Design of a Localized Air Exhaust System for an Undergraduate Dental Practice Room Under Covid-19 Context, a Case Study

Elías Espínola
Universidad Nacional de Asunción
Asunción, Paraguay
eliasespinola@fiuna.edu.py

Vivian González
Universidad Nacional de Asunción
Asunción, Paraguay
vcgonzalez@fiuna.edu.py

Liz Esquivel
Universidad Nacional de Asunción
Asunción, Paraguay
lizesquivel@fiuna.edu.py

Abstract

The COVID-19 pandemic has caused a significant impact on the global economy and significantly slowed down human activity. Paraguay (PY), a small developing South American country, was not an exception. Since this pandemic reached this country, the local government implemented a series of lockdown and restrictive measures since early 2020. In order to contain the spread of this virus, school activities such as course lectures, especially laboratory practices, were placed on hold indefinitely. As a result of restrictive access to online learning in Paraguay, the faculty of dentistry requests the faculty of engineering (FIUNA) to design a safe ventilation system to allow resuming laboratory practices in the faculty of dentistry. This led to the urgent need to reopen universities, schools, and other academic institutions to resume teaching activities.

Computational Fluid Mechanics was utilized to validate a localized air exhaust system design. This work helped to support the use of this engineering tool as part of the engineering design procedure. Results showed that a localized airflow of 150 CFM near the patient's head combined with a transparent wall and personal protective equipment helped avoid virus spreading. Different exhaust configurations were taken into consideration. In addition, air flow lines and velocity profiles surrounding the dental chair are shown in this work. It is essential to remark here that this worked to help the school administration to reopen laboratory activities. This helped to confidently decide on resuming practices and, at the same time, keeping students and faculties safe.

Keywords

Airborne Transmission, CFD, Dental Chair, Ventilation, Air Exhaust System

1. Introduction

The COVID-19 pandemic hit the world in an unprecedented manner in early 2020, forcing its population to unforeseen lifestyle changes and leaving them in an urgent need for safety guidelines in order to shield themselves from the dangerous unfamiliar virus. As it happened in many regions of the world, a strict curfew was placed in Paraguay in the second week of March in an attempt to restrain the transmission of the virus once the first case of the disease was confirmed in the country. Schools and universities were ordered to close while authorities and officials worked on safety protocols for students and education workers and, in a similar way, many other working sectors were demanded to cease their activities in the time that health authorities contemplated how hazardous their professional endeavors were in the context of the pandemic. Such was the case of dentists, with good reason, and as such, dentistry students doing laboratory practices and internships were hugely affected by these precautionary measures. This work came as an official request from the faculty of dentistry at the National University of Asuncion. A proposal to design a proper air shield was accepted and the design of a localized air ventilation system was conducted by undergraduate students from the Department of Mechanical Engineering. Inappropriate airflow in an enclosed space is one of the main factors in the spread of this virus. When combined with personal protective equipment, proper air ventilation can significantly reduce the spreading of an airborne virus.

1.1 Objectives

Several aircraft companies designed localized ventilation systems to provide safe air circulation to passengers. This same concept was adopted to design a localized air suction system at the dental chair for safe practice at the faculty of dentistry. The main objective of this research is to analyze the improvement of ventilation in the dental room through the use of the air exhaust system and compare different configurations to find the most effective one.

2. Literature Review

The COVID-19 pandemic hit the world in an unprecedented manner in early 2020, forcing its population to unforeseen lifestyle changes and leaving them in an urgent need for safety guidelines in order to shield themselves from the dangerous unfamiliar virus. As it happened in many regions of the world, a strict curfew was placed in Paraguay in the second week of March in an attempt to restrain the transmission of the virus once the first case of the disease was confirmed in the country. Schools and universities were ordered to close while authorities and officials worked on safety protocols for students and education workers and, in a similar way, many other working sectors were demanded to cease their activities in the time that health authorities contemplated how hazardous their professional endeavors were in the context of the pandemic. Such was the case of dentists, with good reason, and as such, dentistry students doing laboratory practices and internships were hugely affected by these precautionary measures.

Thorough research on virus spread mechanisms was conducted throughout the world. For instance, the work by Ai et al. (2019), where measurements and evaluations were conducted for airborne transmission of COVID-19 between room occupants during short-term events. There, it was concluded that person to person proximity increases the probability of contagion by a significant degree.

