# Analysis of the possibility of dispersion of SARS-CoV-2 in bathrooms using computational fluid dynamics (CFD)

## L.S. Vieira (1), L.H. Oliveira (2), F.A. Kurokawa (3), A.S. Oizuni (4)

(1) ludmila.vieira@usp.br

(2) lucia.helena@usp.br

(3) fernando.kurokawa@usp.br

(4) amandasayuri@usp.br

(1, 2, 3, 4) Department of Construction Engineering, Polytechnic School, University of São Paulo, Brazil

## Abstract

The epidemiological crisis generated by the SARS-CoV-2 virus, which caused the COVID-19 pandemic, caused profound changes around the world, challenging science and social relations. Although the first studies on the mode of transmission of the disease indicated that it occurs through droplets and respiratory secretions, researchers has raised the hypothesis that the infection could also occur through the oral-fecal route. When considering this other possibility of transmission, public restrooms and those in health units whose users have been have been diagnosed with COVID-19 become critical places due to the greater potential for spreading the disease. Considering the dynamics of flushing in the toilets, the force of water turmoil would be able to cause the suspension of aerosol particles and water droplets that could contain the infectious agent that could be inhaled by other users. In this context, this article analyzes the possibility of SARS-CoV-2 dispersion in toilets, using the concepts of computational fluid dynamics (CFD), through the OpenFOAM software to perform computational simulations. A toilet model traditionally marketed in Brazil was adopted as a geometric reference for the computational model. The behavior of the multiphase system was observed for a laminar flow, in order validate the simulation, in a preliminary way. The results obtained do not identify the potential for dispersion of aerosols and droplets contaminated by SARS-CoV-2 when a turbulence model is not applied.

## Keywords

Computational fluid dynamics; SARS-CoV-2; toilet.

# 1 Introduction

The World Health Organization (2020) has recognised the possibility of transmission of SARS-CoV-2 through aerosols, which can come from body secretions and excretions, spread through activities such as talking, coughing, sneezing, especially in closed environments.

In atmospheric air, its transmission capacity can also be affected by meteorological and climatic factors, such as temperature, humidity and certain pollutants, such as particulate matter, carbon monoxide and nitrogen oxides (Barcelo, 2020).

Although the first studies on the mode of transmission of COVID-19 indicated that it occurs through droplets and respiratory secretions, researchers began to hypothesize that the infection could also occur through fecal-oral route. Suspicion about this new means of contamination began in early 2020, after a person in the USA presented to an urgent care clinic with a history of nausea and vomiting for two days, besides the common complaint of persistent dry cough, demonstrating that the virus could also be present in the patient's gastrointestinal system (Holshue, 2020).

A study developed by Bourouiba (2021) captured an image at 2,000 frames per second of the trajectories of droplets generated due to the flushing of a toilet in a hospital. Data analysis revealed that a large number of the generated droplets are not visible to the naked eye. These emissions represent more than 6 mL and can remain suspended in the air for a long time compared to the larger visible droplets (up to 6 mm in diameter) that settle on surfaces.

Weidhaas (2021) reported the presence of SARS-CoV-2 in a sewage samples collected from areas whose users were diagnosed with COVID-19, corroborating the reflection that the virus would be present in sanitary appliances. Thus, the bathrooms, especially those of public use and health care facilities, such as hospitals and outpatient clinics, would be critical sites because of the enormous potential for disease dissemination. The concern becomes even greater due to the dynamics of flushing the toilet, since the force of swirling water is capable of causing the suspension of aerosol particles that could contain the infectious agent.

Johnson et al. (2013) analysed three models of toilets with similar flush volumes and found that all of them were capable of generating bioaerosols with the actuation of the flush, their production being proportional to the increase of the flush energy. Analyzing the dynamics of bioaerosol production, Liu et al. (2020) performed a simulation using computational fluid dynamics (CFD) as methodology and demonstrated that, during toilet flushing, massive upward transport of aerosol particles, presumably infected with SARS-CoV-2, can occur.

