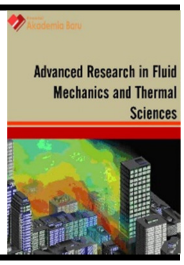
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



The application of multiphase DEM for the prediction of fat, oil and grease (FOG) deposition in sewer pipe lines

Open Access

Iman AF Husain[1], Ma'an Fahmi Alkhatib[1,*], Mohammed Saedi Jami [1], Mohamed E.S Mirghani[2], Zaki Zainudin[1], Asif Hoda[3]

[1] Bioenvironmental Engineering Research Center (BERC), Kulliyah of Engineering, International Islamic University Malaysia, Kuala Lumpur, Malaysia
[2] International Institute for Halal Research and Training (INHART), International Islamic University Malaysia, Kuala Lumpur, Malaysia
[3] Mechanical Engineering Faculty, Jubail University College, Jubail Industrial City, Kingdom of Saudi Arabia

ARTICLE INFO

Article history:
Received 2 July 2016
Received in revised form 1 February 2017
Accepted 4 August 2017
Available online 13 November 2017

ABSTRACT

Fat oil and grease (FOG) deposition into sewer pipes can block the pipes and restrict the wastewater flow causing backflows and sanitary sewer overflows (SSOs). Understanding the wastewater flow and transport of FOG particles is a key step for predicting the particles deposition and blockage formation. ANSYS FLUENT was used for simulating the flow of FOG particles and its deposition onto the sewer pipe. The multiphase Eularian-Lagrangian model with discrete Phase method (DPM) was utilized for developing the CFD model. The kinetic parameters and physical values are based on previous experimental work and literature. The CFD Eularian-DEM multiphase model has shown a good potential for simulating the wastewater flow and demonstrated the applicability of CFD to simulate and track the transport and deposition of FOG particles into the sewer pipe walls.

Keywords:
ANSYS FLUENT, Multiphase, Eulerian model, DPM, Deposition, FOG, sewer pipe

1. Introduction

CFD is maturing into a powerful and pervasive tool with each solution representing a rich tapestry of mathematical physics, numerical methods, user interfaces and state-of-art visualization techniques. In its present-day form, it can be used to efficiently quantify the complex dynamic processes that occur during fluid motion. Among the processes quantifiable by CFD are fluid flow, phase change, solid and fluid interactions, prediction of solid stress and chemical reactions. Engineers may apply both experimental and CFD analyses which complement each other. For example, engineers may obtain global properties such as lift, drag, pressure drop, or power experimentally but use CFD to obtain details about the flow field such as shear stress, velocity, and pressure profiles. Experimental data are also used to validate CFD solutions by comparing the computationally and experimentally determined global quantities. In such cases, CFD is employed

* *Corresponding author.*
E-mail address: maan@iium.edu.my (Ma'an Fahmi Alkhatib)

to shorten the design cycle through controlled parametric studies thereby reducing amount of experimental testing [1].

The development of both numerical techniques and digital machines with increasing computational power allowed in the last decades a wide application of CFD methods to many areas of fluid dynamics including Environmental Fluid Mechanics (EFM). Cushman-Rosin & Gualtieri defined environmental fluid mechanics as the scientific study of naturally occurring fluid flows or air and water on our planet Earth, especially those that affect the environmental quality of air and water [2]. Within EFM, for many years CFD methods have found a wide application in the analysis of natural water systems, such as rivers, lakes, estuaries and coastal waters. Examples of such applications include the study of the flow and of the transport and mixing of contaminants and sediments within those systems. Also, CFD methods have been applied in wastewater engineering. Some of the applications include analysis of combined sewer systems overflows [3], evaluation of the efficiency of activated sludge reactor [4] design of channel hydraulic flocculators [5], evaluation of disinfection calculation methods [6], municipal sewer system design [7] and modeling of solid transport [8]. All commercial CFD software for single-phase fluid are performed in the Eularian framework, however, stochastic Lagrangian tracking calculation has been employed for particles tracking in multiphase simulations [9].