Another publication, such as the one from Lu et al. (2020), carried out a case study of airflow from an air conditioning system inside a restaurant in Guangzhou, China. The study demonstrated that airflow paths were a crucial factor in the virus spread mechanism, as was the layout of the tables at the restaurant.

Based on this information, a plausible origin for aerosol transmission of COVID-19 is the air surrounding a dental chair at a dental office during a dental procedure, since during the use of dental drills, air-water syringes and others, this causes dentistry professionals and students to come into contact with a visible spray that can contain particle droplets of water, saliva, blood, microorganisms and other debris, as stated by Lee et al. (2020). This led the Faculty of Dentistry at the National University of Asuncion to request the Faculty of Engineering of said university to bring forth a solution to this issue.

A potential solution to this is, as inspired by Lin (2020), is the design of a ventilation mechanism in the form of an air exhaust system for a dental office in an optimal position with respect to the chair in order to drain a substantial amount of contaminated air from the small, enclosed space, thus minimizing the possibility of contagion and the probability of virus stagnation in the room. To achieve this result, it is of great importance to identify the critical points in the vicinity of the chair with a high contamination assessment.

For this purpose, a Computational Fluid Dynamics (CFD) analysis is performed in order to observe the airflow lines and the velocity profile in the room during a dental practice, and both elements will be used as a tool to obtain an ideal model to propose an optimal solution. Various authors such as Posner et al. (2003), Gan et al. (1994), Abanto et al. (2004) have already published about airflow modeling. Their experimental research showed that the placement of objects between air inlets and outlets, as well as the shape of ceiling diffusers, have an impact on the air distribution inside the room, demonstrating how decisive indoor setting configuration is for proper ventilation.

Yang et al. (2014), through simulations of residential indoor air quality, demonstrated that certain areas of a room do not have adequate air circulation; hence, a potential location for contaminant accumulation is developed. A study on the impact of airflow profile in indoor air quality conducted by Sekhar et al. (2020) revealed that gas concentrations differ in various areas of an enclosed space; this is CO2, CO, bacteria, fungi, etc.

Consequently, in order to evaluate the air quality inside the dental office, air exchange rate values (ACH) will be calculated for different exhaust system configurations, and subsequently analyzed to propose an appropriate solution that will have a positive impact on the studied environment.

For this analysis, commercial CFD software is used. Flow Simulation from SolidWorks was utilized on a regular PC at the Faculty of Engineering of the National University of Asunción (UNA). Previous studies in Paraguay such as Kurita et al. (2020) and Kehler et al. (2021) were conducted on similar situations using the same computational tool.

3. Methods

3.1 Model and Mesh Configuration

In order to create the digital model for the analysis, we have rebuilt it from on-site measurements. Figure 1 shows an actual dental clinic room picture compared to the CAD model. A 2018 commercial version of a software from Dassault Systèmes, SolidWorks, was utilized for modeling and simulation of the environment. The mesh configuration was arranged to obtain enough elements in critical areas such as the boundary layer, orifices, sharp curvatures, and rough surfaces. Minimum boundary layer thickness calculations showed a value of 7.28 mm approximately.

