Curtailing infection spread via drain pipelines: using interfacial hydrodynamics for removing bacterial and viral biofilms

P P SHAHABAZ and JANANI SRREE MURALLIDHARAN*

Department of Mechanical Engineering, Indian Institute of Technology Bombay, Mumbai 400076, India
e-mail: js.murallidharan@iitb.ac.in

MS received 1 February 2021; revised 3 July 2022; accepted 25 August 2022

Abstract. Drainage systems contain biological contaminants like bacteria and viruses flowing through them. Additionally, these pipelines also have organic matter known as biofilms growing on their walls. These biofilms in fact act as incubation zones for further growth of bacteria and coronaviruses. Standard water treatment routines with traditional cleaning agents are known to be not able to clean or sterilize microbes located in the inner layers of the biofilm. A recent study has identified specialised fluids which are effective in removing biofilms but these need to be used prudently. The present study proposes to use ‘interfacial hydrodynamics’ to ensure that the cleaner-fluid (CF) is transported effectively to the location of the biofilms at the pipe walls, and allowed to be in contact with the biofilms for a sufficient amount of time so as to ensure its effective removal. The present study has used CFD technique of Multi-fluid VOF and has demonstrated that relative superficial velocities of cleaner-fluids and sewage water can be controlled, so as to achieve flow regimes that ensure delivery of cleaner fluid to the periphery of the tube walls. Our simulations indicate that most effective cleaning can be achieved by using a cleaner-fluid with a high viscosity (~ 5000 cP). In such cases, a low- medium velocity (~0.05-0.3 m/s) of CF and water would ensure that the cleaner fluids are in constant contact with the pipe walls. Other suitable viscosity and velocity combinations have also been proposed. Flow parameters that can be used to monitor and cross-verify expected flow patterns on-site have also been proposed.

Keywords. Interface tracking; CFD; flow regimes; VOF.

1. Introduction

Most cities around the world have drainage systems to collect rain runoff, wastewater, and sewage. The ‘effluent’ flowing through these pipelines usually contains biological and chemical contaminants. Though, standard water treatment routines are thought sufficient to remove most of the biological contaminants in both drinking and wastewater [1], in 2003, it was a sewage leak that caused a SARS outbreak (a disease caused by a type of coronavirus SARS-CoV) in Hong Kong. Though, WHO has emphasised that wastewater should be treated in well-designed and well-managed centralized wastewater treatment plants, the incidence in 2003, has clearly established that the standard treatment protocols are not sufficient to remove infection from drain lines. Such contamination of drainages is indeed possible and this paper wishes to help mitigate the broader problem.

In general, viruses’ survival in water is found to be dependent on: (a) Temperature, (b) Light exposure, (c) Organic matter (adsorption of virus onto organic matter) and (d) Presence of antagonist microorganisms [2]. Organic matter tends to act as incubation zones for coronaviruses. One such organic matter, and a potential incubation zone for coronavirus inside pipelines, are ‘biofilms’. Biofilms are thin, slimy bacterial growths that line the pipes of many aging drinking water and sewage systems. These biofilms are found to release 58% of the bacteria in the distributed water [3]. Biofilms are also found to accumulate viruses [4]. Thus preventing biofilm growth on the walls of drain pipelines would to an extent help reduce virus and bacterial contaminants being present in drain pipelines. Simple hydrodynamic means to prevent biofilms’ growth, such as using fast flowing water, have been found ineffective [5] as biofilms can withstand high shear stresses especially in dead-ends. Traditional cleaning agents are also known to be ineffective and do not clean or sterilize microbes located in the inner layers of the biofilm [6]. A recent study has identified a class of fluids known as enzyme fluids [7, 8] to be effective for removing biofilms in pipes. These enzyme fluids are specifically being highlighted here, as these fluids are organic and thus will be safe for human use. In
principle, any cleaner fluid which is thought appropriate can also be used. However, any of these specialised cleaner fluids tend to be expensive and need to be used prudently. The best way to ensure minimal use of these fluids, and to achieve effective cleaning of large pipelines is to ensure that the cleaner fluid (CF) is transported effectively to the location of the biofilms, and allowed to be in contact with the biofilms for a sufficient amount of time so as to ensure its effective removal. The present study proposes to use ‘multiphase flow’ hydrodynamics to achieve the same, and to reduce the propensity of infection outbreak that can originate from biofilms in sewage pipelines.

1.1 Concept of flow regimes

Multiphase flows have fluids interacting among themselves and result in a wide array of flow configuration formations, known as flow regimes (for example, see tables 5-7). Out of all the flow regimes possible, certain flow regimes are more suited to efficiently transport cleaner-fluids to the biofilms. The temporal and spatial distribution of these flow regimes depend on the flow rates of individual phases, the configuration of the flow domain, and also on the properties of the fluids. Flow pattern maps of liquid-liquid systems, provide this information, however such flow pattern maps available in literature are currently limited in their applicability range. This paper, with the aim of recommending best flow conditions to achieve effective cleaning, has carried out simulations of liquid-liquid systems for various parametric ranges, and has thus helped expand the applicability range.

