27TH INTERNATIONAL CONGRESS OF MECHANICAL ENGENEERING
COBEM 2023

Brazilian Society of Mechanical Sciences and Engineering
27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-2399
INVESTIGATING INTERNAL FLOW AND THE INFLUENCE OF INTERNAL MIXERS ON THE USEPA DYNAMIC FLUX CHAMBER FOR ODORANT COMPOUND EMISSIONS

Kamilla Furtado Cupertino
Companhia Espírito-santense de Saneamento
kamilacupertino@gmail.com
Laize Nalli de Freitas
Jane Meri Santos
Bruno Furieri
Universidade Federal do Espírito Santo
laizenalli@gmail.com, jmerisantos@yahoo.com.br, furieribruno@gmail.com
Philipe Uhlig Siqueira
Pontifícia Universidade Católica do Rio de Janeiro
philipe.uhlig@gmail.com
Ademir Abdala Prata Jr.
Universidade Federal do Alagoas
University of New South Wales
ademir@unsw.edu.au

Abstract. *The USEPA dynamic flux chamber (USEPA DFC) is widely utilized for measuring air pollutants emitted from various area sources, including Wastewater Treatment Plants, due to its versatility and accuracy. The volatilization mechanisms of compounds in the chamber are governed by mass transfer equations, such as the Two Film Theory. According to this theory, the emission flux is directly related to the concentration difference between the liquid and gaseous phases, regulated by a constant known as the global mass transfer coefficient. Despite its widespread use, there is limited literature available on the internal flow characteristics of the DFC. Some studies suggest that the use of internal mixers can enhance sampling efficiency. Therefore, the objective of this study was to investigate the internal flow of the DFC and assess the influence of internal mixers on flow distribution and velocity. To achieve this, computational fluid dynamics (CFD) simulations were conducted using Ansys Fluent software. The geometry of the USEPA DFC model, with variations in inlet flows and the addition of two internal fans, was used for the simulations. Acetic acid was chosen as the pollutant for analysis. The fluid flow within the chamber was assumed to be isothermal, incompressible, and turbulent, with the k-w SST turbulence model employed. The analyzed inlet flows were 2, 5, and 10 L/min, and two fans of different sizes (large and small) were introduced, with ascending and descending flow configurations considered. The simulation results revealed that both the variation in flow rate and the addition of fans had a direct impact on friction velocity and shear stress, particularly along the chamber wall. The inclusion of fans improved mixing inside the chamber; however, it also had a minor effect on the mass transfer coefficients of the analyzed compound, albeit not significantly. In summary, this study provided insights into the internal flow characteristics of the USEPA DFC and examined how the introduction of internal mixers, represented by fans, affected flow distribution and velocity. The findings underscore the importance of considering these factors when using the DFC for measuring emissions, contributing to improved sampling accuracy and reliability.*

Keywords: *Fluids mechanics. Odor compound. Dynamic flux chamber. Acetic acid. Wastewater Treatment Plant.*

1. INTRODUCTION

Computational fluid dynamics (CFD) is an exceptionally valuable tool in engineering. By utilizing numerical methods and algorithms, CFD enables engineers to develop computational models that simulate and analyze fluid flow in various adverse situations encountered in both industrial and everyday life scenarios. These simulations provide crucial insights into the behavior of fluids, helping engineers make informed decisions, optimize designs, and address challenges (Xu *et al.*, 2017). With CFD, engineers can virtually explore and understand the complex phenomena of fluid dynamics, such as turbulence, heat transfer, multiphase flows, and aerodynamics. By accurately representing the geometry, boundary conditions, and physical properties of the fluid, CFD simulations can predict and visualize how fluids interact with objects, surfaces, and their surroundings (Patrachari *et al.*, 2012).

The applications of CFD are diverse and wide-ranging. In industries like aerospace, automotive, energy, and environmental engineering, CFD plays a vital role in the design and analysis of aerodynamic profiles, combustion processes, cooling systems, and pollutant dispersion, among others (Patrachari *et al.*, 2012). By leveraging CFD, engineers can simulate and evaluate different scenarios without the need for costly and time-consuming physical prototypes or experiments. This allows for faster and more efficient design iterations, optimization of performance, and identification of potential issues before the actual implementation. Furthermore, CFD can aid in risk assessment, decision-making, and troubleshooting by predicting and mitigating potential fluid-related problems (Xu *et al.*, 2017; Patrachari *et al.*, 2012).