In this context, the aim of this paper is to analyse the possibility of dispersion of SARS-CoV-2 by means of aerosols from Brazilian toilets, after flushing, employing the concepts of computational fluid dynamics (CFD).

## 2 Computational Fluids Dynamics Approach

It is possible to simulate any flow through channels, pipes and around a body, whether rigid or mobile, by processing the equations that govern fluid dynamics quickly, providing highly accurate results, with a minimised margin of error and, still simulating scenarios that are faithful to reality (Kurokawa, 2019).

Despite the great advantages of the CFD technique, Kurokawa (2019) also points out that there are complex situations to be modeled such as, for example, the turbulence phenomenon. McDonough (2007), reiterates that most fluid flows encountered in engineering practice are turbulent and states that their understanding is "one of the most intriguing, frustrating and important problems in classical physics". Thus, in this article, a simplification of the model was adopted, considering that it is a laminar flow.

Assuming the flow to be incompressible, the equations necessary to solve problems of flow of Newtonian and isothermal fluids are the equations of conservation of mass and momentum, given respectively by Equations 1 and 2.

$$\nabla \cdot \mathbf{u} = 0 \qquad \text{Eq. 1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla(\mathbf{uu}) = \frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + F \qquad \text{Eq. 2}$$

where, $\mathbf{u}$ is the velocity vector, $t$ is time, $\rho$ is the fluid density, $p$ is the pressure, $\nu$ is the viscosity coefficient and, $F$ represents the external forces.

The differential equations, together with the boundary conditions, are solved numerically in the OpenFOAM environment using the finite volume method (FVM), in which the domain of interest is subdivided into a finite number of smaller parts, called control volumes, and the quantities of interest are calculated approximately in each of these volumes. The resulting solution satisfies the conservation of quantities as mass, quantity of movement, energy in any finite control volume.

The method of calculation of the numerical solution employed by the tool is the PISO - Pressure-Implicit with Splitting of Operators (Issa, 1986). However, a modification was performed in the solver of the tool, pisoFoam, for the calculation of the numerical solution. The objective was to change the time step on-the-fly from stability and convergence constraints, i.e., reduce the number of iterations with the automatic adjustment of the time step during processing conditioned by the analysis of the number of Courant (Courant et al., 1967).

The interpolation scheme adopted for the discretization of the 1st order temporal derivatives was Implicit Euler, and for the discretization of the diffusive and advective terms the central differences scheme was adopted. In the case of the solution of the pressure variable algebraic system, it was used the Geometric-algebraic multi-grid (GAMG) iterative method with Gauss-Seidel buffer

# 3 Methodology

The field of analysis of this research are restrooms, having as key element a toilet. Figure 1 shows a restroom and Figure 2 shows an elevation of the toilets.

