Advances in Environmental Technology 9(4) 2023, 294-309

Iranian Reasearch Organization for Science and Technology (IROST)

Advances in Environmental Technology

Journal home page: https://aet.irost.ir/

Numerical simulation of turbulent water-solid-particle flows to predict the solid deposition process and the velocity distribution of water in sewage pipes

Farida Merrouchi[1*], Ali Fourar[2], Abdelatif Zeroual[1], Hala Makhloufi, Bouzghaya Aymen[1]

[1]*Faculty of Science and Applied Science, Hydraulic Department, University of Oum El Bouaghi, Oum El Bouaghi, Algeria.*

[2]*Faculty of Technology, Hydraulic Department, University of Batna2, Batna, Algeria.*

ARTICLE INFO

Document Type:
Research Paper

Article history:
Received 31 March 2022
Received in revised form 15 October 2023
Accepted 18 October 2023

Keywords:
Pipe clogging
Solid transport
Eulerian-Lagrangian model
Dry weather
Rainy weather

ABSTRACT

Research on the dispersion and deposition of solid particles in sanitation networks is crucial due to its role in channel blockage and overflow of wastewater and stormwater systems. Conventional detection methods are excessively costly and demand a significant time investment, while predictive mathematical models are prone to uncertainties. This study aims to assess the influence of solid particles on fluid flow and incorporate the effects of added mass and pressure gradient into the equation governing particle behavior. It is motivated by observations in Algeria, where the density of solid particles is notably high, thereby accentuating their impact on wastewater flows. To achieve these objectives, a bidirectional Eulerian-Lagrangian coupling method is employed, combining the advantages of various turbulence models, including the k-ω-sst model and the standard discrete random walk (DRW) model. This approach enhances our understanding of solid particle dispersion and deposition in sanitation networks, contributing to more efficient management and prevention of pipe obstructions, with implications for environmental preservation and the sustainability of urban sanitation systems. The use of turbulence models recommended in this study is inspired by Kolmogorov's pioneering work on turbulence, while the integration of added mass and pressure gradient forces falls within the context of particle dynamics in suspension. By leveraging in-situ data and incorporating the aforementioned forces, this innovative approach deepens our understanding of the processes involved in solid particle dispersion and deposition in urban drainage networks. These advancements are pivotal for the management and prevention of pipe obstructions, thus contributing to the preservation of the environment and the sustainability of urban sanitation systems.

*Corresponding author
E-mail: merrouchi@univ-oeb.dz
DOI:10.22104/AET.2023.6162.1697

1. Introduction

Liquid-solid turbulent flows have a wide range of applications, particularly in the transport of wastewater in sewage works, such as wastewater treatment plants and sewerage systems; these latter are used to carry domestic effluents, industrial discharges, and run-off water. Several studies [1,2] have shown that these waters sometimes transport high concentrations of non-negligible substances and solid matter that can alter the quality of the receiving environment, in particular suspended solids (MES). In most of these networks, the hydraulic conditions do not ensure the transport of all the solid phases, and substantial deposits occur, leading to the clogging of the pipes. This causes the reduction of their hydraulic capacities, which can become catastrophic during heavy rains and subsequent overflows [3]. Understanding the solid transport process is fundamental for flow regulation in sanitation systems. Therefore, predicting the velocities of the liquid and particulate phases in wastewater pipes is of great importance. This study aims to expand our knowledge in hydraulics by examining often overlooked aspects, such as the variable size and density of solid particles, and their impact on the unstable, turbulent flow of wastewater in hydraulic infrastructures. We introduce two essential forces: the force due to the pressure gradient and the added mass force. To date, many studies, whether numerical or experimental, have attempted to predict the critical sediment deposition velocity [4-5-6]. The transport of solid materials in pipelines is multiform and inhomogeneous, with a significant concentration of sludge at the bottom of the pipe, primarily due to the effect of gravity [7]. Research conducted in our laboratory, see Table 1, reveals that wastewater transported by the sanitation networks in Eastern Algeria exhibits a particle size distribution of solid particles ranging from 0.0004 to over 0.02 m. These particles are classified as both fine and coarse, representing a dual-size granulometry particle transport [8]. During transportation, particles exert a significant influence on the fluctuation of wastewater flow velocities in transverse directions, altering the rotational structures of the flow. This complex dynamic is particularly manifested through vortex movements. The reduction in flow velocity during dry periods, characterized by infrequent and light precipitation, leads to an increase in particle concentration at the bottom of the pipe. This accumulation can result in the formation of a dense and slippery bed, promoting pipe wear and clogging. This phenomenon complicates the transport process [8]. As a result, it becomes imperative to accurately predict the distribution of velocity and concentration, as well as their variations, within a pipeline while considering various environmental conditions. These predictions are essential not only for locating deposition zones but also for anticipating pipe wear issues, thereby contributing to improving their economic efficiency. Numerous numerical and experimental studies have been conducted to deepen our understanding of the physics of flows combining water and solid particles in pipelines. Experimental work has significantly enriched our understanding of the distribution of solid particle concentration in pipelines, with significant contributions from researchers such as Kaushal and Tomita (2007) [9], Kaushal et al. (2005) [10], Gillies, Shook, and Xu (2008) [11], Roco and Shook (1983) [12], Miedema (2017) [13], and others. Their numerous experimental studies have considered a range of variables, including pipeline diameter, particle size, and flow conditions. Experimental research has also revealed an interesting phenomenon, namely a mismatch between the velocities of solid particles and water in turbulent flow within a pipeline. These experiments, conducted with particles of various sizes and densities, have shown that even in the case of dilute flow, the velocity of particles in the pipelines is generally lower than that of water, as evidenced by observations made in references [14-15], F. Ravelet et al. [16], and Salah Z et al. (2016) [17]. In summary, experimental deposition studies are time-consuming and often challenging to conduct with precision [18]. Thanks to recent technological advances, including the expansion of computing capabilities and progress in the field of numerical modeling, Computational Fluid Dynamics (CFD) has become an essential tool for analyzing multiphase turbulent flows, especially those involving solid particles, in hydraulic systems evacuation. Several models have been developed to accurately characterize these multiphase turbulent flows, among which the Euler-Lagrange model stands out. This model tracks the movement of individual particles while evaluating the hydrodynamic forces applied to them by the turbulent fluid. However, it should be noted that this method requires an explicit formulation of the interaction between particles and turbulence. On the other hand, the approach known as Euler-Euler treats the particle sedimentation phase as a continuum. The literature contains relatively few studies on the impact of particle size, density, and unsteady variations in volumetric concentration on turbulent flows. This study introduces two forces, namely the pressure gradient force and the added mass force, in which the influence of particle inertia on the flow is explicitly addressed. Messa and Malavasi (2015) [19], as well as Messa, Malin, and Malavasi (2014) [20], developed a two-phase model that they later integrated into the PHOENICS software to simulate liquid-solid flows in horizontal pipes. This model considers essential aspects, such as turbulent dissipation, momentum exchange, and the impact of shear stress on solid particles in contact with the wall. Sanjeev et al. (2017) [21] studied this problem and developed two modeling frameworks for simulating the transport of suspended sediments in open channels. One framework, known as the partial two-fluid model (PTFM), combined mass and momentum equations to derive a mixing equation. The other framework, known as the complete two-fluid model (CTFM), solved the two-phase flow for both liquid and solid particles but did not consider the importance of density ratio. However, the CTFM results did not outperform the PTFM in simulating the transport of diluted suspended sediments in open channels. In 2020, Enzu Zheng et al. [22] presented the DNS-DEM model, designed to study turbulent flows of fluids containing large particles. Subsequently, they conducted experiments to validate their model. The results of these experiments revealed a remarkable agreement between the velocity and concentration profiles predicted by the model and the experimental observations, with deviations generally less than 10%. Sibo Wang et al. (2020) [8] studied the flow process of sludge containing large particles in a horizontal pipeline without phase change using the Eulerian-Lagrangian model. To track the motion of the particles, they analyzed the dynamic characteristics of these sludges as they were transported through the pipeline. The simulation results closely matched the experimental data, highlighting the superiority of simulation in describing flow characteristics in the pipeline. None of the previously discussed models in this study can conclusively demonstrate the ability to accurately calculate the concentration distribution of particles near the wall, especially when it comes to the transport of particles with dual particle sizes. Furthermore, these models do not effectively integrate the influence of forces on solid transport. Among previous research addressing the density ratio in fluid-solid particle transport, the following studies can be mentioned. Mohammad et al. (2020) [23] proposed a new theory to predict the transport and deposition of asphalt particles under both laminar and turbulent flow conditions. Gao.x et al. (2022) [24] used the Eulerian-Lagrangian approach in their study to predict the deposition rate of solid particles. They integrated the virtual mass force and pressure gradient force into the solid particle momentum equation. Simulation results showed that the deposition rate closely matched experimental data when accounting for both forces. In this study, we focus on the mechanism of water flow in the presence of solid particles with densities higher than that of water within a three-dimensional and unsteady turbulent flow in a combined sewer system. The Euler-Lagrange DDPM numerical method, which is based on granular theory, was employed to conduct this analysis. Within our investigation, two fundamental forces into the motion equations were incorporated, namely the added mass force and the pressure gradient force. These forces were accounted for based on the actual operating conditions of the wastewater collection system, whether in dry or rainy periods. Additionally, we considered the effects resulting from variations in the inflow velocity to the pipeline and laboratory measurements related to particle diameters and density.