Within environmental engineering, particularly in air emission studies, Computational Fluid Dynamics (CFD) is a powerful tool that can be utilized for modeling the dispersion of atmospheric pollutants in urban areas (**?**). CFD enables the simulation of complex flow patterns, allowing for a better understanding of the behavior of pollutants and their distribution within a given environment.

In addition to modeling the dispersion of atmospheric pollutants, CFD can also be applied to study the emission of particulate matter (Furieri *et al.*, 2014). Particulate matter, consisting of solid particles and liquid droplets suspended in the air, plays a significant role in air pollution and can have adverse effects on human health. By employing CFD simulations, researchers can investigate the emission sources, dispersion patterns, and concentration levels of particulate matter in various scenarios, aiding in the development of effective mitigation strategies.

Furthermore, CFD can contribute to the feasibility analysis of enclosing devices used for quantifying emissions from area sources (Prata Jr *et al.*, 2016; Andreão and de Feroni, 2021). Enclosing devices are structures designed to capture and measure emissions from specific sources, such as industrial facilities or exhaust stacks. By employing CFD simulations, engineers can assess the performance and effectiveness of different enclosure designs, optimizing their configuration and ensuring accurate quantification of emitted pollutants.

Among the devices mentioned above, the Dynamic Flow Chamber (DFC) is a notable enclosure device commonly used for quantifying emissions from area sources in air emission studies (Prata Jr *et al.*, 2016; Andreão and de Feroni, 2021). The DFC is designed to capture and measure emissions from specific sources, like wastwater treatment plant ponds (Hudson *et al.*, 2006), reservoirs (Moreno-Silva *et al.*, 2020), landfills (Liu *et al.*, 2015) and . The DFC consists of an enclosed chamber or hood that is placed around the emission source. The chamber is equipped with sampling probes or inlets to capture the emitted pollutants. The airflow within the chamber is controlled, using, for exemple, monometers. Computational Fluid Dynamics (CFD) simulations can be employed to assess the performance and effectiveness of the DFC in capturing emissions. By modeling the airflow patterns and pollutant dispersion within the chamber, engineers can optimize the design and configuration of the DFC to ensure efficient and accurate quantification of emitted pollutants. Figure 1 shows a scheme of Dynamic Flux Chamber (DFC).

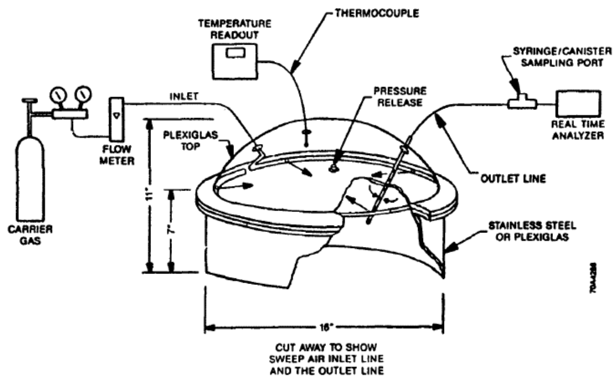

Figure 1: Schematic representation of DFC by Kienbusch (1986).

Despite the widespread utilization of the Dynamic Flow Chamber (DFC) in air emission studies, the literature on the internal flow characteristics of this device remains limited. This knowledge gap necessitates further investigation, particularly regarding the potential benefits of incorporating internal mixers to enhance sampling efficiency. Therefore, the objective of this study was to investigate the internal flow behavior of the DFC and evaluate the impact of internal mixers on flow distribution and velocity. To achieve this, Computational Fluid Dynamics (CFD) simulations were conducted using the Ansys Fluent software.