Fat oil and grease (FOG) deposition into sewer pipes have been an ongoing concern to wastewater municipalities [10]. The understanding of FOG particles transport and deposition into sewer pipes is crucial for the prediction of the pipe blockages and hence, avoiding the sewer overflows. One way to study FOG deposition is through developing a computational model for simulating the flow and deposition formation.

In this paper ANSYS FLUENT 15.0 was used for the modeling and simulation of FOG particles deposition into the walls of the domestic sewer pipes using Eularian-DEM mehod.

2. Simulation Detals

A. Case investigated

Wastewater flow by gravity force in an inclined pipe was investigated. The pipe is circular pipe with 0.1016 m diameter and 1.5 meter length. The flow consists of wastewater as the main phase with the FOG particles as the secondary phased dispersed within the primary wastewater phase. Fig. 1 shows the description of the studied case.

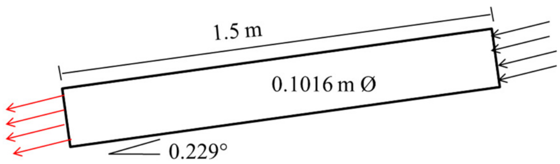


Fig. 1. Description of the simulated pipe

B. CFD model

An analysis system using FLUENT was created in ANSYS Workbench 15. This analysis system is a ready-made stencil that includes all the individual systems or applications needed for the CFD analyses; in this case it includes geometry creation, mesh generation, solver set-up and postprocessor.

A top-down approach was followed for creating the computational domain by using the sweep logical operation on cylindrical primitive shape. A cylindrical shaped pipe with 0.1016 m diameter and 1.5 m length was created using ANSYS DesignModeller.

TGrid meshing software was used for creating the computational mesh (Fig. 2). Multi zone meshing was used by creating a wall-boundary, inlet, and outlet and fluid zones using tetrahedral mesh. Tetrahedral meshes are composed of four-sided triangle-faced elements which can be largely automated but the current mesh generation software [11]. The generated mesh consists of 284620 elements with 15 layers of inflation to improve the calculations near the wall.

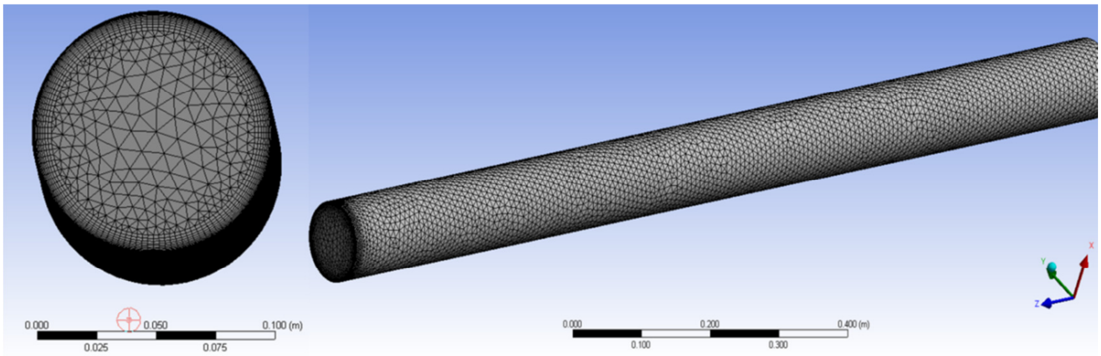


Fig. 2. The resulted tetrahedral mesh with inflation

The solver was set to pressure-based with steady state analysis. The absolute velocity formulation was selected for this analysis. Multiphase Eulerian model was chosen for analyzing the wastewater/ FOG flow. This model employs Eulerian-Lagrangian method. The Eulerian phase (primary phase) is the wastewater while the dense discrete phase (secondary phase) is the FOG particles which are studied using lagragian approach. Implicit scheme was selected for the volume fraction parameters. The DPM particles were injected from the inlet at 0.02 kg/s flow rate with ten different diameters using rosin-rammler diameter distribution (0.001-0.05 m diameter). Stochastic tracking of the turbulent dispersion of the particles was used by selecting the discrete random walk model. In this stochastic tracking approach, ANSYS Fluent predicts the turbulent dispersion by integrating the trajectory equations for individual particles using the instantaneous fluid velocity, $\bar{u} + u'(t)$ along the particle path during the integration.