For the present work, using CFD simulations, we demonstrate that interfacial hydrodynamic concepts can be used to ensure precise transfer of cleaner fluid to tube walls, and also to ensure sufficient contact, in order for proper removal of biofilms. In particular, we show that high viscosity CFs, followed by medium viscosity CFs, are most suitable for cleaning purposes. In our simulations, the low viscosity fluids, which are traditionally thought best, are largely ineffective. This unexpected result is due to the fact that high viscosity flows have the tendency to retain their interfaces against the forces exerted by surrounding water and are easily transported to the pipe wall. The theory relies on interfacial hydrodynamics driven by the difference in viscosities and relative superficial velocities, and is therefore not limited to water and a specific cleaner fluid combination. In addition, we show that the computational technique used here can be an effective tool that can be used to predict the flow regime formation for drain pipeline flows, for a wide range of fluid combinations. To ensure that real-life conditions mimic what is predicted by simulations, one needs to closely monitor various parameters of the flow; and in this paper we also provide a list of fluid dynamic parameters and simple criterions that indicate adherence/non-adherence to expected flow regimes, and can be used to monitor the real field conditions. More broadly, our results suggest that high viscosity CFs are most suited for optimal cleaning of biofilms formed on pipeline walls. It is however noted that, high viscosity CFs require higher pumping power, and thus, in places where this is difficult to achieve, operating conditions for implementation with medium viscosity CFs have also been suggested. The method suggested is highly targeted and ensures minimal usage of cleaner-fluid.

2. Methodology

2.1 Choice of the technique

Flow regimes have different sizes/types of interfaces. Consequently, different mathematical techniques are required to capture it accurately. Dispersed flows have been traditionally solved using Eulerian two-fluid models. On the other hand, the interface tracking models have been developed for the modelling of flows having large distinctive interfaces between the phases [9, 10]. Although two fluid methods are the most suited for dispersion regimes, their applicability to mixed flows (with multiple scales) is not very accurate. Since both methods discussed above (two fluid and interface tracking) are being specific to a particular flow regime with regard to their applicability, attempts have been done to utilize a coupled approach for the modeling and simulation of multi-scale flows involving mixed regimes [11]. Here in these coupled methods, the multi-fluid approach and interface tracking approach were suitably selected for specific regimes where their application would be justified with respect to the accuracy acquired. When the interface scales are small enough to be resolved, the Eulerian multi-fluid dispersion models can be selected. Conversely, when the interface is larger than or comparable with the grid dimension, and the mesh is being resolved enough to capture the interface, the tracking method has to be brought in. The unified models formed by the coupling of both the models had to employ a criterion for switching between the different models [12–14]. The interface tracking methods, especially those implemented in the two fluid Eulerian frameworks are based on the solution of the volume fraction equation for volume of the fluid in the cells. Similar type of equation is solved for the implementation of VOF approach. Due to the complexity of the flow regimes being dealt with, and the wide range of possibilities, the coupled Multi-fluid VOF is considered the most suitable method to capture most of the flow regimes occurring in pipelines and hence is used in this study [15].

Though interface tracking is implemented using VOF, a suitable interface sharpening technique is required to properly resolve the interface. A geometric reconstruction method performs the interface sharpening by the discretization of advection part in the volume fraction equation. Since this reconstruction algorithm relies on the piece-wise linear approximation in the cells with a suitable slope, the method can only provide sharp interfaces [16]. High resolution
2.2 Multi-fluid VOF: two fluid model with interface capturing approach

The Eulerian–Eulerian model solves separate equations for mass and momentum equations for each phase. In addition to this, the Multi-fluid VOF model, solves an equation for the volume fraction in the computational cells similar to the VOF scheme. This improved version of Eulerian method is capable of capturing different flow regimes [18]. ANSYS FLUENT commercial platform has been used to carry out the simulations. The governing equations for a two-fluid model with two continuous phases, as well as the volume fraction equation is shown below.

Continuity equation for volume fraction: Considering the phase, \( \frac{\partial (\alpha_k \bar{u}_k)}{\partial t} + \nabla \cdot (\alpha_k \bar{u}_k \bar{u}_k) = 0 \) \( \tag{1} \)

For the primary phase, volume fraction is computed based on the equation: \( \sum_{k=1}^{n} \alpha_k = 1 \)

Continuity for phase:

\[ \frac{\partial (\rho \alpha \bar{u}_k)}{\partial t} + \nabla \cdot (\rho \alpha \bar{u}_k \bar{u}_k) = 0 \] \( \tag{2} \)

Momentum equation for phase:

\[ \frac{\partial (\rho \alpha_k \bar{u}_k \bar{u}_k)}{\partial t} + \nabla \cdot (\rho \alpha_k \bar{u}_k \bar{u}_k \bar{u}_k) = -\alpha_k \nabla p + \nabla \cdot \left( \frac{\alpha_k}{\rho \alpha_k} \bar{F} \right) \]

\[ + \rho \alpha_k \bar{g} + \bar{F}_{\text{Drag},k} + \bar{F}_{\text{CSF}} \] \( \tag{3} \)

\( \bar{F}_{\text{CSF}} \) is the phase stress-strain tensor, and \( p \) is the pressure shared by phases. Here, interfacial forces such as lift, virtual mass force, wall lubrication, and turbulent dispersion were neglected in the simulations, and only the drag force, was considered. Anisotropic drag law has been used for the basic Multi-fluid VOF model. It is most suited when there is an obvious difference in the drag imparted in the normal and tangential directions with respect to the interface, which normally is the case for the free surface flows. Under this anisotropy drag formulation, two different methods are available: based on the symmetric drag law and the second one utilizing the viscosity effects. Under this, the viscous drag component in the principal direction:

\[ K_p = (K_{\text{visc}})_p \alpha_i \alpha_j \] \( \tag{4} \)

\( \alpha_i \) and \( \alpha_j \) being the volume fractions of the \( i^{th} \) and \( j^{th} \) phases respectively. The viscous drag component in the principal direction is given by

\[ (K_{\text{visc}})_p = \frac{\mu}{k} \lambda_p \] \( \tag{5} \)

\( \lambda_p \) is the friction factor vector given in the principal direction. \( k \) is the length scale. \( \mu = \mu_i \alpha_i + \mu_j \alpha_j \), the mixture viscosity. The surface tension force is estimated as a Continuum Surface Force (CSF), following the work of [19]. An additional source term is appeared in the momentum equation on implementing this model for accommodating the surface tension force in the VOF calculations at the interface. Same/similar densities of both liquids are chosen since we want to prevent stratification based segregation. Other effects considered negligible in this case is wettability effects with the wall.