2. METHODOLOGY

For the simulations, several considerations were taken into account. The fluid inside the DFC was assumed to be isotropic at a temperature of 20^{o}C, incompressible, and turbulent. The selected pollutant for the simulation was acetic acid (HAAc). To obtain the variables of interest, such as the concentration of HAAc and flow velocity, conservation equations for mass, momentum, and the mass of the chemical species were solved. Turbulence was addressed using the Reynolds averaging procedure, and the Boussinesq analogy was utilized to model turbulent fluxes. Specifically, the $\kappa - \omega$

SST (Shear Stress Transport) model developed by Menter (1994) was employed to determine the turbulent viscosity. To ensure accuracy, all equations were solved in a coupled manner, considering a stationary regime. The Ansys Fluent software, based on the Finite Volume Method, was employed to solve the conservation equations. Additionally, mesh sensitivity tests were performed to assess the influence of mesh resolution on the simulation results. Table 1 describes the Boundary conditions used on CFD. Figure 2 represents the computational domain and mesh used.

Table 1: Boundary conditions.

Boundary	**Velocity**	**Turbulence**	**Concentration**
Inlet airflow holes	Prescribed Mass Inflow Rate	I=5%	$C = 0$
Gas-liquid interface	${u_i}^1 = 0$	κ and $\omega = 0$	$C_{G,i} = C_L K_H$
Outlet probe	Prescribed Mass Outflow Rate	-	-
Pressure relief	$\Delta P^2 = 0$	$dK/dx_i = 0$ and $dw/dx_i = 0$	$dC/dx_i = 0$
Walls (lateral, dome, probe line and fan blades)	$u_i = 0$	κ and $\omega = 0$	$dC/dx_i = 0$

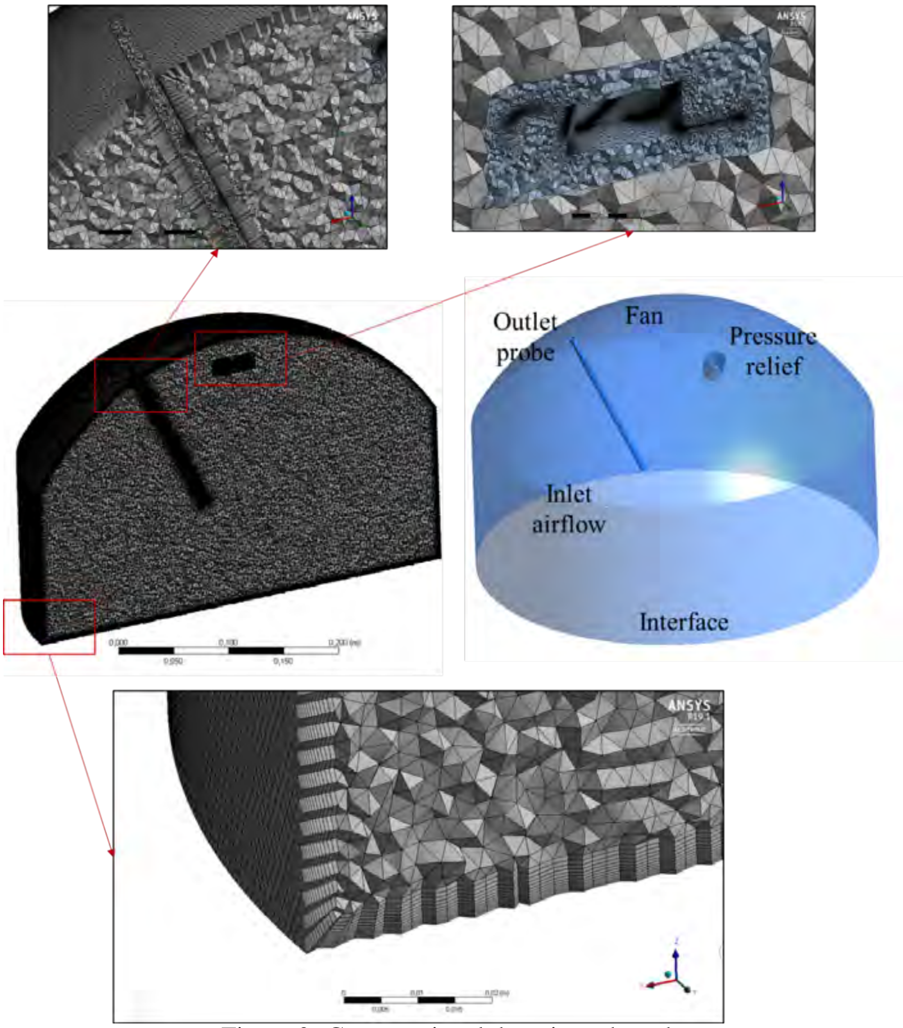

Figure 2: Computational domain and mesh.