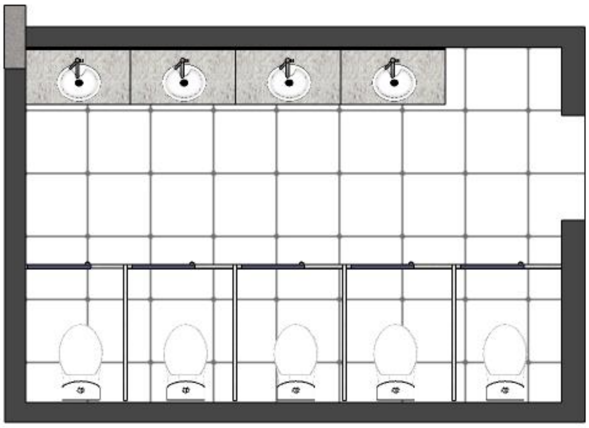


**Figure 1 - Schematic plan of restroom**
**Fonte: Vieira (2022)**

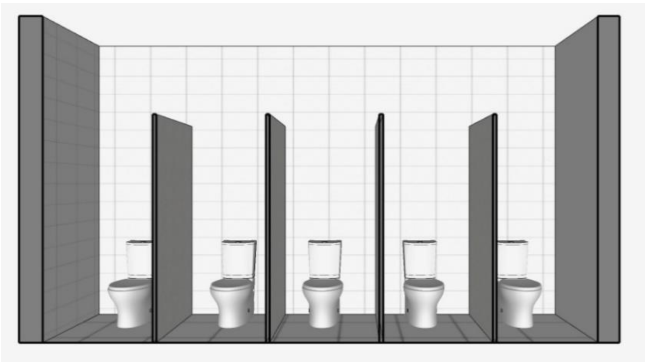


**Figure 2 - View of the toilets boxes**
**Fonte: Vieira (2022)**

This restroom is located in the city of São Paulo, where gravitational acceleration is 9.81 $m/s^2$ and air temperature is assumed to be 27 degrees Celsius.

The toilet reference model adopted has a nominal flush volume of 6 litres. The water enters the basin laterally, producing a vortex-like flow, enhancing the discharge of solid waste. It is a consolidated brand in the Brazilian market, being widely used in public establishments. Figures 3 and 4 show the cross-sectional and longitudinal views and sections, respectively.

**Figure 3 - Views of the toilet**
**Fonte: Vieira (2022)**

**Figure 4 – Toilet sections**
**Fonte: Vieira (2022)**

From this reference of a toilet, Figure 5 (a), to validate the parameters and conditions of the computer simulation, a simplified geometric model of a toilet was adopted Figure 5 (b) and, for the mesh generation, the tool *SnappyHexMesh* from OpenFOAM was used.

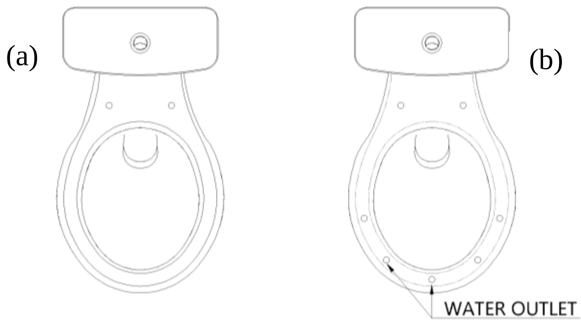



**Figure 5 – Original toilet (a) and simplified toilet (b)**

**Fonte: Vieira (2023)**

Regarding computational resources, they were used, in a first moment, for model validation:

- notebook with Intel Core i5-7200U processor and Turbo Boost function, NVIDIA GeForce 940MX graphics card with 2GB of dedicated RAM and 8GB DDR4 memory;
- software version OpenFOAM-v2106, June 2021, installed in a virtual machine with Linux operating system (Ubuntu); and
- for the post-processing stage, we used the software Paraview (embedded in the OpenFOAM package), version 5.4.1, which allows interactive visualization of the results.

It is considered, for this computer simulation, the analysis of a multiphase system, since the presence of two phases is verified: water in liquid state and air. The model does not consider the presence of solids. Simulating multiphase flows is not an easy task either. The complex nature of multiphase flows is due to the transient nature of the flows; the existence of dynamically changing interfaces; discontinuities; the interactions of small-scale structures such as bubbles and particles; particle-particle interactions; mass transfer and phase change; turbulence; among other factors (Berlemont, 1995).