2.3 Problem statement

Charles [20] conducted experimental studies on oil-water flows and a wide range of useful data of hydrodynamic parameters were published. This was utilized by Shi [21] and many other researchers as a benchmark for the validation purpose of their respective models. Shi [21] has considered a T-junction tubular domain with dual inlets set up - one each for the CF and water fluids (CF in the horizontal branch and water in the vertical branch). Each branch is having 200 mm length and axial length extending up-to 8 meters downstream of the meeting junction. The simulation conducted in this present manuscript has been carried out against the data presented in this paper. A 2D approximation was employed. A 2D domain was deemed sufficient since the metering of most hydrodynamic parameters in the experiment being carried out in an ‘averaged’ approach, the 2D cases can present with equivalence for comparative validation without loss of generality. Model Parameters used in the simulation is given in table 1.
Boundary Conditions and Initialization of Domain

Here the superficial velocities of CF and water have been chosen as the inlet boundary conditions. But in the practical cases, it needs to be noted that the velocity profile would have already attained the developed stage sufficiently early before entering the domain of interest. Hence developed velocity profiles for CF and water have been generated by performing separate simulation in a rectangular channel for single fluid (of length large enough to cover the developing stage and ensuring developed profile condition at the outlet), and exporting the velocity profiles from outlet of that domain to the present domain to be used as regular boundary conditions in the normal simulation cases (Respective constant velocities have been used at the inlet of that rectangular channel to get corresponding developed profile at the outlet). These profiles are formed by suitably selecting the laminar/turbulent model for the single fluid based on the analysis of the Reynolds number. It was observed that for most of the studied cases, the CF velocity profile (horizontal) was laminar in nature and water profile (vertical) was turbulent in nature. The density difference of the fluids was observed to aid in a natural stratification of fluids during the flow development (gravity being applied in the negative Y direction) with the lighter phase occupying top region (close to upper wall) and the heavier one close to the bottom wall, below. Initialization of the domain has been tested individually using both the fluids—CF and water. But it could be observed that the solution generated was independent of the type of initialization (followed by patching with the fluid) carried out either with CF or with water. Hence water initialization has been followed for all the cases for uniformity.

2.4 Procedure of analysis, validation and mesh independence

The study has been performed for three different fluid systems consisting of three different CF viscosities—low, medium and high (table 1). This is being done to compare

Table 1. Model parameters for simulation.

| Details of the domain: | T-junction tubular domain with dual inlets, one each for the CF and water fluids (CF in the horizontal branch and water in the vertical branch). Each branch is having 200 mm length and axial length extending up-to 8 meters downstream of the meeting junction. The width is 26 mm. The results of flow pattern post the mixing junction is what is relevant to the focus of the study. So results of the flow pattern in the downstream pipe is presented
| Dual inlet with water from the vertical limb and CF horizontally
| Initialization of the domain with water (Although change of initialization with CF resulted in same flow pattern generation)
| Vertical limb is placed 200 mm away from the horizontal inlet section for CF entry

| Details of the mesh: | Structured mesh generated, Hexahedral mesh elements
| Wall refinement applied close to the walls with a bias factor 10

| Material Properties |
| Viscosity CF: 16.8 cP (Low) 612 cP (Medium) 5000 cP (High)
| Density of CF: 998 kg/m3 or 860 kg/m3 (based on experimental data point)
| Viscosity of water: 1 cP, Density: 998 kg/m3
| Surface Tension: 0.045 N/m as the constant parameter for CF (low viscosity)-water interaction, 0.036 N/m for (medium viscosity)-water interaction, and 0.02 N/m for (high viscosity)-water interaction

| Boundary Conditions |
| Superficial velocities of CF and water for inlets
| Outlet is provided with pressure-outlet boundary (fluids ejecting to atmospheric)
| No slip boundary conditions for walls. No wall contact angle is considered.

| Model features |
| Eulerian Multifluid multiphase model which solves momentum equations for individual fluids in every computational cell and along with a suitable interface sharpening scheme
| Implicit method (due to the offer of unconditional stability)
| Anisotropic drag for fluids interaction
| Continuum Surface Force Surface tension model
| Turbulence is modeled using k-w SST model

| Discretization schemes |
| P-V coupling: SIMPLE
| Gradient: Least squares cell based
| Volume fraction: Geo-Reconstruct
| Momentum, Turbulence: Second order

Sādhana (2022) 47:228
the flow evolution features (including the regime of flow and other quantitative flow parameters like velocities of individual phases, etc.) as the CF viscosity varies. Table 2 shows the different cases considered including the boundary conditions (superficial velocities of CF and water), multiphase model chosen and the viscosity of the CF selected.

To make the results generated independent of the number of meshes (or no of elements) utilized, a grid independency study has been carried out for different mesh cases (from coarse to finer). Optimum number of meshes for the case has been identified by comparing the variation of some hydrodynamic flow parameter like pressure, volume fraction in the axial direction for different mesh cases. Below table 3 shows the various meshes generated along with mesh sizing parameters utilized.