Table 2 presents the investigated scenarios for all cases.

The constant concentration defined at the interface was calculated based on Equation 1, where $C_{g,i}$ is HAAc concentration at interface [kg/m^3]; C_L is the liquid solution concentration [kg/m^3]; K_H is the Henry's Law Constant[-].

$$C_{G,i} = C_L K_H = 23.53 \frac{mL}{L} 6.75E^{-6} = 1.6E^{-4} \frac{kg}{m^{-3}} \tag{1}$$

Table 2: Numerical simulation cases.

Case	**Flow orientation from fan**	**Fan rotation** [rev/min]	**Inlet sweep air flow rate** [L/min]
No fan	-	-	2, 5, 7, and 10
With fan	Downward	6000	5
With fan	Upward	6000	5
With fan	Upward	3200	5
With fan	Downward	3200	2, 5, and 10

3. RESULTS

In Figure 3, we observed a significant improvement in the mixing of HAAc concentration within the bulk flow after the inclusion of a fan in the DFC. This outcome was anticipated as the fan helps in enhancing the fluid dynamics and promoting better dispersion of the HAAc molecules. The insertion of the fan creates turbulence and induces a more vigorous mixing action, leading to a more uniform distribution of HAAc throughout the bulk flow. This is particularly beneficial for processes that rely on efficient mixing, such as chemical reactions or heat transfer. Moreover, the fan's presence in the DFC has a noticeable effect on the mean bulk concentration. We observed that the mean bulk concentration is slightly higher when utilizing the small fan up configuration. This suggests that the fan's enhanced mixing capability leads to a higher overall concentration of HAAc in the bulk flow. These findings highlight the positive impact of incorporating a fan in the DFC system, as it not only improves the mixing efficiency but also influences the average concentration of the target compound. Such insights are crucial for optimizing DFC designs and maximizing the performance of related processes.

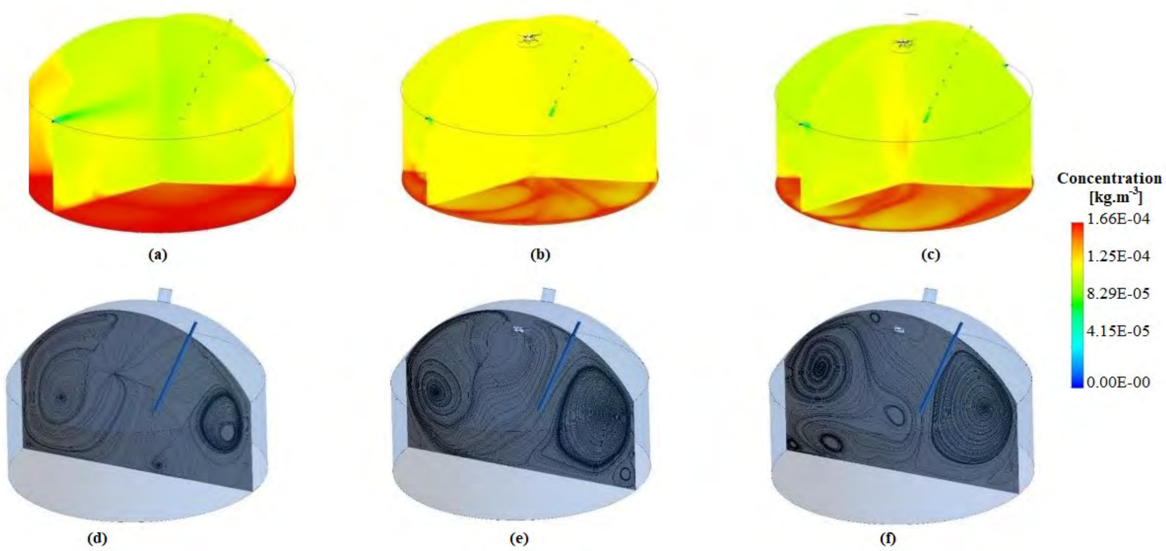

Figure 3: Computational domain and mesh.