## 4 Results and discussions

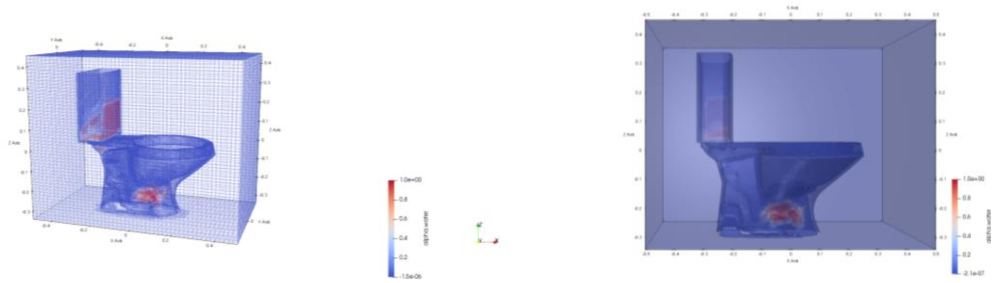
In this study, the Volume of Fluid (VOF) method, developed by Hirt and Nichols (1981), is used to evaluate the interface between the two phases. In this method, the phases of a multiphase system are mathematically treated as continuous and interpenetrating, in which the volume of each phase cannot occupy the volume of another phase. Thus, introducing the concept of volume fraction of phases, using a scalar indicator function, which varies from zero (without material) to one (completely filled with material), as follows:

$\alpha q = 0$: no fluid;

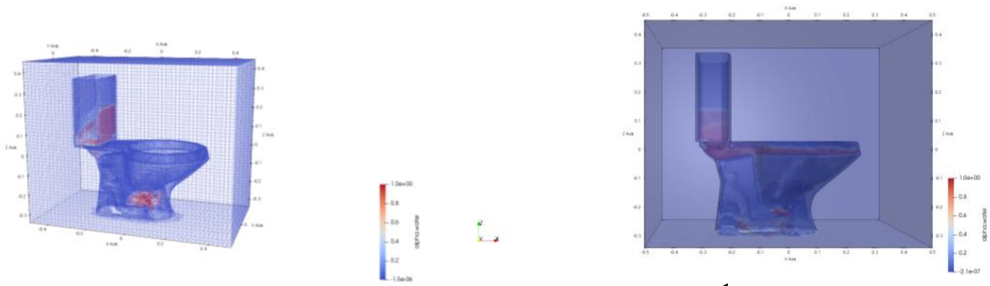
$0 > \alpha q > 1$ interface; e

$\alpha q = 1$ with fluid.

The solution of the system of equations composed of the equations of the volume fraction of each phase combined to a single equation of the quantity of motion, are all integrated in each cell of the model mesh. Using conventional computational resources, the mesh of this model was generated with 302.031 cells. Figure 6 illustrates the visualized fluid behavior for each of the identified simulation times, and visualization of the generated mesh.