Below figures 1 and 2 show the sample screenshots of the Mesh cases-III and IV used for the study.

| Case No. | Viscosity of CF (cP) | Multiphase model | Superficial velocity of CF (m/s) | Superficial velocity of water (m/s) |
|----------|----------------------|-------------------|----------------------------------|-----------------------------------|
| L-1      | 16.8                 | Multi-fluid VOF   | 0.015                            | 0.03                              |
| L-2      | 16.8                 | Multi-fluid VOF   | 0.06                             | 0.03                              |
| L-3      | 16.8                 | Multi-fluid VOF   | 0.15                             | 0.03                              |
| L-4      | 16.8                 | Multi-fluid VOF   | 0.048                            | 0.244                             |
| L-5      | 16.8                 | Multi-fluid VOF   | 0.244                            | 0.244                             |
| L-6      | 16.8                 | Multi-fluid VOF   | 0.487                            | 0.244                             |
| L-7      | 16.8                 | Multi-fluid VOF   | 0.055                            | 0.55                              |
| L-8      | 16.8                 | Multi-fluid VOF   | 0.274                            | 0.55                              |
| L-9      | 16.8                 | Multi-fluid VOF   | 0.55                             | 0.55                              |
| L-10     | 16.8                 | Multi-fluid VOF   | 1.1                              | 0.55                              |
| H-1      | 5000                 | Multi-fluid VOF   | 0.06                             | 0.23                              |
| H-2      | 5000                 | Multi-fluid VOF   | 0.10                             | 0.05                              |
| H-4      | 5000                 | Multi-fluid VOF   | 0.12                             | 0.18                              |
| H-6      | 5000                 | Multi-fluid VOF   | 0.12                             | 0.61                              |
| H-7      | 5000                 | Multi-fluid VOF   | 0.12                             | 0.81                              |
| H-15     | 5000                 | Multi-fluid VOF   | 0.40                             | 0.40                              |
| H-17     | 5000                 | Multi-fluid VOF   | 0.40                             | 0.80                              |
| M-1      | 612                  | Multi-fluid VOF   | 0.015                            | 0.03                              |
| M-2      | 612                  | Multi-fluid VOF   | 0.05                             | 0.1                                |
| M-3      | 612                  | Multi-fluid VOF   | 0.25                             | 0.1                                |
| M-4      | 612                  | Multi-fluid VOF   | 0.25                             | 0.25                              |
| M-8      | 612                  | Multi-fluid VOF   | 0.274                            | 0.55                              |
| M-9      | 612                  | Multi-fluid VOF   | 0.06                             | 0.03                              |

For the different sets with varying number of mesh elements (from coarser to finer), the flow parameters corresponding to Case L1 (low viscous CF-water, superficial CF velocity: 0.015 m/s and water velocity: 0.03 m/s) have been adopted for the simulations. The results obtained for the particular constant flow time (since being transient flow—here 38 sec is the flow time reached by each set of simulation) have been compared to identify the optimum grid set which saves considerable computational time without a compromise on the accuracy. Table 4 shows the various hydrodynamic parameters, that were compared against the experimental data obtained from Charles [20]. The major flow parameter being compared here is the pressure gradient in the axial direction (the direction of flow). The data extracted from the experimental paper is listed alongside the pressure gradient values obtained from the simulation. The pressure gradient taken from the simulation here is an averaged figure for the duration of flow in which the flow more or less appeared to be stabilized.

The table shows the values of pressure gradient and water holdup calculated for the meshes considered. This data should be read along with the graphs shown nearby. The graphs (figure 3) represent the variation of averaged pressure (in the axial direction) and the averaged CF fraction for different mesh cases. On comparison, the absolute values for the different cross sections in the domain have been found to nearly coincide for the mesh cases Mesh-III (3.89 L elements), Mesh-IV (5.79 L elements) and Mesh-V (7.29 L elements). For the lower mesh sizes (coarser) the
deviation in these values was large, and this deviation from Mesh-III onwards was considerably small for finer meshes. By comparing other validation data also (pressure gradient and water holdup), Mesh-III was giving sufficiently accurate results with no appreciable gain being expected in terms of accuracy for Mesh-IV and Mesh-V. Hence for the purpose of facilitating remaining simulations, Mesh-III with a total of 3.89 lakh hexahedral elements has been chosen as the optimum one. Based on this study the remaining simulations will be done using Mesh-3 with a total of 3.89 lakh hexahedral elements which is considered as the optimum one.

3. Results

Considering that a cleaner-fluid (CF) would already have the chemical properties to remove the biofilms on the pipeline walls, the challenge is to ensure ‘efficient’ and ‘sufficient’ contact between the cleaner-fluid and the pipeline walls. This is clearly a fluid dynamics problem, since one particular flow regime would maximise the contact while another might not. Achieving a particular flow regime, depends on the relative fluid properties, which in this case is sewage water and cleaner-fluid. Sewage water is assumed to have properties close to water and is an

![Figure 1. Sample Screenshot of Mesh-III: 3.89 lakh elements.](image1)

![Figure 2. Sample Screenshot of Mesh-IV: 5.79 lakh elements.](image2)
invariable factor in all the combinations investigated. So the problem now depends on the cleaner-fluid property, and the single most dominant factor is its viscosity. Experimental data of liquid-liquid flows of low viscosity and high viscosity fluids is available. However, medium viscosity ranges are less explored. Thus, flow regimes achievable for medium viscosity is unknown. In this study, following the demonstration that the simulation convincingly captures the liquid-liquid flows at both high and low viscosity CFs, the middle viscosity range was also simulated, to ensure completeness of data.

3.1 Flow morphology

Here, we present the flow morphology obtained using simulations for low, medium, and high viscosity CF cases. In some cases where experimental observations of the flow regimes are available, the simulation results have been compared with them. Tables 5, 6 and 7 present the flow regimes obtained for low, medium and high viscosity cleaner-fluids respectively. It was observed that the flow patterns matched expected trend for most cases except when the water velocities (sewage) were low. In low water velocity cases, we know that, quantitatively, the average void fraction across a cross section is captured reasonably well by the simulation, and it the spatial distribution that is not captured accurately. This mismatch is attributed to the inability of the interfacial force models used in the CFD simulation, which are essentially empirical models, to provide the correct force quantities, when it falls below the model’s applicability range. The exercise of tuning these models for improving flow regime predictions at low water velocity cases is not carried out here since water velocity in sewage lines tend to be of the medium to high velocity ranges. It is observed from the simulations that two major parameters govern flow patterns: (a) the magnitude of superficial velocity ($s_{vel}$) of CF and water (low, medium, high) (b) their ratio R ($R = \frac{CF\ s_{vel}}{water\ s_{vel}}\ R > 1, =1, <1$).