2. Mathematical modeling of the liquid phase, and solid transport

2.1. Governing equations

2.1.1. Continuous Phase

The DDPM formulation utilizes equations similar to those used in the multi-fluid approach but with the additional solution of the particle motion equation for each parcel, with parcel properties projected onto the Eulerian grid. Consequently, each grid cell has an average solid velocity and volume fraction, with no need to solve the continuity and momentum equations of the solid phase in the Eulerian framework. This model aims to represent a large number of particles, taking into account the collisions between particles by the introduction of the kinetic theory of granular flow (KTGF), which determines the particle-particle interactions on the Eulerian grid, that are then reintroduced into the equation of motion as a source. Here, Equation (1) is the conservation of mass equation for an individual phase, and Equation (2) is the corresponding conservation of momentum equation.

Conservation of mass equation

$$\frac{\partial}{\partial t}(\alpha_f\,\rho_f) + \vec{\nabla}\cdot(\alpha_f\rho_f\vec{u_f}) = 0 \tag{1}$$

Conservation of momentum equation

$$\begin{aligned}\frac{\partial}{\partial t}(\alpha_f\,\rho_f\vec{u_f}) + \vec{\nabla}\cdot(\alpha_f\rho_f\vec{u_f}\vec{u_f}) &= -\alpha_f\nabla p + \nabla.\left(\bar{\bar{\tau}}_f'' + \bar{\bar{\tau}}_f''\right) + \alpha_f\rho_f\vec{g} \\ &\quad - \sum_i^n K_{DPMi}\left(\vec{u_f} - \vec{u_p}\right) - \sum_i^n \vec{F}_p\end{aligned} \tag{2}$$

where u_f is the velocity vector, $\bar{\bar{\tau}}_f''$ is the shear stress tensor, *p* is the solid phase index, and K_{DPMi} is the interphase exchange coefficient due to drag calculated for the solid volume fraction in the Eulerian frame. The subscript *f* denotes the carrier fluid, α is the volume fraction, ρ_f is the density of the continuous phase in kg/m^3, $u_f$represents the fluid velocity in m/s, and p indicates the fluid pressure, Pa. The interaction between the liquid and solid phases in the DDPM model can be studied by three drag laws models: Gidaspow (1994), Syamlal (1993), and Wen and Yu (1966) [25,26,27]. Previous studies have shown that the results for mass flow and particle distributions with the Gidaspow and Wen models are almost identical but considerably different from the results of the Syamlal model. Therefore, the Gidaspow model was adopted in our calculations.

$$K_{DPMi} = \begin{cases} \dfrac{3}{4}C_D\dfrac{\alpha_p\alpha_f\rho_f\left|\vec{u}_f - \vec{u}_p\right|}{d_{pi}}\alpha_f^{-2.65}, & \alpha_f > 0.8 \\ 150\dfrac{\alpha_p(1-\alpha_f)\mu_f}{\alpha_f d_{pi}^2} + 1.75\dfrac{\rho_f\alpha_p\left|\vec{u}_f - \vec{u}_p\right|}{d_{pi}}, & \alpha_f \le 0.8 \end{cases} \tag{3}$$

where the drag coefficient C_D is calculated by the correlations of Morsi and Alexander (1972) [28].

$$C_D = \frac{k_1}{Re_p} + \frac{k_2}{Re_p^2} + k_3 \tag{4}$$

The particle Reynolds number presented in relation (5) is calculated by the formula below.

$$Rep_i = \frac{d_{pi}\rho_f\alpha_f\left|\vec{u}_f - \vec{u}_p\right|}{\mu_f} \tag{5}$$

The low and high Reynolds number model $k-\omega$-sst was used to complete the calculation, which combines the benefits of the two approaches ($k-\varepsilon\ ;k-\omega$) and includes a mixing function F_1that goes from $k-\omega$ in the near-wall zones (low Reynolds number) to $k-\varepsilon$ free flow (fully developed flow). The literature has proven that the $k-\omega$ -sst model is able to describe the boundary layer perfectly near the walls [29,30]. The transport equations of the $k-\omega$ -sst model for unsteady, incompressible flow are formed by obtaining the corresponding turbulent kinetic energy equation k and the turbulent frequency equation ω.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\partial k u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right) + \tilde{G}_k + Y_k + S_k \tag{6}$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\partial\omega u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_\omega\frac{\partial\omega}{\partial x_j}\right) + \tilde{G}_\omega + Y_\omega + S_\omega \tag{7}$$

In these equations, $\tilde{G}_k$ represents the production of turbulent kinetic energy due to mean velocity gradients, $\tilde{G}_\omega$ represents the generation of the dissipation frequency ω, Γ_k and Γ_ω are respectively the effective diffusivity of k and ω, Y_k and Y_ω respectively represent the dissipation of k and ω due to turbulence, and S_k and S_ω are source terms defined by the user, respectively.

2.1.2. Dense Discrete Particle Motion Model

The equilibrium equation of the force acting on a particle is utilized to determine its trajectory. Assuming spherical and smooth particles, the position of a specific particle in the cartesian coordinate system can be determined by the following equation:

$$\frac{d\vec{x}_p}{dt} = \vec{u}_p \tag{8}$$

The particle velocity u_p can be calculated by solving the force balance on the particle, which is expressed as:

$$\frac{d\vec{u}_p}{dt} = f_{Dp}(\vec{u}_f - \vec{u}_p) + \vec{g}\left(\frac{\rho_p - \rho_f}{\rho_i}\right) + \vec{f}_1 + \vec{f}_2 + \vec{f}_{interaction,p} \tag{9}$$

The first term on the right-hand side represents the drag force acting on an individual particle, where u_f is the velocity of the fluid phase and u_p is the velocity of the particle. $f_{Dp}$is defined in the relationship below.

$$f_{Dp} = \frac{\mu_f}{\rho_p dp_i^2}\frac{18C_{Dp}Re_{pi}}{24} \tag{10}$$

For the solid-liquid system, the virtual mass force is included in this study.

$\vec{f}_1$ is the virtual mass force: $\vec{f}_1 = C_{vm}\frac{\rho_f}{\rho_p}\left(\frac{d\vec{u}_f}{dt} - \frac{d\vec{u}_p}{dt}\right)$ (11)

$\vec{f}_2$ is the force due to pressure gradient: $F_2 = \left(\frac{\rho_f}{\rho_p}\right)(\vec{u}_f \nabla \vec{u}_f)$ (12)

where C_{vm} is the virtual mass factor with a default value of 0.5, since varying its value has no influence on the calculation results. The last term $\vec{f}_{interaction,i}$ represents the additional acceleration acting on a single particle from the interaction between the particles, which is calculated from the stress tensor of KTGF (the kinetic theory of granular flow).

$$\vec{f}_{interaction,p} = -\frac{1}{\rho_p}\nabla.\overline{\overline{\tau}}_i \tag{13}$$

In this study, the dispersion of particles in a turbulent flow was simulated by the stochastic method, often called DRW (random walk models).