Moreover, accretion model was activated to account for the particles deposition into the wall and the formation of the deposit layer. The pressure and velocity where coupled using phase coupled SIMPLE scheme while the second order Upwind spatial discretization method was selected for the calculation of momentum, turbulent kinetic energy and turbulent dissipation rate. First order Upwind was selected for volume fraction discretization. The under-relaxation factors which were assigned as solution controls are shown in Table 1.

Table 1
Under-relaxation factors for the solution controls

Volume fraction	Turbulent kinetic energy	Turbulent dissipation rate	Turbulent viscosity	Discrete phase sources	Pressure	Momentum
0.5	0.4	0.6	0.7	0.5	0.3	0.7

C. Governing equations

The transport equations for turbulent kinetic energy k and turbulent kinetic energy dissipation ε for this model are calculated from Eqns. (1) and (2), while the software solves the conservation Eqns. (3) and (4) for the momentum of each phase [12].

1) k-ε model transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}] + G_k + G_b - \rho\varepsilon \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}] + G_{1\varepsilon}\frac{\varepsilon}{k}(G_k + G_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (2)$$

where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients while G_b is the generation of turbulence kinetic energy due to buoyancy. $C_{1\varepsilon}$,$C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants while σ_k and σ_k are the turbulent Prandtl numbers of k and ε, respectively. The values for the k-ε model equations are listed in Table 2.

Table 2
Values for the k-e model constants

$C_{1\varepsilon}$	$C_{2\varepsilon}$	$C_{3\varepsilon}$	σ_k	σ_k
1.44	1.92	0.09	1.0	1.3

2) Continuity equations:

The volume fraction of each phase is calculated from a continuity equation

$$\frac{1}{\rho_{rq}}[\frac{\partial}{\partial t}(\sigma_q \rho_q) + \nabla.(\alpha_q \rho_q \vec{v}_q) = \sum_{p=1}^{n} \quad (\dot{m}_{pq} - \dot{m}_{qp})] \quad (3)$$

where $\rho_{rq}$is the phase reference density or the volume averaged density of the q^{th}phase in the solution domain.

The conservation of momentum for the FOG particles phase $s^{th}$is

$$\frac{\partial}{\partial t}(\sigma_s \rho_s \vec{v}_s) + \nabla.(\alpha_s \rho_s \vec{v}_s \vec{v}_s) = -\alpha_s \nabla p - \nabla p_s + \nabla.\bar{\bar{\tau}}_s + \alpha_s \rho_s \vec{g}$$
$$+\sum_{l=1}^{N} \quad (K_{ls}(\vec{v}_l - \vec{v}_s) + \dot{m}_{ls}\vec{v}_{sl}) + (\vec{F}_s + \vec{F}_{lift,s} + \vec{F}_{vm,s} + \vec{F}_{td,s}) \quad (4)$$

where $p_s$is the $s^{th} particles\ pressure$, K_{ls} is the momentum exchange coefficient between fluid *l* and particles phase *s*, *N* is the total number of phases, and $\vec{F}_s$, $\vec{F}_{lift,s}$, $\vec{F}_{vm,s}$ and $\vec{F}_{td,s}$ are defined as the external body force, the lift force, the virtual mass force and the turbulent dispersion force, respectively.

D. Boundary conditions

In this CFD model only a 1.5 m section of the sewer pipe is simulated, however, the inlet and outlet are assumed to be open to the manhole. The pressure at the inlet and outlet is the atmospheric pressure, while the inlet velocity was assumed to be 1.5 m/s. As the specification method for the turbulence, 5% intensity and 10 viscosity ratio were selected. For the DPM particles (i.e. FOG) the escape option was selected for the inlet and outlet while trap was selected for the condition at the wall. The trap option allows the particles that come into contact with the wall to be trapped in that boundary and thus form the deposition.