| Case L1 with different meshes | Average Pressure gradient in the flow direction (Pa/m) | Experimental Pressure gradient for Case L1 | Average CF fraction in the flow direction | Average water holdup |
|------------------------------|---------------------------------|---------------------------------|---------------------------------|------------------|
| Mesh-I: 1.91 lakh elements   | 12.2                            | 10.2                            | 0.387                           | 0.613            |
| Mesh-II: 2.45 lakh elements  | 11.7                            | 10.2                            | 0.39                            | 0.61             |
| Mesh-III: 3.89 lakh elements | 11.2                            | 10.2                            | 0.411                           | 0.589            |
| Mesh-IV: 5.79 lakh elements  | 10.7                            | 10.2                            | 0.415                           | 0.585            |

Figure 3. Variation of (a) averaged pressure and (b) averaged CF fraction.
### Table 5. Comparison of Flow Patterns for Low viscosity CF and water flow.

| Case No. | Case L1 | Case L2 | Case L3 | Case L4 |
|----------|---------|---------|---------|---------|
| Superficial velocity of CF (m/s) | 0.015 (low) | 0.06 (low) | 0.15 (medium) | 0.048 (low) |
| Superficial velocity of water (m/s) | 0.03 (low) | 0.03 (low) | 0.03 (low) | 0.244 (medium) |
| $R = \frac{CF_{s_{vel}}}{water_{s_{vel}}}$ | 0.5 | 2 | 5 | 0.2 |
| Observed flow pattern in the experiments | CF bubbles entrained in water | CF Plugs in water | Thin liquid film close to walls, similar to CAF | CF bubbles entrained in water |
| Observed flow pattern in the CFD simulation | CF plugs (large bubbles) | CF Plugs observed. But didn’t separate from the stratified flow | CF slugs and separation from tail end of slugs to small bubbles of CF in water | CF bubbles entrained in water |

| Case No. | Case L5 | Case L6 | Case L8 |
|----------|---------|---------|---------|
| Superficial velocity of CF (m/s) | 0.244 (medium) | 0.487 (high) | 0.274 (medium) |
| Superficial velocity of water (m/s) | 0.244 (medium) | 0.244 (medium) | 0.55 (high) |
| $R = \frac{CF_{s_{vel}}}{water_{s_{vel}}}$ | 1 | 2 | 0.5 |
| Observed flow pattern in the experiments | CF Plugs in water | Thin liquid film formation close to walls, similar to CAF | CF Plugs in water |
| Observed flow pattern in the CFD simulation | Large CF plugs (more like slugs, with bridging of pipe section) observed | Dispersion of water in CF slugs, later observed film development close to walls | Large bubbles (no defined plugs) of CF (more like dispersion) in water matrix |
Superficial velocity is a hypothetical flow velocity calculated as if the given phase or fluid were the only one flowing or present in a given cross sectional area. This is the common metric of reporting operating conditions in multiphase flows. Based on the simulations conducted (tables 5, 6, 7), the following conclusions can be drawn.

For Low Viscosity CFs: When both water and CF velocities are low, CF forms plugs. If CF is marginally faster, and if both phases are moving with low to medium velocities, then it forms longer plugs. With increasing difference in velocity (water velocity increasing), the plugs get deformed and elongated as water tends to exert higher force on the interfaces of plugs. At higher velocities, intermittent slugs of CF are formed i.e. slugs that fill the entire pipeline can be seen.

For medium viscosity CFs: For low velocities, and with CF flowing slower than water, larger CF, plugs can be seen as CF has higher viscosity and can retain shape better. With increasing velocities of water, the plugs tend to get deformed, but the increased viscosity of CF enables it to counter this near the wall where the velocity of water is lower. Thus, water successfully disperses CF finely in the entire medium, but at the walls a fine CF film is formed. For cases where CF velocity is more than water velocity water plugs are formed instead of CF plugs.

For high viscosity CFs: Due to high viscosity, CF interfaces remain intact closer to the walls. Thus, a coating of CF is constantly present at the wall. This is more pronounced at medium velocities of water. At high water velocities, water disperses CF finely in the entire medium. If CF velocity exceeds water velocities in these conditions then a mix of dispersion and slugs with wavy interface are seen. The understanding of how to apply these aforementioned observations will be discussed in the discussion section.

3.2 Generic trends of macroscopic parameters

Visual verification of flow regimes is not possible in sewage pipelines. Thus, it would be essential to monitor macroscopic parameters at various locations to verify the flow regime that is actually occurring. Since we have validated the regimes predicted by CFD wherever data was available, in the previous section. In this section, we will study flow property variation as predicted by CFD in the different flow regimes, and see if the trend can be used as an indicator of a specific flow regime. This would then enable engineers to measure and monitor these parameters and consequently identify the flow regime that is occurring. To this end, this section presents certain unique trends observed in the axial variation of pressure, CF fraction, and CF velocity which might be monitored to suitably identify the flow regime. How these can be useful is discussed in the discussion section.