3. Nature and position of the problem

3.1. Flow configuration and simulation set-up

Figure 1 depicts the test pipe, a forced corrugated PEHD horizontal pipe with D = 1000 mm and a length of 5 m that serves as the unit-type main pipe supplying the Ain-Beida wastewater treatment plant in the wilaya of Oum El Bouaghi, Algeria. In the pipes in which the wastewater circulates, the fluid that transports solid particles through the pipe, with an average velocity ranging from 0.5 to 4 m/s, is regarded as the carrier fluid and is typically water.

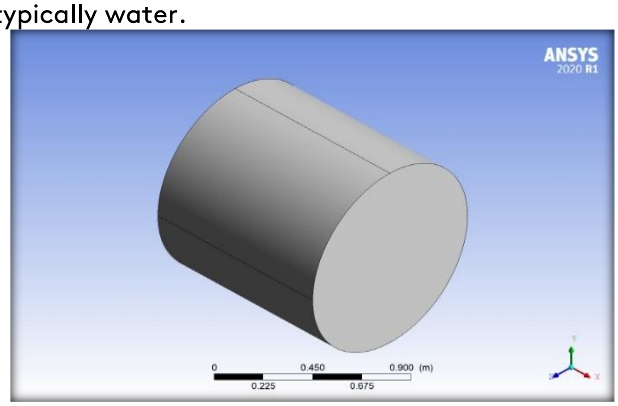

Fig. 1. Geometry of the pipe.

3.2. Characteristics of solids

In order to study the effect of the sizes and densities of the solid particles on the hydrodynamic behavior of turbulent flows in the sewerage works, samples of wastewater were taken from the sewerage pipe at two different times (dry and rainy); thus, the diameter and density of solid particles could be measured with the aim of using this data in numerical simulations. The simulation was performed for a mixture of solid particles of varying diameters and densities (see table number 1) as a particulate phase and water as a continuous phase; the corresponding inlet velocity (volumetric flow measured on the cross-sectional area of the pipe) for both dry and rainy weather was equal to 0.5 and 4 m/s, respectively.

Table 1. Physical characteristics of solid particles and different mixtures.

Condition	Velocity (m/s)	Density of the solid mixture measured (kg/m^3)	Average diameter of solid particles (m)	Minimum diameter of solid particles (m)	Max diameter of solid particles (m)
Dry weather	0.5	2354.9	0.0069	0.0004	0.02
Rainy weather	4	2109.1	0.0075	0.00063	0.02

4. CFD modeling

4.1. Computational mesh

The 3-D computational domain utilized in this study is depicted in Figure 2. The meshing process was comprised of two main stages: surface meshing and volume meshing. The surface grids were crucial for ensuring high-quality volume grids. The quantity of grids is a key parameter that determines the level of accuracy in computational simulations [31]. An investigation into the influence of grid quantity on computed results was carried out to determine a range of grid quantities that provided relatively better accuracy; the optimum mesh size was determined through a mesh sensitivity analysis. In this work, the accuracy was not affected by the quantity of grids within a range of 600,000 to 1.2 million grids. In consideration of time and simulation accuracy, this study utilized approximately 555,272 grids. The non-uniform unstructured grid used for discretizing the entire computational domain comprised hexagonal elements (refer to Figure 2). GAMBIT 2.4.6, grid generation software, was utilized for generating the grid.

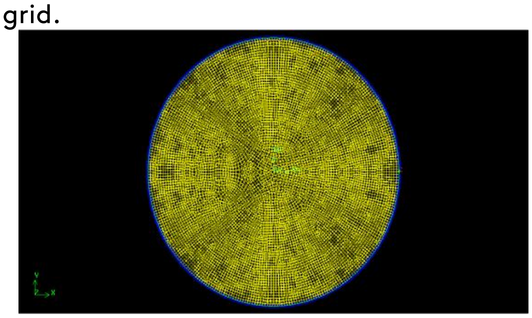

Fig. 2. Representation of the mesh density (555272 meshes, size 0.005).

4.2. Boundary conditions

The following real conditions were taken into account in this study: Vdry weather water inlet = 0.5 m/s and V rainy weather water inlet = 4 m/s; the initial and boundary conditions corresponded to those obtained from the technical report of the National Office for Sanitation (ONA) of Ain Baieda, Algeria. Regarding the sediment simulations, different densities were taken between (2354.9 kg/m^3 and 2109.1 kg/m^3) and different sizes (ranging from 0.4 mm to 20 mm). All the relevant input parameters of the simulation are summarized in Table 2.

Table 2. Boundary conditions for water and particulate phase.

Water	• Inlet water (velocity inlet) • Outlet water (pressure outlet) • Wall of the conduit (wall)
Particulate pollutants	✓ Pipe inlet and outlet: «Escape »,the particles exited the computation domain ✓ Lateral wall of the pipe: « Reflect » ✓ Bottom wall of the pipe« Trap »

4.3. Numerical methods

Numerical simulations were conducted using ANSYS Fluent software to solve the set of equations. The finite volume method was employed to solve the equations through an implicit scheme of first order in both time and space. For solving the boundary conditions, the pseudo-transitory method and the coupled algorithm were utilized with the double precision pressure-velocity solver. The PRESTO! pressure solution method was selected to achieve better convergence and maintain high accuracy. Standard discretization schemes were employed for pressure terms, whereas for convection and divergence terms, second-order upwind discretization schemes were utilized. In all unsteady simulations, the convergent criteria were established such that the residual in the control volume for each equation was less than 10^{-6}.

5. CFD simulation results

5.1. Sensitivity test

Validation of the mathematical model is important before its use in numerical modeling to ensure its accuracy; the model has been validated through various applications, ranging from simple to complex. We were initially interested in validating the numerical model for a negligible concentration of solid particles (pure water) to validate the numerical methods used to solve the Navier-Stokes equations, as well as their implementation. This study demonstrated the ability of the ANSYS Fluent software to model a single-phase, three-dimensional turbulent flow in a low-slope horizontal pipe. Subsequently, due to the lack of experimental data regarding the deposition of solid particles with a dual particle size (both fine and coarse) in urban combined sewer systems, our numerical model was validated by drawing upon the experimental work conducted by Roco and Shook in 1983 [12]. This choice was influenced by the similarities between the flow configuration of our problem and that of a water-sand system. Moreover, a bibliographic review highlighted that the studies conducted by Roco and Shook were more explicit and precise compared to the data presented in other articles [8].

5.1. 1. Pure water case

Figures 3 and 4 represent the distribution of flow velocity contours in the vicinity of the walls and at different sections of the pipe. In this study, the velocity field at different distances was known from the pipe using the k-omega-sst model and an average velocity measured at the inlet of the pipe of 0.5 m/s (dry weather) and 4 m/s (rainy weather), recognizing the areas of strongly turbulent flow so that the zones of particle deposit could be located, limiting the long-term good functioning of the sewer networks. The velocity in the vicinity of the pipe wall was observed to be nearly stagnant and close to zero. This was attributed to the high viscous shear stress in the boundary layer and the application of a slip-free boundary condition on the wall. The presence of this layer resulted in a gradual reduction in the velocity of fluid particles in the adjacent layer due to friction. To compensate for this velocity reduction, the fluid velocity at the midsection of the pipe increased to maintain constant mass flow through the pipe. Therefore, the more it moved radially towards the center of the pipe, it was observed that the axial velocity increased and reached a higher value. In general, the single-phase velocity for both velocities was symmetric with respect to the pipe axis (see Figures 3 and 4), and the liquid density remained constant over the cross-section of the pipe, which fully corresponded to the specialized literature (Cheng, 2004) [32-33].

