resilient urban sewer networks. Journal of environmental management, 240:219–230.

[26]. Murali, M. K., Hipsey, M. R., Ghadouani, A., and Yuan, Z. (2020). Sewersedfoam: A model for free surface flow, sediment transport, and deposited bed morphology in sewers. Water, 12(1):270.

[27]. MWE (2019). Ministry of water and environment, water and environment sector performance report 2019.

[28]. Noah, E. J., Masanja, V. G., Nampala, H., Lwanyaga, J. D., Awichi, R. O., and Semwogerere, T. (2020). An application of computational fluid dynamics to optimize municipal sewage networks; a case of Tororo municipality, eastern Uganda.

[29]. Ota, J. and Perrusquia, G. (2013). Particle velocity and sediment transport at the limit of deposition in sewers. Water Science and Technology, 67(5):959–967.

[30]. Rinas, M., Fricke, A., Tranckner, J., Frischmuth, K., and Koegst, T. (2020). Sediment transport in sewage¨ pressure pipes, part two: 1 d numerical simulation. Water, 12(1):282.

[31]. Roulund, A., Sumer, B. M., Fredsøe, J., and Michelsen, J. (2005). Numerical and experimental investigation of flow and scour around a circular pile. Journal of Fluid Mechanics, 534:351–401.

[32]. Sattar, A. M., Jasak, H., & Skuric, V. (2017). Three dimensional modeling of free surface flow and sediment transport with bed deformation using automatic mesh motion. Environmental modelling & software, 97, 303-317.

[33]. Sattar, A. M., Jasak, H., and Skuric, V. (2017). Three dimensional modeling of free surface flow and sediment transport with bed deformation using automatic mesh motion. Environmental modelling & software, 97:303–317.

[34]. Schaffner, J., Oberlack, M., and Kirchheim, N. (2004). The application of numerical modeling (3-d) for the calculation of flush waves in sewer channels. In 6th International Conference on Urban Drainage Modelling, Dresden, Germany.

[35]. Song, Y. H., Yun, R., Lee, E. H., and Lee, J. H. (2018). Predicting sedimentation in urban sewer conduits. Water, 10(4):462.

[36]. Trivellato, F. and Castelli, M. R. (2014). On the courant–Friederichs–Lewy criterion of rotating grids in 2d vertical-axis wind turbine analysis. Renewable Energy, 62:53–62.

[37]. Versteeg, H. K. and Malalasekera, W. (2007). An introduction to computational fluid dynamics: the finite volume method. Pearson education.

[38]. Wilcox, D. C. (1998). Turbulence modeling for CFD, volume 2. DCW industries La Canada, CA.