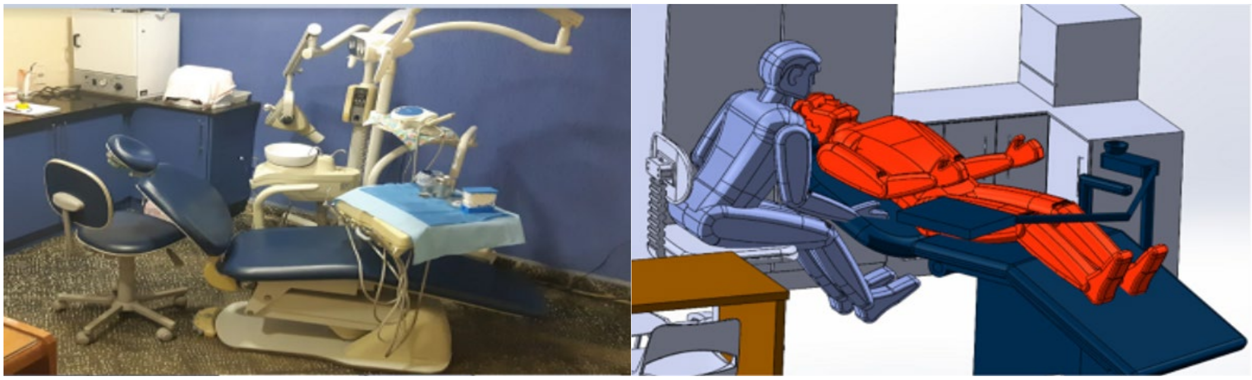

Figure 1. Actual dental practice room and its representative CAD model

This was computed from the mass conservation equation (1), (2) and boundary layer equations (3) and (4), in which, ρ is the air density at 101325 Pascal and 20 Celsius, work fluid is considered incompressible; therefore, density is constant. Cross-sectional areas Ai is measured at inlet and outlet surfaces. Velocities Vi will be calculated from mass conservation equation (1) where inlet flow rates at A and B (see Figure 2) are values from fan and air conditioner specifications, as are outlet flow rates at points C, D, and E. The time derivative component will be neglected since we are analyzing at a steady-state condition.

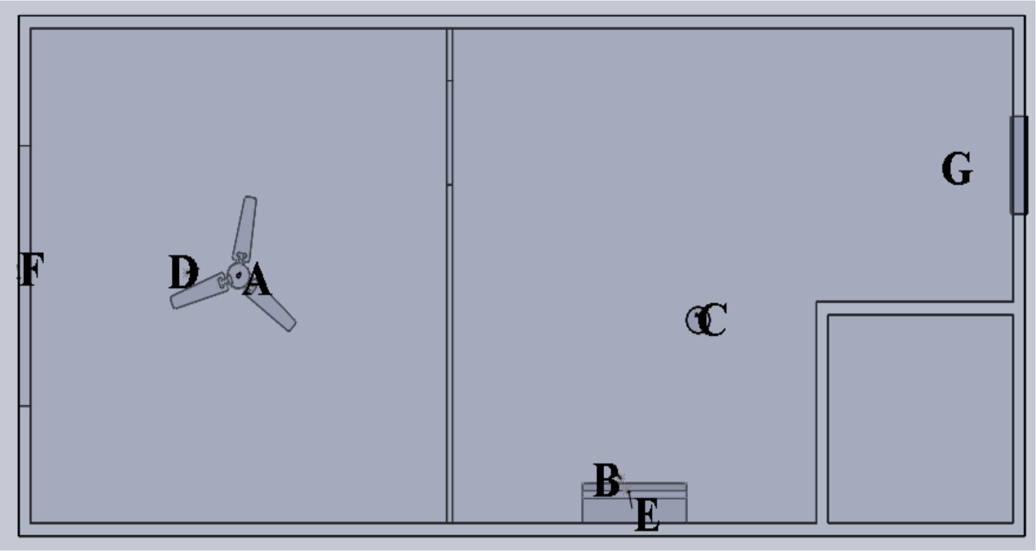

Figure 2. Top view of the practice room with inlet A, B, F and outlet C, D, E and G locations

Depending on the flow regime, i.e., laminar or turbulent, layer thickness δ can be calculated from equations (3) and (4), respectively according to Bernard and Wallace [12]. In this work laminar flow is considered when Rex < 5·103. Transitional to turbulent flow is considered when Rex > 5·103. The Reynolds number on a flat plate is defined as shown in equation (5).

As mentioned previously, we assume incompressible fluid and constant dynamic viscosity μ. Local x values are taken from the worst-case dimensions in inlet and outlet geometry. Then, boundary layer thicknesses at critical locations were calculated.

This calculation gave us the minimum element size we have to implement at all computational domains. Once the calculations finished, CFD simulation mesh configuration was set up to include a minimal number of elements at the critical computational domain. Figure 3 shows the computational domain mesh after calculating the minimum element size. Mesh sensitivity analysis was performed. Mesh sizes were utilized to verify results were not significantly affected, with mesh size values lower than or equal to 3mm.