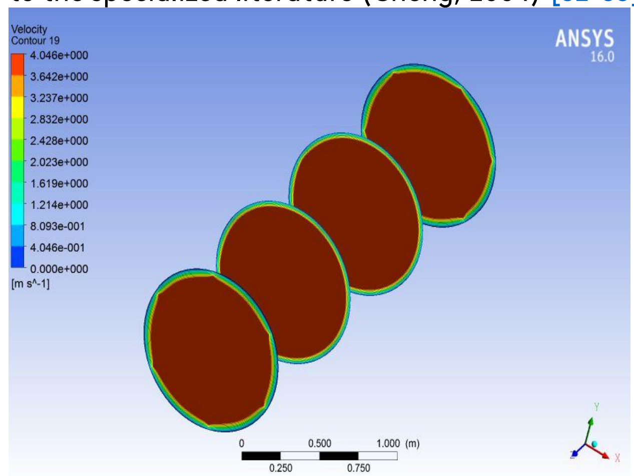

Fig. 3. Representation of velocity contours in (m/s) for rainy weather

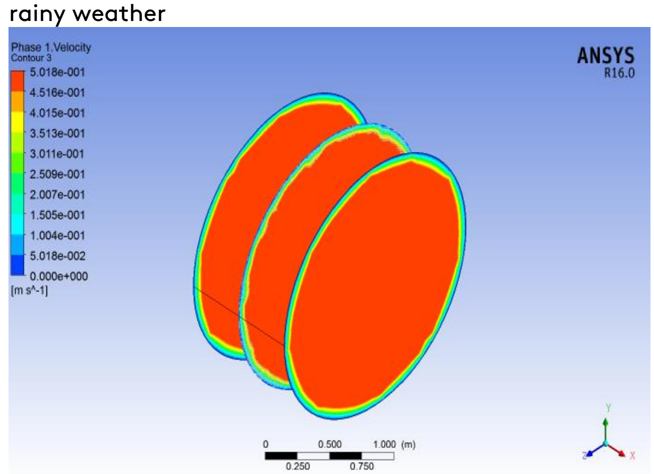

Fig. 4. Representation of velocity contours in (m/s) for dry weather.

5.1.2. Model validation for water-solid particles

In order to assess the impact of the added mass force and the force due to pressure gradients on the deposition of solid particles in sanitation systems, numerical simulations were conducted using two models with the following parameters: a diameter between 0.0004m and 0.02m, and v = 4 m/s. In the first model, only the drag force and turbulent dispersion force were considered in the equation of motion for solid particles. In the second model, in addition to these two forces, the virtual mass force and the force due to pressure gradients were integrated. The simulation results were compared to the experimental data published by Roco and Shook (1983) [12], as shown in Figure 5, illustrating the comparison of velocity distribution. There was a good agreement between the results of the second model and the experimental data, which means that the CFD simulation was accurate enough to predict the deposition of solid particles in the sewer pipe. However, for the first model, there was a difference compared to the results of the second model. This suggested that the absence of the virtual mass force and the force due to pressure gradients in the first model had an impact on the simulation's quality. Therefore, the second model was selected for all subsequent simulations.

.

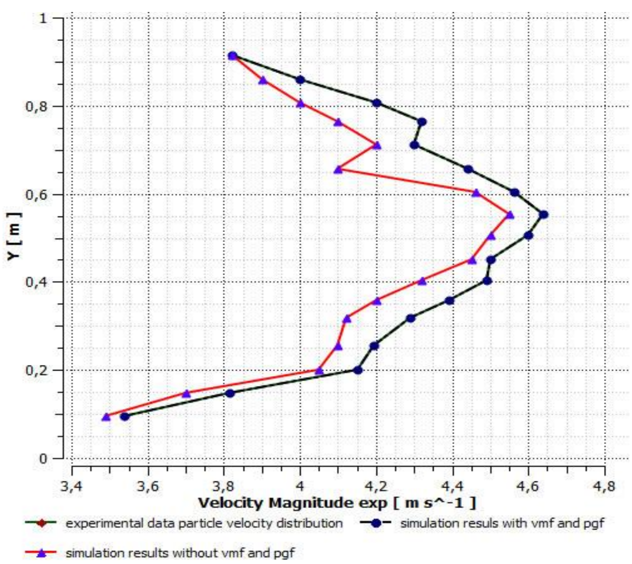

Fig. 5. Comparison of simulation results with experimental data particle velocity distribution for both simulation models.

5.2. A Visual Examination of Water Flow with Solid Particles

Numerical simulations were conducted for two sets of real-world tests (dry weather and rainy weather) in a sewage pipeline located at the entrance of the wastewater treatment plant. The parameters for these simulations are presented in Table 1. Figures 6 and 7 provide a qualitative comparison between the actual flow in the sewage pipeline at the entrance to the treatment plant and numerical simulations under two distinct flow scenarios: rainy weather (Figure 6) and dry weather (Figure 7). During rainy periods, there was evidence of mixing between the liquid and solid particles due to the high flow velocity (v = 4 m/s). This observation aligns with what is typically seen in the real pipeline (Figure 6) and is also consistent with previous findings published by Enzu Zheng et al. (2020) [22] and Sibo Wang et al. (2020) [8]. However, under dry weather conditions, the inclusion of larger solid particles and a lower velocity (0.5 m/s) resulted in a higher virtual mass force and a steeper pressure gradient. This led to an increase in the deposition rate, fostering the development of flow regimes characterized by a sliding bed, and the partial suspension was also observable (Figure 7). The numerical simulations were in agreement with these observations and consistent with the specialized literature, including works by Enzu Zheng et al. (2020) [22], Sibo Wang et al. (2020) [8], and Ming-Zhi Li et al. (2019) [7].

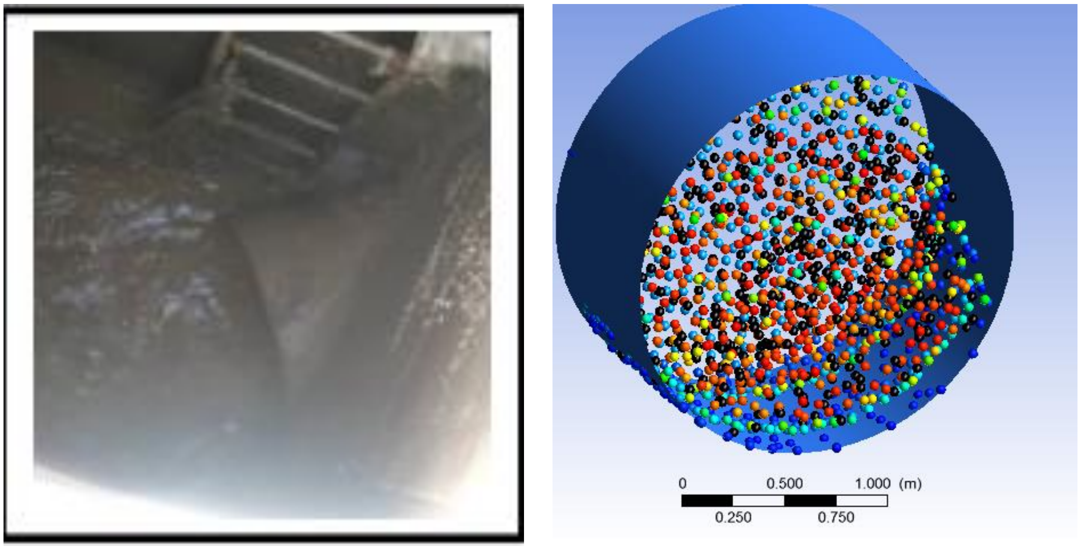

Fig. 6. Qualitative observation of the liquid-solid flow in the pipe (reality and simulation for rainy weather). (The black color represents solid particles and the other

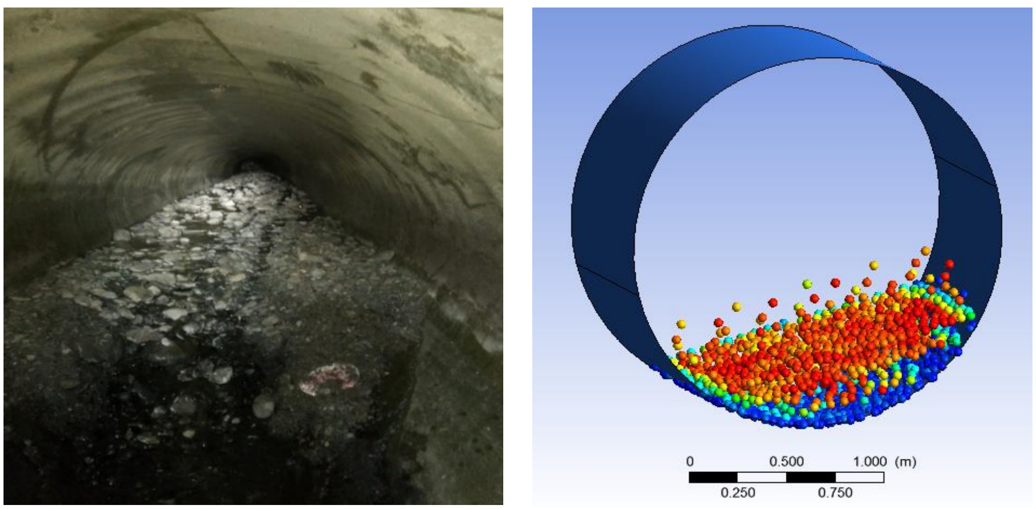

Fig. 7. Qualitative Qualitative observation of the liquid-solid flow in the pipe (reality and simulation for dry weather), where colors represent particle velocity magnitude.