Pressure: It was observed that for a few lower viscosity cases, the injected CF does not retain its trajectory and eventually becomes a dispersed flow regime (e.g. L4 in table 5). It was observed that at the location where the CF becomes dispersed, the pressure value in the pipeline dips towards the negative side, creating a vacuum pressure condition at the section (figure 4a). The next section clearly indicates that, with slugs and plugs the velocities fluctuate due to the inherent difference in phasic velocities. In a dispersed regime, such fluctuations disappear and so do related pressure fluctuations. This stabilization of the dispersed regime appears to help the smooth falling of the pressure. However, the dispersed flow results in a steep pressure drop typically higher than other flow regime (also observed in Chakrabarti [22]) Meanwhile, for cases L1 and H1, plots have been presented for comparison of variation of pressure along the central axis in the flow direction (figure 4b, c). For the plug type flows as encountered in the case L1, the axial variation of pressure fluctuates suggesting a higher localized pressure due to the onset of plugs. For H1, which is a combination of an intact film flow near the wall with wavy interface and some dispersed drops in the mean flow, the pressure drop is significant and also demonstrates small fluctuations corresponding of the webby structure of the interface. Thus, a smooth (and significant drop) or fluctuating trend of pressure is an indicator of certain flow regimes.

CF fraction and CF velocity: For many of the cases it could be observed that the trends for CF fraction and CF velocity were morphologically similar. Severe fluctuation could be observed in the L3 case (figures 5a, b), wherein slugs of CF were observed as the dominant flow feature. So generally speaking, the distribution of large bubble-like plugs or slugs could produce severe fluctuations in the velocity of CF inside the domain. As the flow regimes transformed to the dispersion patterns, the velocity variations were observed to attain a quasi-stable state with little fluctuations. This implies that for a uniformly dispersed flow regime, the velocity fluctuations can be minimum although the absolute magnitude of velocities may be higher. Similar plots can be observed for the flow regime comprising of wavy stratified along with the dispersion of CF in water. Here, although there was a considerable oscillation (owing to the impact of wavy stratified structures) in the volume fraction plot, the CF velocity stood nearly a constant in the axial direction, even though the value was higher than the inlet boundary condition velocity value (figure 5c, d). Another significant observation was that, for higher inlet superficial velocities, the velocity values at different sections in the domain were reaching values higher than the inlet values. The larger velocities of CF plugs could be attributed to the lesser share of cross-section available for flow leading to a nozzle-like effect causing acceleration. Oscillatory nature of the volume fraction graphs with higher frequency and lower amplitude could also be correlated with bubbly flows while large
amplitude small frequency oscillations of volume fraction graphs represented plug or slug flows. The utility of these observations are discussed in the next section.

4. Discussion

Literature states that biofilms that grow in sewage pipelines are potential locations where bacteria and virus can aggregate. Through this work, we demonstrate that interfacial hydrodynamic concepts can be used to ensure removal of biofilms formed on pipeline walls. The method suggested is target specific i.e. it transports cleaner-fluid right to the walls of the pipeline, but at the same time ensures effective usage of cleaner fluid with minimal wastage.

First and foremost, the present study has tested the CFD technique of Multi-fluid VOF and has demonstrated that it is a reliable tool that can be used to predict different flow regime formations, for multiphase horizontal flows in pipelines. This method has been shown to have better accuracy when compared with other popular interface capturing multiphase techniques whenever a wide range of flow regimes need to be predicted. This is a one-of-a-kind simulation catering to a range of fluid viscosity ranging right from high to low.

The main result of our numerical simulations is that we are able to recommend superficial velocities at which cleaner fluids need to be pumped into the sewage line, so as to achieve flow regimes that ensure the delivery of cleaner fluid to the periphery of the tube walls. Since the biofilms only grow on the walls of the pipelines, it is only efficient and effective, if cleaner fluids that are injected into the pipeline move towards the walls of the pipeline and not be dispersed in the main flow. Due to the uncertainty of their location chemical reaction-based removal will only be effective if cleaner fluid is always in contact with the wall throughout the length of the pipeline. The idea here is to ensure that the flow regime is that of a continuous flow of cleaner fluid at the wall, and this paper does use interfacial hydrodynamics. Our simulations indicate that most effective cleaning can be achieved by using a cleaner fluid with a high viscosity (≈5000 cP). In such cases, a low-medium velocity (≈0.05-0.3 m/s) of CF and water would ensure that the cleaner fluids are in constant contact with the pipe walls (figure 6a). On the other hand, injecting cleaner fluids at very low superficial velocities (≈0.03 m/s) is not recommended as the thickness of the film at the wall becomes very thin (figure 6b); consequently, this should be used only when the pipes are either new or in cases where cleaning cycles are usually very frequent.

When using medium viscosity CFs (≈600 cP), the flow configuration as shown in figure 3c, can be achieved; but such a flow regime can be achieved only when injecting cleanerfluid at medium superficial velocity into a pipeline.
| Case No. | Case H1 | Case H2 | Case H4 | Case H6 | Case H7 | Case H15 |
|---------|---------|---------|---------|---------|---------|---------|
| Superficial velocity of CF (m/s) | 0.06 (low) | 0.1 (medium) | 0.12 (medium) | 0.12 (medium) | 0.12 (medium) | 0.4 (high) |
| Superficial velocity of water (m/s) | 0.23 (medium) | 0.05 (low) | 0.18 (medium) | 0.61 (high) | 0.81 (high) | 0.4 (high) |
| \( R = \frac{CF_{s_{vel}}}{water_{s_{vel}}} \) | 0.26 | 2 | 0.66 | 0.2 | 0.15 | 1 |
| Observed flow pattern in the experiments | Inversion of flow | Concentric Annular flow | Lumps of CF dispersed in water | Lumps of CF dispersed in water | CAF |
| Observed flow pattern in the CFD simulation | CAF flow, but an appreciable communication between top and bottom films through webby structures, seeming like a kind of entrapped water plugs in the webs of CF films | Concentric Annular Flow | Thin wafer type films for the CAF flow. Later transformed to CF dispersion | Large wavy stratified, with CF dispersion | CAF |
where waste water is flowing at a high superficial velocity. This is a very narrow restrictive range of operation. Moreover, this flow pattern would require large volumes of CF, since a lot of the cleaner fluid is also present in the core flow. This might be suitable for cleaning effluents from highly contaminated regions such as hospitals, where disinfecting the water in the core is also essential.