3. Results and Discussion

The solution of the set-up model has converged after 400 iterations and met the residuals convergence criteria of 10-6. The results of the CFD simulation showed that the velocity of the FOG particles at the inlet was very high even near the wall. However, the FOG velocity contours at the outlet show that the velocity reached zero at the wall, indicating deposition of the particles that stick to the wall (Fig. 3). The maximum FOG particle velocity was at the center of the pipe away from the wall, this is because the particles are dispersed by the wastewater and transported by the flow at the same velocity.

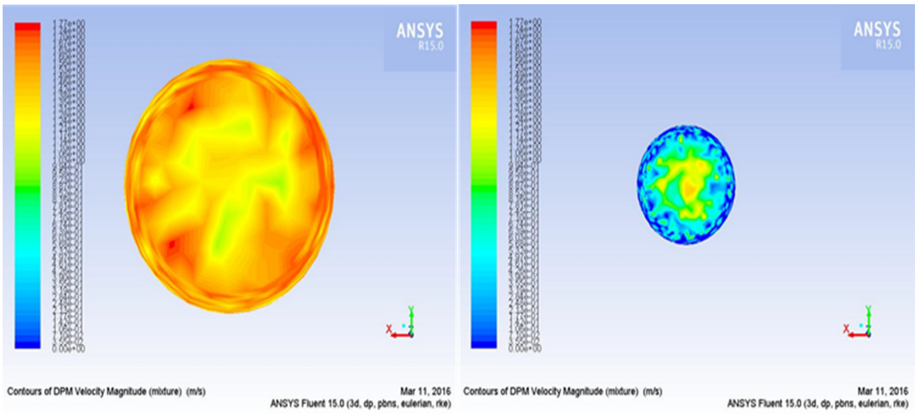


Fig. 3. Velocity contours of FOG particles at the inlet (left) and outlet (right)

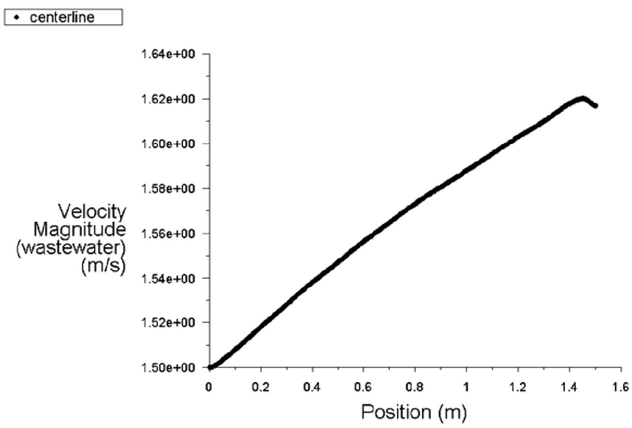


Fig. 4. development of the waste water velocity profile at the center line of the pipe