For dry weather in fig 7, a pipe section far from the inlet was chosen so that the flow reached a statistically stable state

5.3. DDPM concentration profile

The profiles of the DPM volume fraction on the vertical centerline of the pipe for the two cases of our simulations (dry weather and rainy weather) are shown respectively in Figs. 8 and 10.

5.3.1. Dry weather

Figures 8 and 9 respectively depict profiles of the volume fraction distribution and solid volume concentration contours for dry weather conditions, characterized by low-flow velocity. Due to this reduced velocity, the presence of large-diameter solid particles, and the high density of solid particles, the distribution of solid volume fraction exhibited significant asymmetry. This asymmetry arose from particle deposition, leading to increased collisions among the particles themselves and between the particles and the conduit walls. These collisions have a substantial impact on result accuracy, and their influence extends beyond the effects of the forces integrated into the model. Consequently, a high particle concentration was recorded in the lower part of the conduit, while a lower concentration was observed in the upper section of the pipeline, where a few particles rebound off the top of the bed. It is also noteworthy that the concentration increased as the solid particle volume fraction rose. These findings were consistent with those of Ravlet et al. (2013) [16], Salah et al. (2016) [17], Yamaguchi et al. (2011) [34], Enzu Zheng et al. (2020) [22], Gao X et al. (2022) [24].

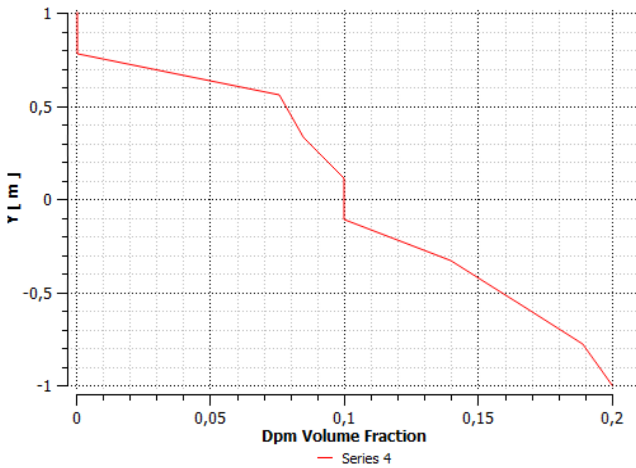

Fig. 8. DPM volume-fraction profile in the centerline of a pipe Test1 (V = 0.5M/S).

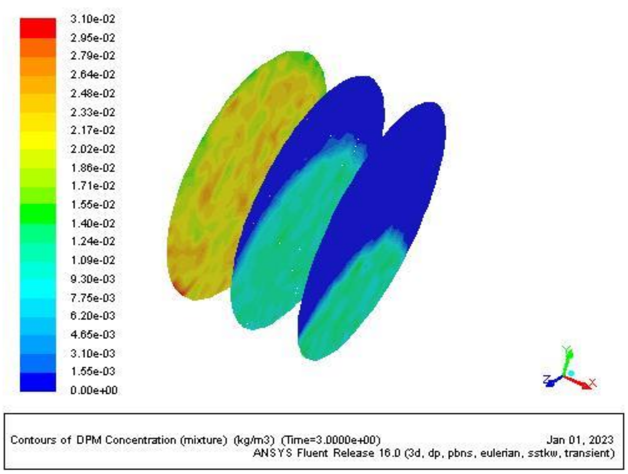

Fig. 9. 3D DPM concentration distribution in the pipe V=0.5M/S.

5.3.2. Rainy weather

Figure 10 depicts profiles of the volume solid fraction for rainy weather conditions characterized by high flow velocity. Due to this elevated velocity and the presence of solid particles of varying sizes, the volume solid fraction decreased, giving rise to a region where particles collided intensely, creating a highly turbulent liquid environment. Strong turbulent vortices could lift particles from the bed or eject them due to collisions with other bed particles. Finally, in the region farther from the bed, the average particle concentration was relatively low, and particles remained suspended due to fluid velocity fluctuations. These results align with the research conducted by Enzu Zheng [27,7].

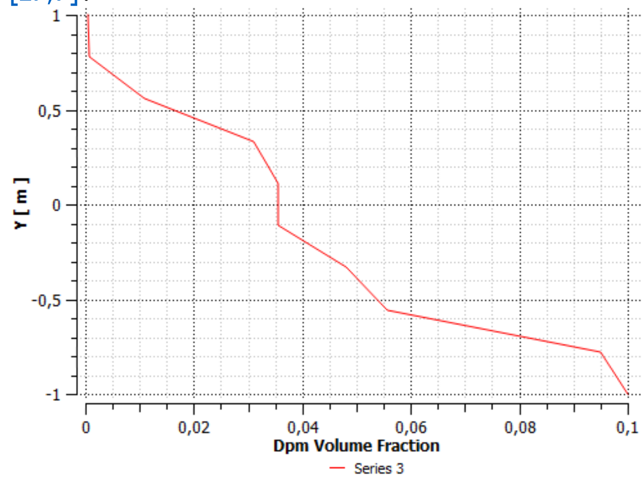

Fig. 10. DPM volume-fraction profile fraction in the centerline of a pipe: Test 2, V=4M/S.

5.4. Effect of solid particles on the velocity field

Figures 11 and 12 present comparison profiles that were derived from simulations involving the flow of water containing solid particles. These particles had a density of 2354.9 kg/m³ and a diameter ranging from 0.0004 m to 0.02 m for the first case (dry weather); for the second case (rainy weather), they had a density of 2109.1 kg/m³ and a diameter ranging from 0.00063 m to 0.02 m. The presence of solid particles and variations in flow velocity significantly impacted the velocity profiles of both the water and the solid phase. These profiles were plotted along a vertical diameter at the center of the pipeline, with a velocity at the pipeline's inlet (0.5 m/s in dry weather and 4 m/s in rainy weather) (Figures 11 and 12). For the liquid phase, velocity profiles in both simulation cases were symmetrical, with the maximum velocity slightly below the upper wall of the pipeline. In contrast, these profiles were asymmetrical for the particle phase, exhibiting a significant difference between the velocities of the liquid phase and the particle phase. Additionally, the relative velocity of particles in suspension was very low, and the point of maximum velocity gradually shifted downward. Our results were consistent with prior research published by Ming-

Zhi Li et al. (2019) [7]. Figure 12 reveals that the velocity of the particle phase in dry weather was slower than that of the liquid phase, except near the wall. Near the wall, the water velocity was zero due to the no-slip condition, while the particle phase exhibited tangential motion relative to the wall, allowing it to slide and roll, resulting in a higher flow velocity. The presence of larger particles led to an asymmetric liquid-solid flow profile, particularly at higher concentrations towards the bottom of the pipeline. These findings align with previous studies [7-8]. In the case of turbulent flow (rainy weather), the increased flow velocity and turbulent regime were responsible for the thorough mixing of the fluid and solid particles (Figure 11). The flow was modeled with the DDPM model for zero slip velocity.