In Figure 4, the presence of a fan inside the DFC resulted in a significant improvement in the mixing of HAAc concentration within the bulk flow. This outcome was expected, as the fan introduces increased turbulence and enhances the fluid dynamics, leading to better dispersion of the HAAc molecules. The fan's inclusion promotes more efficient mixing, which is particularly beneficial for processes that require thorough blending, such as chemical reactions or heat transfer. By creating turbulence and increasing the interaction between different fluid layers, the fan facilitates the homogenization of HAAc concentration throughout the bulk flow (Lyman *et al.*, 2018; Tran *et al.*, 2018; Parker *et al.*, 2013) .

Notably, the fan's effect on the mean bulk concentration was also observed (Figure 5). We found that the mean bulk concentration is slightly higher when utilizing the small fan up configuration. This indicates that the fan's improved mixing capability contributes to an overall higher concentration of HAAc in the bulk flow. Furthermore, it's important to mention that the fan's influence on mixing efficiency and bulk concentration is dependent on various factors. These factors include the fan's size, rotational speed, and placement within the DFC. Additionally, the properties of the fluid, such as viscosity and flow rate, can also impact the mixing performance (Lyman *et al.*, 2018; Tran *et al.*, 2018).

The findings from this study underscore the importance of integrating a fan into DFC systems to achieve improved mixing and concentration uniformity. By comprehending the relationship between fan configuration and bulk concentration,

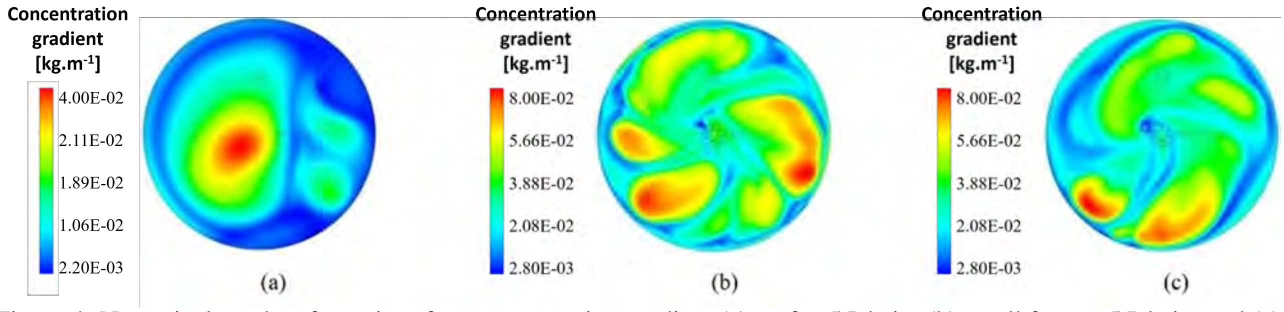

Figure 4: Numerical results of near interface concentration gradient (a) no fan 5 L/min; (b) small fan up, 5 L/min; and (c) small fan down, 5 L/min.

it becomes possible to optimize DFC designs and enhance the efficiency of associated processes. Further investigations that explore the influence of different fan parameters on mixing performance will yield valuable insights for future applications. In addition, through further examination of the flow patterns near the liquid-gas interface inside the DFC, we discovered zones of high friction velocity in cases where the fan was present. Interestingly, these zones exhibited convergent flow, which facilitated the accumulation of compounds and subsequently reduced concentration gradients. Consequently, the emission rates of odorants measured on liquid surfaces were found to be in the same order of magnitude with or without the use of an internal fan, contrary to the initial intuitive hypothesis that a fan would enhance emissions. The utilization of Computational Fluid Dynamics (CFD) techniques aided in understanding why the presence of a fan did not significantly increase odorant emissions from the liquid surfaces. This knowledge is crucial for dispelling preconceived notions and designing DFC systems that effectively manage odorant emissions.