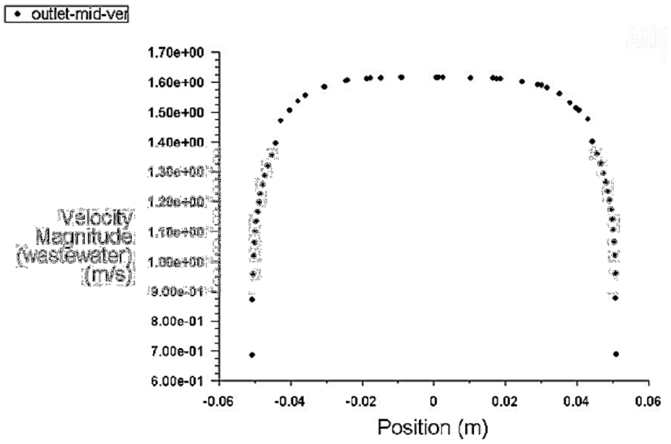


Fig. 5. Wastewater velocity distribution at the outlet horizontal line

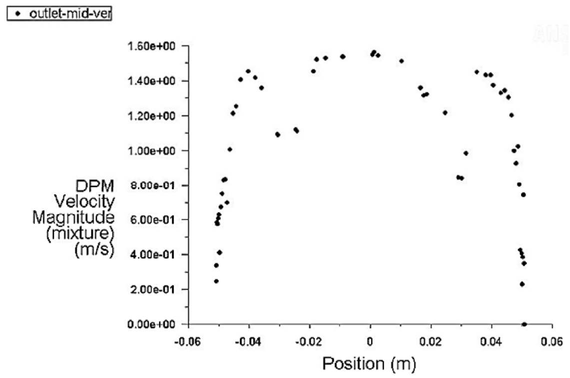


Fig. 6. FOG particles velocity at the outlet center line

Figure 4 shows that the velocity of the wastewater at the centerline has increased from 1.5 m/s initial velocity at the inlet to 1.62 m/s at the outlet which indicates the flow acceleration as a result of the gravity effect. Figure 5 shows the velocity of the wastewater at the horizontal centerline of the outlet. Because of the no-slip condition, the velocity at the wall is zero and increases gradually until it reaches the maximum value at the center of the pipe. On the other hand, the zero velocity of the FOG particles at the horizontal line of the outlet at the wall indicates that the particles are deposited on the wall. The low and high values of the particles velocity at the center of the outlet can be associated with the diameter of the particles whereby, larger particles have smaller velocity values than the smaller ones (Figure 6).

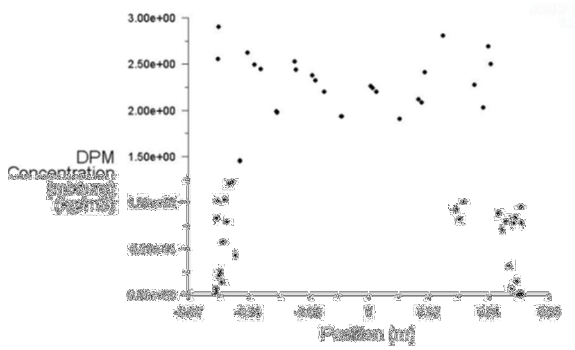


Fig. 7. FOG concentration at the outlet midline

Figure 7 shows the concentration of FOG particles at the vertical center line at the outlet. The amount of the particles near the wall boundary reached the highest values of 3.0e+00 kg/m^3. The velocity profile, volume fraction and turbulence parameters of the wastewater phase have been successfully obtained. All the results were presented in the form of xy plots at the centerline along the pipe and the vertical midline at the outlet. Contour plots of velocity, concentrations, pressure, and volume fractions were generated at slice plane in the center of the pipe.

The volumetric flow rate of the secondary phase in relation to the total volumetric flow rate is known as the phase volume fraction [13]. Figure 8 shows that the volume fraction of the FOG particles near the wall is higher than the center of the pipe indicating the deposition of the particles and formation of Eulerian film at the wall. In straight pipe, particles are trapped in the viscous sublayer of the flow near the wall and remain there for long period until they interact with the wall and deposit due to friction or bounce [14]. The concentration of the particles at the wall ranged from 0 to 3 kg/m3. It can be noticed that the volume fraction and concentration of FOG particles at the wall vary randomly, this may be due to the effect of the wastewater turbulence [15]. Similar observation was made from the pilot scale operation where the deposit thickness on the wall was not the same along the pipe. This indicates that while diffusional deposition occurs along the streamwise lines below the near-wall particle accumulating patterns, free particles deposit more evenly over the wall [16].

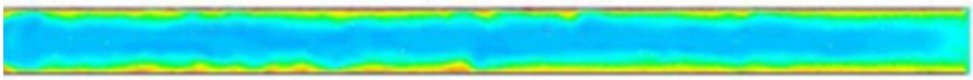


Fig. 8. Contours of the FOG particles volume fraction

4. Conclusion

The attempts have been made to show the flow characteristics of wastewater-FOG particles flow in gravity sewer pipe. ANSYS Fluent was used for the CFD simulation with the selection of two-phase Eularian model with DPM method. The model has shown a good potential for simulating the wastewater flow and presenting the deposition of FOG particles into the sewer pipe walls. Some flow patterns such as particles dispersion has not been predicted yet, however the flow characteristics including volume fraction, pressure, velocity profile and kinetic energy were discussed. The simulation results show that the maximum volume fraction of FOG particles appears at the lower section of the pipe. This model could be useful for predicting sewer pipe blockages.