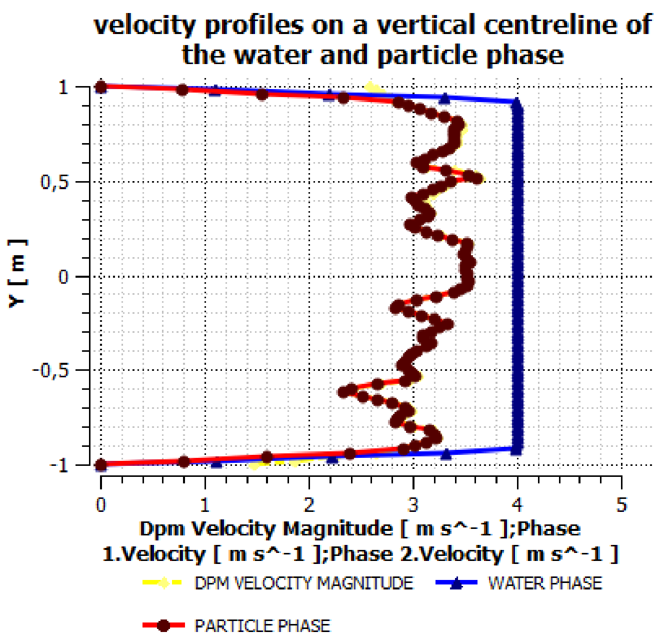

Fig. 11. Comparison of velocities of water and particle phase with different particles size for rainy weather.

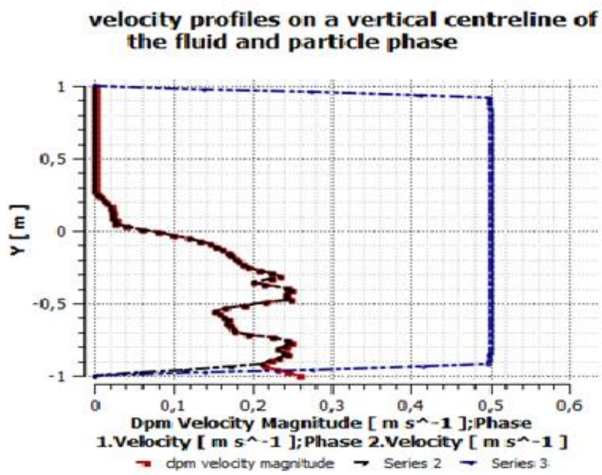

Fig. 12. Comparison of velocities of water and particle phase with different particles size for dry weather

5.5. Effect of virtual mass and pressure gradient force on the deposition of solid particles

This study examined the influence of the virtual mass force and pressure gradient on the solid particle deposition rate in a sewage pipeline. The simulation parameters included a flow velocity of 0.5 m/s. The solid particles had a density of 2354.9 kg/m^3 and diameters ranging from 0.0004 m to 0.02 m for the first case (dry weather); for the second case (rainy weather), they had a density of 2109.1 kg/m^3 and diameters ranging from 0.00063 m to 0.02 m, with a flow velocity of 4 m/s. Figures 13 and 14 present a comparison of solid particle deposition rates for the two simulation scenarios, namely dry and rainy weather, considering and neglecting the virtual mass force and pressure gradient forces. It is clearly demonstrated in Figures 13 and 14 that the solid particle deposition rate increased as the size of these particles increased and the velocity of the liquid phase decreased. This increase was attributed to the higher virtual mass force and pressure gradient due to the presence of larger diameter solid particles. It is also noteworthy that there was a significant difference between the results obtained when considering these two forces and when excluding them from the simulation. Our results were in agreement with the work published by [24].

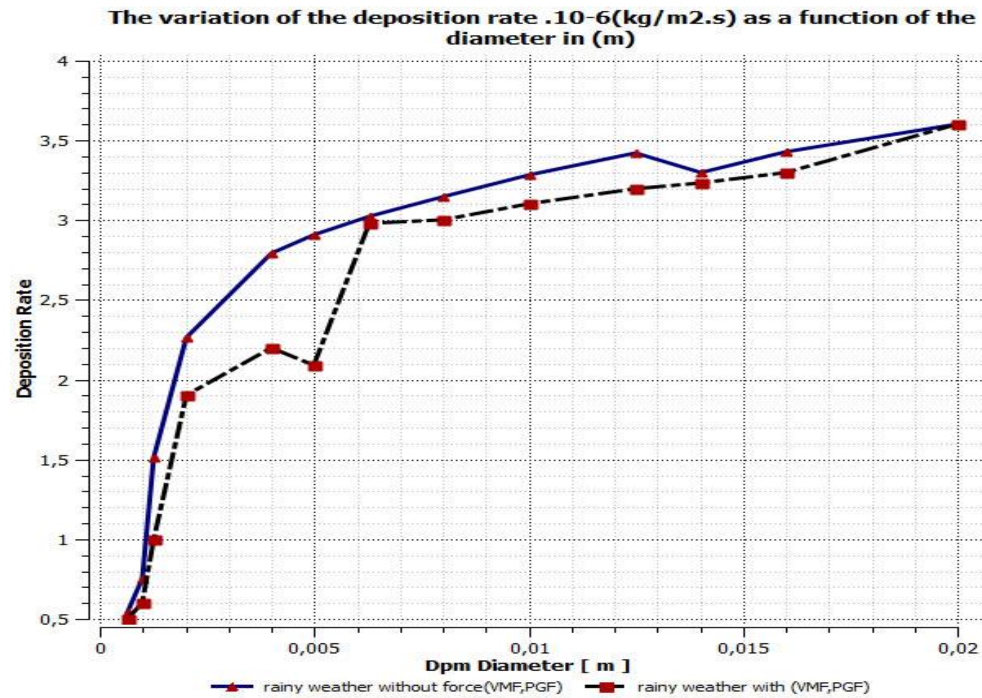

Fig. 13. Effect of pressure gradient, virtual mass force on the solid particulate deposition rate (rainy weather).

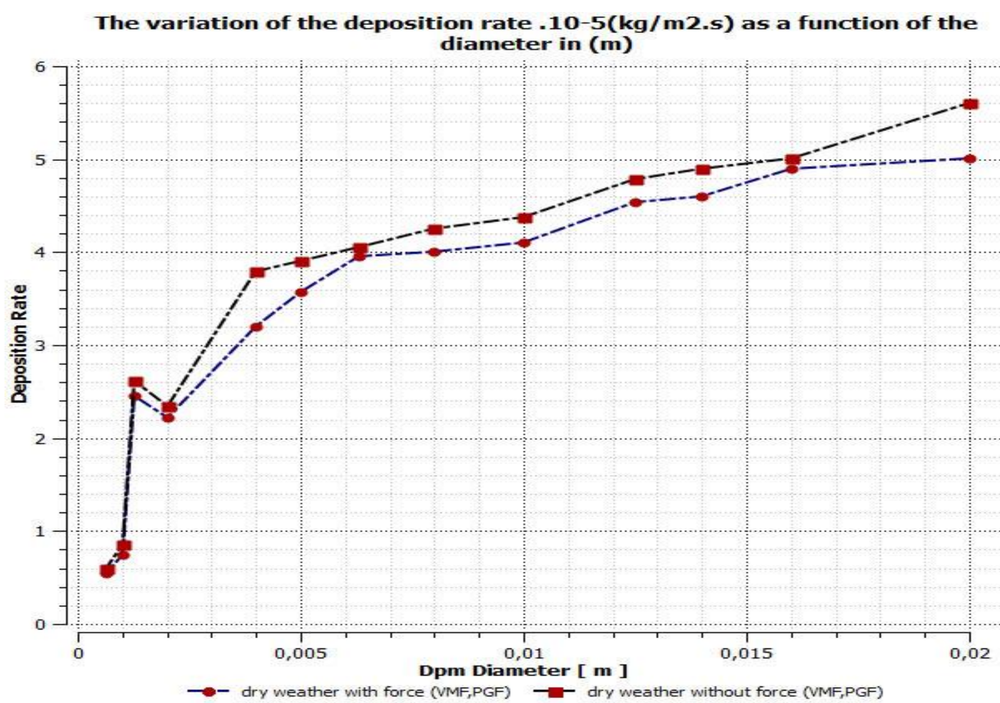

Fig. 14. Effect of pressure gradient, virtual mass force on the solid particulate deposition rate (dry weather).