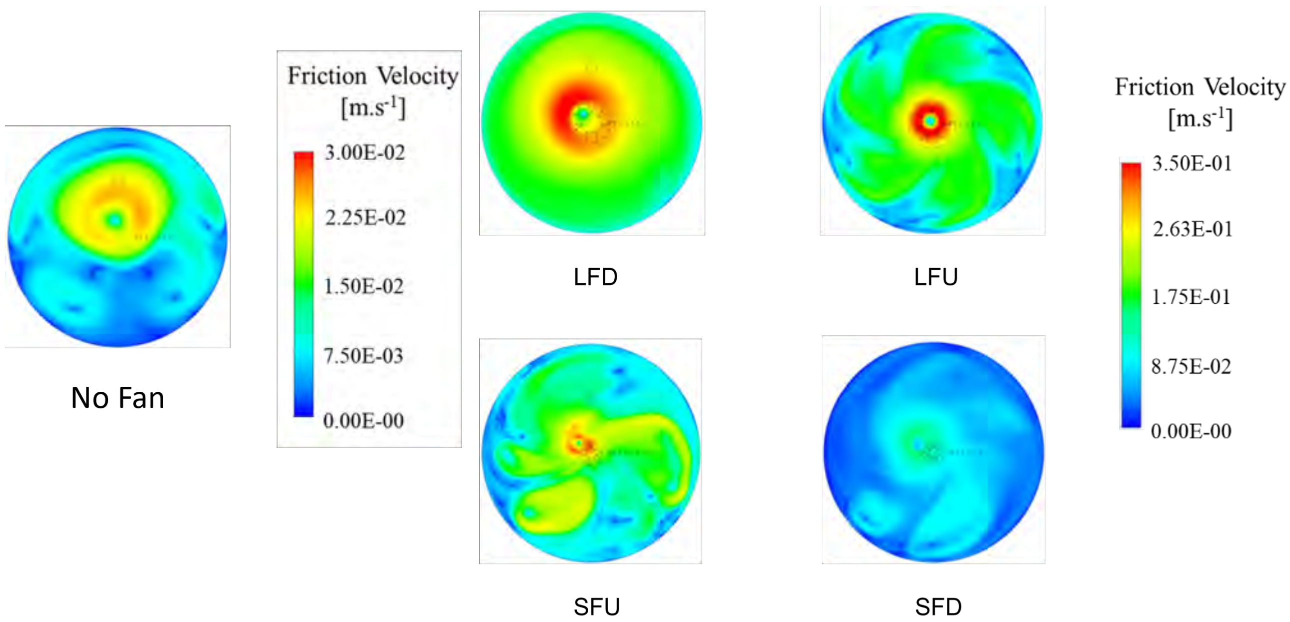

Figure 5: Numerical results for friction velocity for set ups: no fan, large fan down(LFD), large fan up (LFU), small fan up (SFU) and small fan down (SFD).

4. CONCLUSION

In conclusion, this study investigated the internal flow behavior of the Dynamic Flux Chamber (DFC) and evaluated the impact of internal mixers on flow distribution and velocity. By employing Computational Fluid Dynamics (CFD) simulations, the study provided valuable insights into the performance and effectiveness of the DFC in capturing emissions.

The results demonstrated that the incorporation of a fan inside the DFC significantly improved the mixing of HAAc concentration within the bulk flow. The fan-induced turbulence enhanced fluid dynamics, promoting better mixing of the HAAc and leading to a more uniform distribution throughout the bulk flow. This improved mixing is particularly advantageous for processes that rely on efficient blending. Furthermore, the presence of a fan in the DFC had a noticeable effect on the mean bulk concentration of HAAc. The small fan up configuration exhibited a slightly higher mean bulk concentration, indicating that the fan's enhanced mixing capability resulted in an overall higher concentration of HAAc in the bulk flow.

The study highlighted the importance of incorporating a fan in DFC systems to achieve enhanced mixing and improved concentration uniformity. The findings emphasized the need to consider factors such as fan size, rotational speed, and placement within the DFC to optimize design and maximize performance. Additionally, the examination of flow patterns near the liquid-gas interface inside the DFC revealed zones of high friction velocity with convergent flow when the fan was present. This led to compound accumulation and reduced concentration gradients, contrary to the initial hypothesis that a fan would enhance odorant emissions measured on liquid surfaces. The utilization of CFD techniques provided a deeper understanding of why the fan did not significantly increase odorant emissions, when we analyze compounds with volatilization controlled by gas phase. Overall, this study demonstrated the valuable applications of CFD in analyzing and optimizing DFC systems, contributing to the advancement of air emission studies and pollution mitigation strategies.