References

[1] Yunus, A. Cengel, and John M. Cimbala. "Fluid mechanics fundamentals and applications." *International Edition, McGraw Hill Publication* 185201 (2006).

[2] Cushman-Roisin, Benoit, Carlo Gualtieri, and Dragutin T. Mihailovic. "Environmental Fluid Mechanics: Current issues and future outlook." *Fluid Mechanics of Environmental Interfaces, Taylor & Francis, Leiden* (2008): 1-16.

[3] Dufresne, Matthieu, José Vazquez, Abdelali Terfous, Abdellah Ghenaim, and Jean-Bernard Poulet. "Experimental investigation and CFD modelling of flow, sedimentation, and solids separation in a combined sewer detention tank." *Computers & Fluids* 38, no. 5 (2009): 1042-1049.

[4] Karama, A. B., O. O. Onyejekwe, C. J. Brouckaert, and C. A. Buckley. "The use of computational fluid dynamics (CFD). Technique for evaluating the efficiency of an activated sludge reactor." *Water science and technology* 39, no. 10-11 (1999): 329-332.

[5] Liu, Jie, Martin Crapper, and G. L. McConnachie. "An accurate approach to the design of channel hydraulic flocculators." *Water research* 38, no. 4 (2004): 875-886.

[6] Wols, B. A., J. A. M. H. Hofman, W. S. J. Uijttewaal, L. C. Rietveld, and J. C. Van Dijk. "Evaluation of different disinfection calculation methods using CFD." *Environmental Modelling & Software* 25, no. 4 (2010): 573-582.

[7] Chen, Zhi, Sangsoo Han, Fa-Yi Zhou, and Ke Wang. "A CFD modeling approach for municipal sewer system design optimization to minimize emissions into receiving water body." *Water resources management* 27, no. 7 (2013): 2053-2069.

[8] Torres, A., G. Lipeme Kouyi, J. L. Bertrand-Krajewski, J. Guilloux, S. Barraud, and A. Paquier. "Modelling of hydrodynamics and solid transport in a large stormwater detention and settling basin." In *11 th International Conference on Urban Drainage*. 2008.

[9] Guha, A. "Transport and deposition of particles in turbulent and laminar flow" *Annu. Rev. Fluid Mech*. (2008) 40:311-341.

[10] Husain, Iman AF, Ma'an Fahmi Alkhatib, Mohamed Saedi Jammi, Mohamed ES Mirghani, Zaki Bin Zainudin, and Asif Hoda. "Problems, control, and treatment of fat, oil, and grease (FOG): a review." *Journal of oleo science* 63, no. 8 (2014): 747-752.

[11] Longest, P., and Vinchurkar, S. "Effects of mesh style and grid convergence on particle depositon in bifurcating airway models with comparisons to experimental data." *Medical Engineering & Physics*, no. 29 (2007): 350-366.

[12] ANSYS Inc, A. 2013. ANSYS Fluent theory guide. Canonsburg: ANSYS Inc.

[13] Tuvnel. 2013. Good practice guide: an introduction to multiphase flow measurment. UK.

[14] Noorani, A., Sardina, G., Brandt, L., and Schlatter, P. "Particle transport in turbulent curved pipe flow." *Journal of fluid mechanic* no. 793(2016): 248-279.

[15] Hossain, A., Naser, J., McManus, K., and Ryan, G. " CFD investigation of particle deposition and distribution in a horizontal pipe." Third international conference on CFD in the minerals and process industries CSIRO, Melbourne, Australia (2003).

[16] Narayanan, C., and Lakehal, D. "Mechanisms of particle deposition in a fully developed turbulent open channel flow" *Physics of fluids* 15, no. 3 (2003): 763-775.