t= 0.1 t= 0.1



t= 1 t= 1

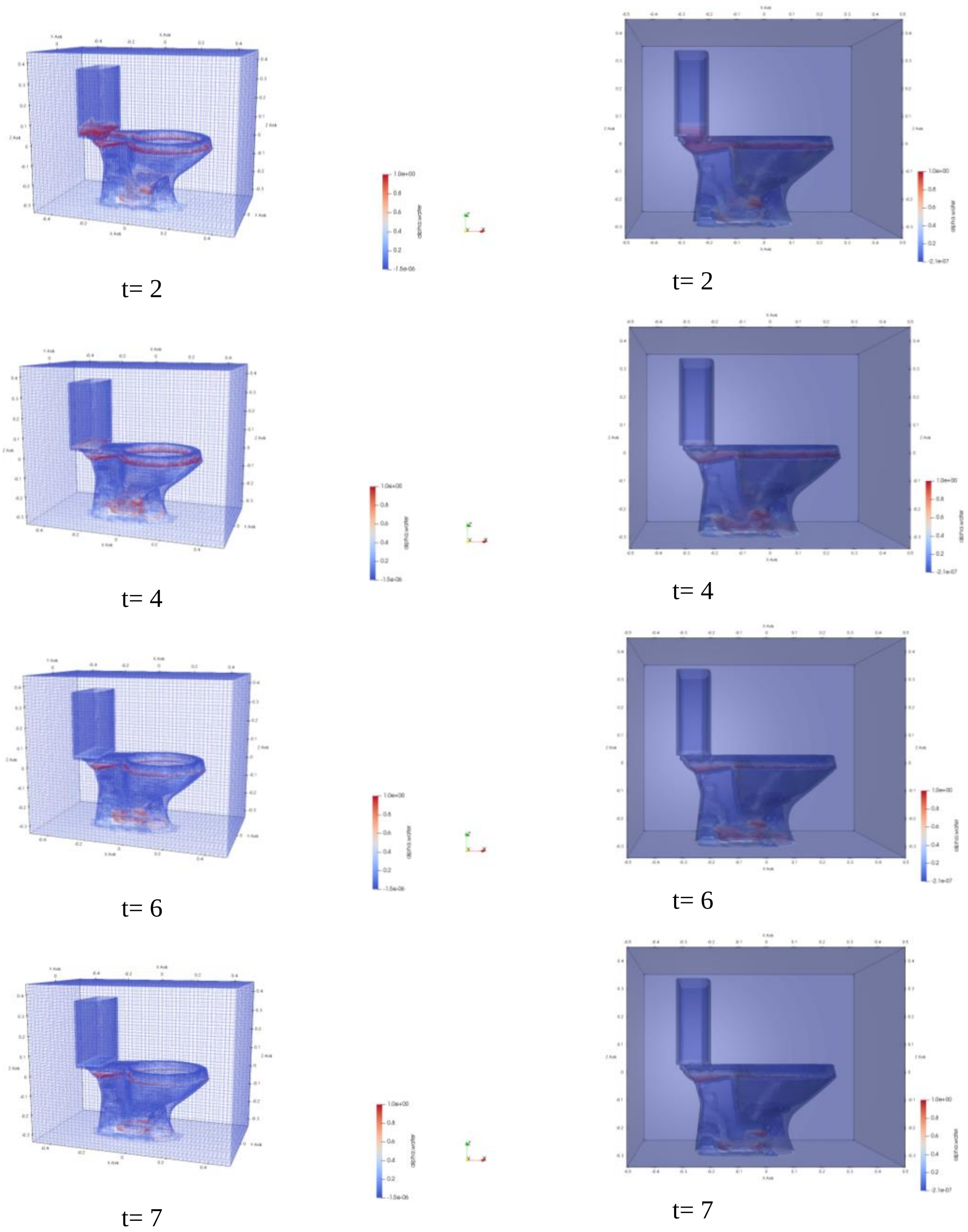



**Figure 6 – Results viewed on Paraview**

Analyzing the simulation results, although a simplified were adopted considering the computational resources available, it can’t be observed that, after the flushing of the toilet, there is the possibility of aerosol suspension beyond the limits of the toilet seat; therefore, the hypothesis of the spread of diseases through aerossol needs validation using more robust experiments.

## 5 Final considerations

The continuation of this study intends to evaluate the behavior of the system by inserting a turbulence model. In the OpenFOAM tool, several turbulence models of the RANS (*Reynolds Average Navier Stokes*) and LES (*Large Eddy Simulation*) type are available. These models cover a relatively large set of flow problems that include time-dependent variables, fluid-structure interaction and heat transfer.

Thus, the the Standard $k - \varepsilon$ (Launder; Spalding, 1974) turbulence model will be used in the next phase of the research. This model is semi-empirical based on the modeling of the transport equations of the turbulent kinetic energy ($k$) and the energy dissipation rate ($\varepsilon$) given respectively, by Equations 3 and 4.

$$\frac{\partial \kappa}{\partial t} + \nabla \cdot (\kappa \mathbf{u}) = \nabla\left(\left(\mu + \frac{\nu_t}{\sigma_\kappa}\right)\nabla \kappa\right) + P - \varepsilon \qquad \text{Eq. 3}$$

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = \nabla\left(\left(\mu + \frac{\nu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right) + C_{1\varepsilon}\frac{\varepsilon}{\kappa}P - C_{2\varepsilon}\frac{\varepsilon^2}{\kappa} \qquad \text{Eq. 4}$$

where $P$ is the mean turbulent kinetic energy production term, $\sigma_\kappa$ and $\sigma_\varepsilon$ are the turbulent diffusion coefficients, and $C_{1\varepsilon}\ and\ C_{2\varepsilon}$ are empirical constants. The turbulent viscosity, $\nu_t$ is calculated by Equation 5.