5. ACKNOWLEDGEMENTS

The authors acknowledge the sponsorship from Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES), Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq), Fundação de Amparo à Pesquisa do Espírito Santo (FAPES) and Companhia Espírito Santense de Saneamento (CESAN), Brazil.

6. REFERENCES

Andreão, W.L. and de Feroni, R., 2021. "Cfd modeling of different mass transfer coefficients on hydrogen sulfide emission in a flux chamber". *Environmental Science and Pollution Research*, pp. 1–14.

Furieri, B., Harion, J., Milliez, M., Russeil, S. and Santos, J., 2014. "Numerical modelling of aeolian erosion over a surface with non-uniformly distributed roughness elements". *Earth Surface Processes and Landforms*, Vol. 39, No. 2, pp. 156–166.

Hudson, N., Gies, A. and Duperouzel, D., 2006. "Assessment of permeable covers for odour reduction in piggery effluent ponds. 2. field-scale trials". *Bioresource technology*, Vol. 97, No. 16, pp. 2015–2023.

Kienbusch, M., 1986. "Measurement of gaseous emission rates from land surfaces using an emission isolation flux chamber".

Liu, Y., Lu, W., Li, D., Guo, H., Caicedo, L., Wang, C., Xu, S. and Wang, H., 2015. "Estimation of volatile compounds emission rates from the working face of a large anaerobic landfill in china using a wind tunnel system". *Atmospheric Environment*, Vol. 111, pp. 213–221.

Lyman, S.N., Mansfield, M.L., Tran, H.N., Evans, J.D., Jones, C., O'Neil, T., Bowers, R., Smith, A. and Keslar, C., 2018. "Emissions of organic compounds from produced water ponds i: Characteristics and speciation". *Science of the Total Environment*, Vol. 619, pp. 896–905.

Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA journal*, Vol. 32, No. 8, pp. 1598–1605.

Moreno-Silva, C., Calvo, D.C., Torres, N., Ayala, L., Gaitán, M., González, L., Rincón, P. and Susa, M.R., 2020. "Hydrogen sulphide emissions and dispersion modelling from a wastewater reservoir using flux chamber measurements and aermod® simulations". *Atmospheric Environment*, Vol. 224. ISSN 18732844. doi:10.1016/j.atmosenv.2020.117263.

Parker, D.B., Gilley, J., Woodbury, B., Kim, K.H., Galvin, G., Bartelt-Hunt, S.L., Li, X. and Snow, D.D., 2013. "Odorous voc emission following land application of swine manure slurry". *Atmospheric Environment*, Vol. 66, pp. 91–100.

Patrachari, A.R., Podichetty, J.T. and Madihally, S.V., 2012. "Application of computational fluid dynamics in tissue engineering". *Journal of bioscience and bioengineering*, Vol. 114, No. 2, pp. 123–132.

Prata Jr, A.A., Santos, J.M., Beghi, S.P., Fernandes, I.F., Vom Marttens, L.L., Neto, L.I.P., Martins, R.S., Reis Jr, N.C. and Stuetz, R.M., 2016. "Dynamic flux chamber measurements of hydrogen sulfide emission rate from a quiescent surface–a computational evaluation". *Chemosphere*, Vol. 146, pp. 426–434.

Tran, H.N., Lyman, S.N., Mansfield, M.L., O'Neil, T., Bowers, R.L., Smith, A.P. and Keslar, C., 2018. "Emissions of organic compounds from produced water ponds ii: Evaluation of flux chamber measurements with inverse-modeling techniques". *Journal of the Air & Waste Management Association*, Vol. 68, No. 7, pp. 713–724.

Xu, G., Luxbacher, K.D., Ragab, S., Xu, J. and Ding, X., 2017. "Computational fluid dynamics applied to mining engineering: a review". *International Journal of Mining, Reclamation and Environment*, Vol. 31, No. 4, pp. 251–275.

7. RESPONSIBILITY NOTICE