When using low viscosity CFs (~16 cP), the best possible flow pattern that can be obtained is that of slug flow (figure 6d). This pattern is achievable when injecting cleaner-fluid at medium-high velocity into a pipe where the sewage water is moving at a low-medium velocity. In these cases, there is cyclic exposure of tube to waste water and cleaner-fluid. It also has to be pointed out that when using low viscosity cleaner fluids, one must avoid injecting them at very low superficial velocities, especially when the waste water is flowing at a high velocity. This might occur inadvertently in monsoon seasons when the sewage flow is quite large and the usual injection velocity will be too low in comparison. The effect of the large difference in the relative velocity causes the cleaner-fluid to completely disperse into the medium and does not provide effective cleaning (figure 6e).

Real life conditions might not adhere to expected behaviour. Thus, there is a need to closely monitor the flow occurring in the pipeline. Since flow regime visualisation is not possible in pipelines, measuring devices need to be used at regular intervals along the length of the pipeline to measure the macroscopic parameters’ variation. The study also provides recommendations of what parameters to monitor on site to ensure adherence to expected patterns and can be applied in a wide range of conditions. Our numerical simulations show that pressure, cleaner fluid fraction, and its velocity are important indicators of the flow pattern existing within the pipe. Our simulations have also shown that the flow regimes that need to be avoided are dispersion regimes, and regimes where cleaner fluid is distributed in the medium in the form of bubbles, or plugs. Uneven stratification also is not desired since that exposes only one part of the pipeline circumference to the cleaner-fluid.

Based on our results, we have been able to identify the trends of macroscopic parameters which indicate that the aforementioned undesirable regimes are occurring. For example, If the average pressure at a cross-section were to change from positive to negative, anywhere along the pipeline, then it implies that the flow has transitioned into a dispersion regime. If, simultaneously, the CF fraction and velocity, have large oscillations then the flow is stratified. However, if the oscillation is small, then it indicates

Figure 4. (a) Axial variation of sectional averaged pressure for Case L4. The Pressure becomes negative indicating transition to a dispersed flow regime. (b) Fluctuations observed in axial variation of linear (longitudinal central axis) pressure for Case L1 and is indicative of plug flow (c) Axial variation of linear (longitudinal central axis) pressure for Case H1.
occurrence of bubbly flow regime. Slug and plug type flow regimes can be identified when the CF fraction and velocity values will fluctuate with large amplitude and smaller frequencies. In some slug and plug type flow regimes, cleaner fluid velocities might become larger than the inlet values anywhere along the length of the pipeline.

It is important to note that the theory presented here is based on certain important assumptions. Firstly, biofilm is not explicitly modelled in the CFD model. This is because no specific understanding of the common deposit size of biofilms is available in literature. Since the interaction between the cleaner fluid and the biofilm is expected to be chemical in nature, it is assumed in this work that if the cleaner fluid remains in touch with the wall then more effective removal can be ensured. Hence, in this work, it was thought best to capture how the cleaner film volume fraction will distribute across the pipe cross-section; and flow patterns wherein maximum cleaner fluid distribution near the walls is observed is recommended as solutions for removal of biofilms.

Relating to the properties of the cleaner fluids, the foremost assumption made in this paper is that the density of the different CF fluids is same as that of water. This assumption is done on the basis of the physical requirement that we would want to prevent gravity driven stratification of CF and may be achieved is by adding chemical additives. Another important assumption has been made with regard to the surface tension of the CF. For this work, surface tension corresponding to oils with viscosity values same as the ones used in this study have been used. Other auxiliary assumptions are: (i) chemical properties of the cleaner fluid and waste water is not significant enough to modify flow features. (ii) Waste water is taken to have pure water properties. (iii) Presence of solid and gas phases are assumed to have negligible effect on the flow. (v) Large parts of the waste system are pipelines and not open channels; a valid assumption especially in cities. Our results, with the aforementioned specific CF-fluid property combinations of surface tension, viscosity and density, have been tested over a range of operating conditions and since the results are presented in terms of superficial velocities, these can be extended to pipelines of different sizes and different velocities. One way to apply the findings of this study, is by using chemical additives similar to the ones suggested above, to ensure that the chosen CF has properties similar to the ones used in this study. Alternatively, the simulation methodology used this study can be quickly used, with a given set of properties of the CF, and

Figure 5. (a) Axial variation of sectional averaged CF fraction for Case L3. (b) Axial variation of sectional averaged CF velocity for Case L3. Large fluctuations in both (a) and (b) are indicative of large slugs in the pipeline. (c) Axial variation of sectional averaged CF fraction for Case H-15. (d) Axial variation of sectional averaged CF velocity for Case H-15. Here, fluctuations in (c) and the absence of those in (d) is indicative of wavy stratified flow in the presence of smaller dispersed bubbles.
corresponding flow conditions that will ensure wetting of the walls by the CF, can be determined. Simulation using the method suggested in this paper will also greatly help in estimating flow behaviour when the geometry is scaled up and that too at a minimal cost. The validity of the theory discussed here can also be tested quite easily by conducting a preliminary test at any local municipal facility, house, building or hospital.

5. Conclusion

Drainage systems contain biological contaminants like bacteria and viruses flowing through them, and they tend to adhere to biofilms that grow on the walls. Virus can adhere to these biofilms and hence cause an outbreak of the disease through these drainage systems. The present work, using CFD simulations, demonstrates that interfacial hydrodynamic concepts can be used to ensure precise transfer of cleaner fluid to tube walls for the proper removal of biofilms. From our simulations, the low viscosity cleaner fluids are largely found to be ineffective as they are unable to retain their interfaces so as to be able to be transported to the wall. They disperse into smaller drops. High viscosity CFs are found most suited, however they require higher pumping power. We are able to show that high viscosity CFs, followed by medium viscosity CFs, are most suitable for cleaning purposes.