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \qquad \text{Eq. 5}$$

where $C_\mu$ is also an empirical constant. The other constants are obtained from the correlation of experimental data from various turbulent flows, and are given by: $C_\mu = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_\kappa = 1.0$ e $\sigma_\varepsilon = 1.3$.

After inserting the turbulence parameters in the computational model, it is estimated that it will be possible to visualize the possibility of aerosol suspension beyond the limits of the toilet seat, a situation that may represent a new potential for disease dissemination through aerosols.

## 6 References

Barcelo, D. (2020). An environmental and health perspective for COVID-19 outbreak: meteorology and air quality influence, sewage epidemiology indicator, hospitals disinfection, drug therapies and recommendations, *Journal of Environmental Chemical Engineering*, 8 (4): 104006.

Berlemont, A., Grancher, M.S., Gouesbet, G. (1995). Heat and mass transfer coupling between vaporizing droplets and turbulence using a Lagrangian approach, *International journal of heat and mass transfer*, 38(16), 3023-3034.

Bourouiba, L. (2021). The fluid dynamics of disease transmission. *Annual Review of Fluid Mechanics*, 53, 473-508.

Courant, R., Friedrichs, K., Lewy, H. (1967).On the partial differential equations of mathematical physics. *IBM Journal Research and Development*, 11, 215-234.

Hirt, C.W., Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundarie. *Journal of computational physics*, 39 (1), 201-225.

Holshue, M.L. et al. (2020). First case of 2019 novel coronavirus in the United States. *New England Journal of Medicine*.

Issa, R.I. (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting, *J. Comput. Phys.*, 62, 40–65**.**

Johnson, D. et al. (2013). Aerosol generation by modern flush toilets. *Aerosol Science and Technology*, 47 (9), 1047-1057.

Kurokawa, F.A. (2019). *Contribuições das investigações de técnicas numéricas para o desenvolvimento de modelagens em CFD para problemas de engenharia civil*. [Tese de Livre-docência, Universidade de São Paulo].

Launder, B.E., Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer methods in applied mechanics and engineering*. 3 (2), 269–289.
Liu, Y. et al. (2020). Aerodynamic analysis of SARS-CoV-2 in two Wuhan hospitals. *Nature*, 582 (7813), 557-560.

Mcdonough, J.M. (2007) *Introductory lectures on turbulence: physics, mathematics and modeling*.

Weidhaas, Jet al. (2021). Correlation of SARS-CoV-2 RNA in wastewater with COVID-19 disease burden in sewersheds, *Science of The Total Environment*, 775:145790.

World Health Organization (2020). *Novel coronavirus (2019-nCoV). Situation report – 1*. <https://www.who.int/docs/default-source/coronaviruse/situationreports/20200121-sitrep-1-2019ncov.pdf?sfvrsn=20a99c10_4>.

# 7 Presentation of Authors

Ludmila Souza Vieira holds a Master's of Civil Engineering from *Escola Politécnica* of the University of São Paulo, Department of Construction Engineering. Her thesis analyzes the dispersion of SARS-CoV-2 by means of aerosols from sanitary basins using CFD.

| | |
|---|---|
| Lúcia Helena de Oliveira is Associate Professor at the Department of Construction Engineering of *Escola Politécnica* of the University of São Paulo, where she teaches and conducts research work on building services. |  |
| Fernando Akira Kurokawa is Assistant Professor at Department of Construction Engineering of *Escola Politécnica* of University of São Paulo, where he teaches and conducts researches on CFD simulations and graphical geometry for engineering. |  |
| Amanda Sayuri Oizuni is an undergraduate student at Department of Construction Engineering of *Escola Politécnica* of University of São Paulo with scholarship granted by PUB-USP for the undergraduating research on Computational Modeling and Prototyping in Civil Engineering. |  |