5.6. Turbulent Kinetic Energy Profiles for Water and Particle Transport

The contours and profiles of the turbulent kinetic energy in m2/s2 to different sections are shown for rainy weather (Figures 15,16) and dry weather (Figures 17,18) in different planes of the pipe, verifying that the turbulent kinetic energy was less intense at the center of the pipe and increased as you approach the wall figure. The increase in the concentration and velocity of the mixture also induced turbulence, the impact of which could be decisive for the circulation of particles in the flow and the reduction of sedimentation. The flow was modeled with the DDPM model for zero slip velocity.

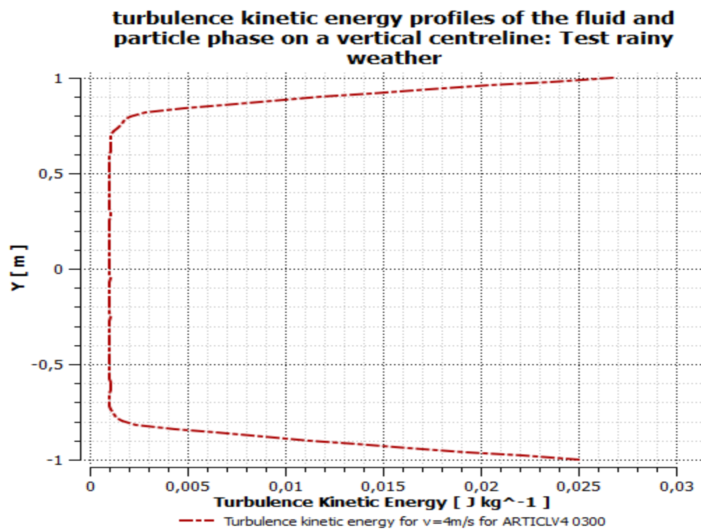

Fig. 15. Turbulence kinetic energy profiles for rainy weather.

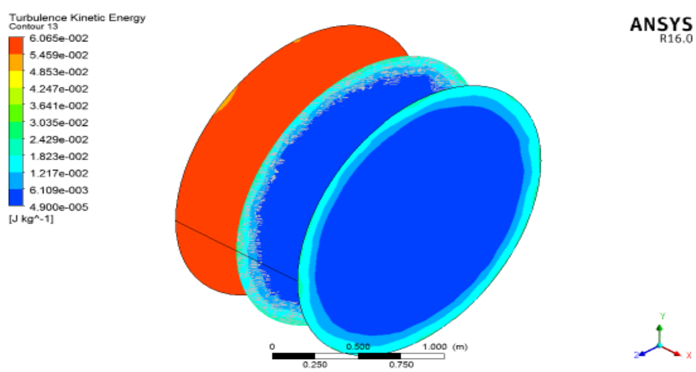

Fig. 16. Distribution of turbulent kinetic energy in m^2/s^2 to different sections for rainy weather.

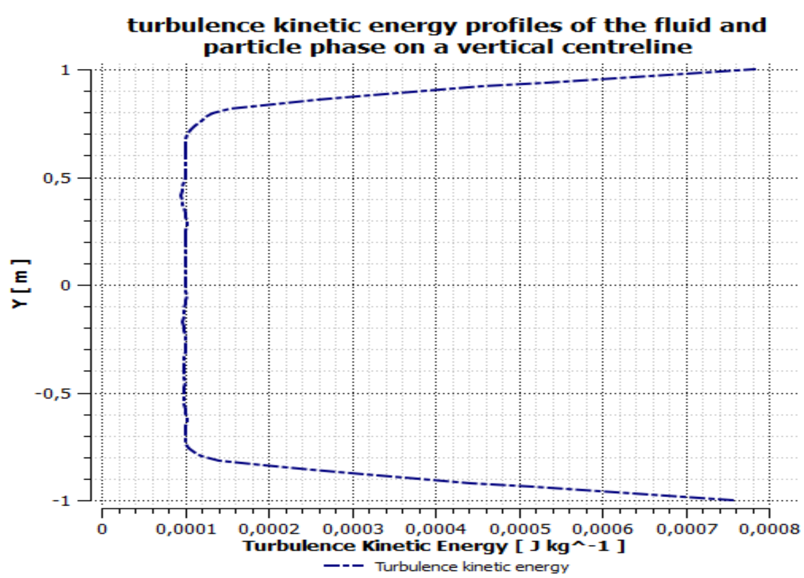

Fig. 17. Turbulence kinetic energy profiles for dry weather.

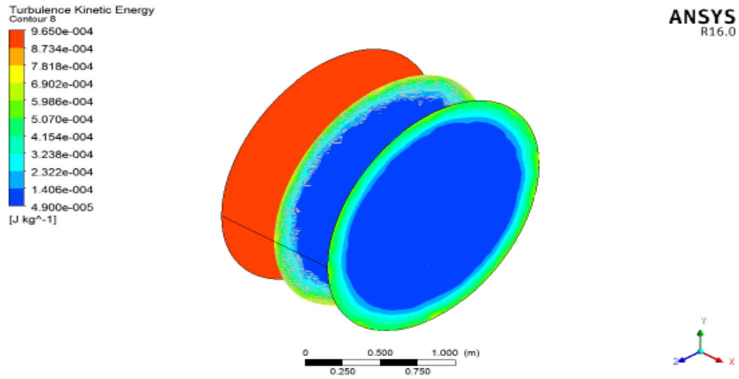

Fig. 18. Distribution of turbulent kinetic energy in m^2/s^2 to different sections for dry weather.

6. Conclusions

This work involved a numerical study of three-dimensional and unsteady two-phase flows with a free surface in sewer pipes, involving the presence of water and solid particles. The main objective was to simulate liquid flows and particle transport in urban wastewater drainage pipes to describe the evolution of the solid and liquid phases. Furthermore, this study aimed to calculate the velocity fields of the carrier liquid (water) as well as those of the transported solid particles. The Euler-Lagrange DDPM approach was chosen to model the flow and transport of large-diameter solid particles with high densities. In order to validate k-omega-sst models in the context of DDPM, simulations were performed in a 1000 mm diameter sewer pipe, with densities reaching 2400 kg/m3. The data concerning solid particles included visual observation of the flow at the entrance of the pipe under dry weather conditions, with an average velocity of 0.5 m/s, and under wet weather conditions, with a velocity of 4 m/s. In both cases (dry weather and wet weather), the flow of water carrying solid particles showed qualitative similarities, such as the formation of sediment deposits in the pipe when velocities were low (dry weather). In the context of velocity profiles, it is important to note the existence of a velocity imbalance between solid particles and water in turbulent flows. These flows, which involve particles of different sizes and densities, are characterized by a particle velocity lower than that of the carrier fluid, i.e., water. When the water velocity was low, particularly during dry periods, an even greater difference in velocity between solid particles and water could be observed. Introducing forces into the model also highlights collisions between particles, as well as between particles and walls. In the final model formation, drag and pressure gradient played a predominant role in particle motion. There was also a strong impact of virtual mass and viscous force in the direction perpendicular to the flow. In summary, force capture revealed the interactions between particles, walls, and the forces acting on their motion, such as drag pressure gradient, virtual mass, and viscous force, contributing to the formation of the model under study.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] Ashley, R. M., Bertrand-Krajewski, J.-L., Hvitved-Jacobsen, T., and Verbanck, M, (2004). Solids in sewers characteristics, effects and control of sewer solids and associated pollutants. IWA Publishing, London.

[2] Chebbo, G., (1992). Solids of Urban Waste in Rainy Weather: characterization and treatability. Doctoral thesis from ENPC, Paris, France.

[3] Guignard, J.-C., Bruyelle, J.-C., & Krommydas, C. (2017). Design and sizing of rainwater management and wastewater collection systems. Technical Instruction 77/284" attached to the Astee Sanitation Commission

[4] Bacchi, V. (2011). Experimental study of the sedimentary dynamics of a steep slope system subjected to weak hydraulic conditions. Doctoral thesis, University of Grenoble, France.