The importance of our result stems from the fact that it permits easy implementation in developing countries using minimal resources/equipment. This method also helps to minimise cleaner fluid usage in order to restrict any environmental impacts. Our result have a wider range of applicability than just applying it to waste water treatment. Since, it is purely defined by fluid viscosity and fluid properties, it can be extended to address 'clean drinking water' problems. The solution proposed is very simple and can be implemented at every housing/building and society level. Thus, our results offer an important new direction of research for interfacial hydrodynamics as a means for addressing different social problems at industrial levels. We offer an easy, affordable and readily implementable solution to curtail the possible outbreak of infection due to unclean drainage systems.

**Nomenclature**

**General symbol**
- \( F \) Force
- \( g \) Gravity
- \( K \) Coefficient of principle direction
- \( l \) Length scale
- \( p \) Pressure
- \( t \) Time
- \( u \) Velocity

**Greek symbols**
- \( \alpha \) Volume fraction of component ‘k’
- \( \rho \) Density
- \( \tau \) Shear stress
- \( \eta \) Viscosity

**Figure 6.** Flow pattern observed for (a) H2, (b) H1, (c) M8, (d) L3 and (e) L7. Red here is indicative of the cleaner fluid, while blue is indicative of the water effluent.
Friction factor

Subscript

k Indicates components
i Indices
j Indices

References

[1] WHO Global 2020 Water, sanitation, hygiene, and waste management for the COVID-19 virus: interim guidance, WHO/2019-nCoV/IPC_WASH/2020.3.https://www.who.int/publications/i/item/water-sanitation-hygiene-and-waste-management-for-covid-19

[2] Gundy P M, Gerba C P and Pepper I L 2008 Survival of Coronaviruses in Water and Wastewater. Food Environment Virology 1(1): 10–14

[3] Chan S, Pullerits K, Keucken A, Persson K M, Paul C J and Rådstro¨ m P 2019 Bacterial release from pipe biofilm in a full-scale drinking water distribution system. NPJ Biofilms and Microbiomes 5: 9

[4] Skraber S, Schijven J, Gantzer C and De Roda Husman A M 2005 Pathogenic viruses in drinking-water biofilms: a public health risk? Biofilms 2(2): 105-117

[5] Simuniˇ c U, Pipp P, Dular M and Stopar D 2020 The limitations of hydrodynamic removal of biofilms from the dead-ends in a model drinking water distribution system. Water Research 178: 115838

[6] Li W, Zheng T, Mac Y and Liu J 2020 Fungi characteristics of biofilms from sewage and greywater in small diameter gravity sewers. Environmental Science: Water Research and Technology 6: 532–539

[7] Liu X, Tang B, Gu Q and Yu X 2014 Elimination of the formation of biofilm in industrial pipes using enzyme cleaning technique. MethodsX 1: 130–136

[8] Stiefel P, Mauerhofer S, Schneider J, Maniura-Weber K, Rosenberg U and Ren Q Enzymes Enhance Biofilm Removal Efficiency of Cleaners. Antimicrobial Agents and Chemotherapy 60: 3647–3652

[9] Cerne G, Petelin S and Tiselj I 2001 Coupling of the interface tracking and the two fluid models for the simulation of incompressible two-phase flow. Journal of Computational Physics 171: 776–804

[10] Tryggyason G, Bunner B, Esmaeeli A, Juric D, Al-Rawahi N, Tauber W, Han J, Nas S and Jan Y J 2001 A Front tracking method for the computations of multiphase flows. Journal of Computational Physics 169(2): 708–759

[11] Yan K and Che D F 2009 A coupled model for simulation of the gas-liquid two-phase flow with complex flow patterns flow patterns. International Journal of Multiphase Flow 36(4): 333–348

[12] Frank T 2005 Numerical Simulation of Slug Flow Regime for an Air-Water two phase flow in horizontal pipes. In: The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11)

[13] Höhne T and Vallée C 2010 Experiments and numerical simulations of horizontal two-phase flow regimes using an interfacial area density model. Journal of Computational Physics 2: 131–143

[14] Bartosiewicz Y, Seynhaeve J M, Vallée C, Höhne T and Laviéville J 2010 Modeling free surface flows relevant to a PTS scenario: comparison between experimental data and three RANS based CFD codes. Comments on the CFD-experiment integration and best practice guideline. Nuclear Engineering and Design 240: 2375–2381

[15] Chen G, Wang Q and He S 2019 Assessment of an Eulerian multi-fluid VOF model for simulation of multiphase flow in an industrial Ruhrstahl-Heraeus degasser. Metallurgical Research & Technology, 116: 6, id.617, 10

[16] Rider W J and Kothe D B 1998 Reconstructing Volume Tracking. Journal of Computational Physics 141(2): 112–152

[17] Wacławczyk T and Koronowicz T 2008 Comparison of CICSAM and HRIC high-resolution schemes for interface capturing. Journal of Theoretical and Applied Mechanics 46(2): 325–345

[18] ANSYS-FLUENT R14.0, Theory Guide, 2011

[19] Brackbill J U, Kothe D B and Zemach C C 1992 A continuum method for modelling surface tension. Journal of Computational Physics 100: 335–354

[20] Charles M E, Govier G W and Hodgson G W 1961 The Horizontal Pipeline Flow of Equal Density Oil-Water Mixtures. The Canadian Journal of Chemical Engineering 27–36

[21] Shi J 2012-2015 A Study On High-Viscosity Oil-Water Two-Phase Flow In Horizontal Pipes, Cranfield University, Cranfield, PhD Academic Year: 2012–2015

[22] Chakrabarti D P, Das G and Ray S 2005 Pressure Drop in Liquid-liquid Two Phase Horizontal Flow: Experiment and Prediction. Chemical Engineering and Technology 1003–1005