[5] Gaveau, N. (2022). Numerical and theoretical results on the Saint-Venant equations coupled to an erosion model or with Coriolis force. Doctoral thesis, University of Orléans, France.

[6] Pedroni, L. (2011). Experimental and numerical study of the sedimentation and consolidation of acid water treatment sludge. Doctoral thesis, Polytechnic School of Montreal.

[7] Li, M.-Z., He, Y.-P., Liu, Y.-D., Huang, C. (2019). Analysis of transport properties with varying parameters of slurry in horizontal pipeline using ANSYS fluent. *Particulate Science and Technology*, 161412. https://doi.org/10.1080/02726351.2019.1621412

[8] Wang, S., Ding, X., Wang, J. (2020). Numerical simulation of coarse particle pipeline transportation based on Eulerian Lagrangian model. *Journal of Physics: Conference Series, 1600*(1), 012005. https://doi.org/10.1088/1742-6596/1600/1/012005

[9] Kaushal, D. R., Y, Tomita. (2007). Experimental investigation for near-wall lift of coarser particles in slurry pipeline using c-ray densitometer. *Powder Technology, 172*(3):177–87. https://doi.org/10.1016/ j. powtec.2006.11.020

[10] Kaushal, D. R., K. Sato., T. Toyota, K., Funatsu, and Y, Tomita. (2005). Effect of particle size distribution on pressure drop and concentration profile in pipeline flow of highly concentrated slurry. *International Journal of Multiphase Flow 31*(7):809–23. https://doi.org/10.1016/j.ijmultiphaseflow.2005.03.003

[11] Gillies, R. G., Shook, C. A., Xu, J. (2008). Modelling heterogeneous slurry flows at high velocities. *The Canadian Journal of Chemical Engineering, 82*(5), 1060–1065. https://doi.org/10.1002/cjce.5450820523

[12] Roco, M. C., Shook, C. A. (1983). Modeling of slurry flow: The effect of particle size. The Canadian *Journal of Chemical Engineering, 61*(4), 494–503. https://doi.org/10.1002/cjce.5450610402

[13] Miedema, S. A. (2017). A new approach to determine the concentration distribution in slurry transport. Dredging Summit and Expo. 2017, USA. http://resolver.tudelft.nl/uuid:4e5db076-8814-43ef-af3d-761d8aa74229

[14] Muste, M., Yu, K., Fujita, I., Ettema, R. (2009). Two-phase flow insights into open channel flows with suspended particles of different densities. *Environmental Fluid Mechanics, 9*(2), 161–186. https://doi.org/10.1007/s10652-008-9102-7

[15] Nezu, L., Azuma, R. (2004). Turbulence characteristics and interaction between particles and fluid in particle-laden open channel flows. *Journal of Hydraulic Engineering-ASCE, 130*(10), 988-1001. https://doi.org/10.1061/(ASCE)0733-9429(2004)130:10(98

[16] Ravelet, F. F., Bakir, F., Khelladi, S., Rey, R. (2013). Experimental study of hydraulic transport of large particles in horizontal pipes. *Experimental Thermal and Fluid Science, 45*, P187–197. http://dx.doi.org/10.1016/j.expthermflusci.2012.11.003

[17] Zouaoui,S., Djebouri, H., Mohammedi, K., Khelladi, S., Aider, A. A. (2016). Experimental study on the effects of big particles' physical characteristics on the hydraulic transport inside a horizontal pipe. *Chinese Journal of Chemical Engineering, 24*(2), 317-322. https://cjche.cip.com.cn/EN/Y2016/V24/I2/317

[18] Teli, S., Kulkarni, S. (2023). Stirred tank reactor with dual impeller Rushton turbine for application of wastewater treatment-Process optimization and CFD simulation. *Advances in Environmental Technology*, 9(3), 174-193. https://doi.org/10.22104/AET.2023.6207.1710.

[19] Messa, G. V., Malavasi, S. (2015). Improvements in the numerical prediction of fully-suspended slurry flow in horizontal pipes. *Powder Technology, 270*, 358–367. https://doi.org/10.1016/j.powtec.2014.10.027.

[20] Messa, G. V., Malin, M., Malavasi, S. (2014). Numerical prediction of fully-suspended slurry flow in horizontal pipes. *Powder Technology, 256*, 61–70. https://doi.org/10.1016/j.powtec.2014.02.005.

[21] Sanjeev, Kumar. Jha. (2017). Effect of particle inertia on the transport of particle-laden open channel flow. European *Journal of Mechanics B/Fluids, 62*, 32–41. https://doi.org/10.1016/j.euromechflu.2016.10.012

[22] Zheng, E., Rudman, M., Kuang, S., Chryss, A. (2020). Turbulent coarse-particle suspension flow: Measurement and modeling. *Powder Technology, 373*, 647-659. https://doi.org/10.1016/j.powtec.2020.06.080

[23] Mohammad, Khamehchi., Mousavi, Dehghani., et al. (2020). A new theory for modeling transport and deposition of solid particles in oil and gas wells and pipelines. International *Journal of Heat and Mass Transfer, 152*, 119568. https://doi.org/10.1016/j.ijheatmasstransfer.2020.119568

[24] Gao, X., Dong, P., Chen, X., Yvan Nkok, L., Zhang, S., Yuan, Y. (2022). CFD modeling of virtual mass force and pressure gradient force on deposition rate of asphaltene aggregates in oil wells. *Petroleum Science and Technology*, *40*(8), 995-1017. https://doi.org/10.1080/10916466.2021.2008972

[25] Gidaspow, D. (1994). Multiphase flow and fluidization: Continuum and Kinetic Theory Descriptions. Academic Press.

[26] Syamlal, M., Rogers, W., O'Brien, T.J. (1993). MFIX Documentation: Theory Guide. National Technical Information Service, Springfield.

[27] Wen, C. Y. (1966). Mechanics of fluidization. In *Fluid Particle Technology, Chem. Eng. Progress. Symposium Series* (Vol. 62, pp. 100-111).

[28] Morsi, S. A. J., Alexander, A. J. (1972). An investigation of particle trajectories in two-phase flow systems. *Journal of Fluid Mechanics*, *55*(2), 193-208. https://doi.org/10.1017/S0022112072001806

[29] Gilles, I. (2016). Euler-Lagrange approach for modeling solid transport in settling structures. Doctoral thesis, INSA Strasbourg, France.

[30] Yan, H. (2013). Experiments and 3D modelling of hydrodynamics, sediment transport, settling and resuspension under unsteady conditions in an urban stormwater detention basin (Doctoral dissertation, INSA de Lyon).

[31] Amara, M., Bouledroua, O., Hadj Meliani, M., Azari, Z., Tahar Abbess, M., Pluvinage, G., Bozic, Z. (2019). Effect of corrosion damage on a pipeline burst pressure and repairing methods. *Archive of Applied Mechanics*, *89*(5), 939-951. https://doi.org/10.1007/s00419-019-01518-z.

[32] Cheng, N. S. (2004). Analysis of velocity lag in sediment-laden open channel flows. *Journal of Hydraulic Engineering*, *130*(7), 657-666. https://doi.org/10.1061/(ASCE)07339429(2004)130:7(6 57)

[33] Merrouchi, F. (2019). Numerical modeling of the variation in velocities of unsteady turbulent flows in free surface channels. Doctoral thesis, University of Batna 2, Algeria

[34] Yamaguchi, H., Niu, X. D., Nagaoka, S., De Vuyst, F. (2011). Solid-Liquid Two-Phase Flow Measurement Using an Electromagnetically Induced Signal Measurement Method. *Journal of Fluids Engineering*, *133*(4). https://doi.org/10.1115/1.4003856

How to site this paper:

Merrouchi, F., Fourar, A., Zeroual, A., Makhloufi, H., & Bouzeghaya, A. (2023). Numerical simulation of turbulent water-solid-particle flows to predict the solid deposition process and the velocity distribution of water in sewage pipes. *Advances in Environmental Technology*, 9(4), 294-309. doi: 10.22104/aet.2023.6162.1697.