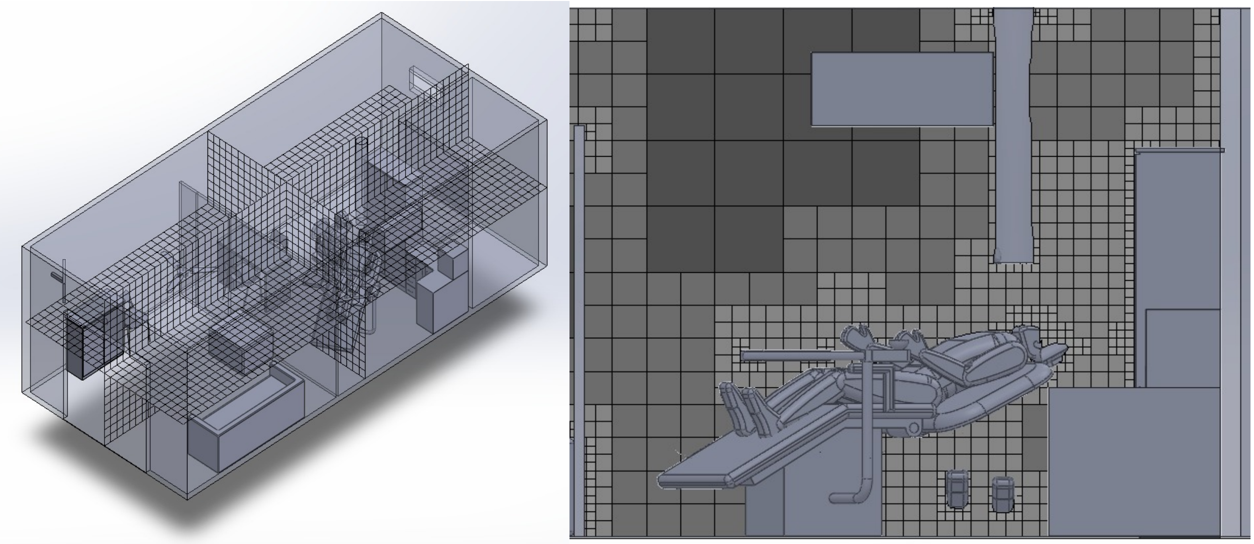

Figure 3. A quick global view of the meshed computational domain

3.2 Boundary Conditions

The fluid considered is air at standard conditions. The office air conditioning system was set to maintain the temperature at 20 Celsius and pressure at 101325 Pascal. Humidity and other environmental parameters are not taken into consideration for this study.

As seen in Fig. 2, there are seven openings in the computational domain. Four are outlets located at the locations C, D, E and G. The other three are inlets located at A, B and F. All other boundary surfaces are considered adiabatic walls with no roughness. These surfaces included human bodies, furniture, instruments, and air conditioning units.
As for the physical conditions, two inlets are set to be at given values from actual measurements and one is set to be at environmental pressure (see Table 2). As for the outlet conditions, three are set to be values from equipment technical specifications and one is set to be at environmental pressure. Table 2 summarizes these boundary conditions. Figure 2 shows the surface locations.

3.3 Room Settings for Simulation

Four different configurations were established on the software to compare airflow plot variations.
Configuration (a), the baseline configuration, was set with inlets A and B enabled as well as outlets D and E. Environment pressures were set at locations F and G. The exhaust system was not considered in this case. Figure 4.
Configuration (b) was modified from (a) by enabling outlet C and locating the exhaust system tube right above the patient’s head. See Figure 5.

Configuration (c) was set with the exhaust system tube displaced 30 cm to the north from its position in configuration (b) with respect to the patient’s head. See Figure 6.

Configuration (d) was set with the exhaust system tube displaced 30 cm to the south from its position in configuration (b) with respect to the patient’s head. See Figure 7.

3.4 Air Exchange Rate Value (ACH)

Air Changes per Hour, also known as ACH, is a parameter that quantifies how efficient the exchange between air from the interior of the control volume and its exterior is. It is defined as the ratio of the outdoor air flow coming into the room per hour, and the room volume.

4. Results and Discussion

4.1 Numerical Results

Table 1, with calculations for boundary layer thickness (δ) using equations (1), (2), (3), (4) and (5), is presented next.

Table 1. Boundary layer thickness (δ)

Surface	Area (m^2)	Velocity (m/s)	Re_x	Thickness (δ) (m)
Inlet A	1,179	0,780	63,208	0.0404046
Inlet B	0,110	1,385	34288	0,0134708
Outlet C	0,200	1,041	31594	0,0072836
Outlet D	1,225	1.385	74148	0,0394936
Outlet E	0,465	2,119	34288	0,0167464
Inlet F	4,06	0,154	31594	0,0828012
Outlet G	0,32	0,067	74148	0,0205788

Table 2, with the utilized boundary conditions for each inlet and outlet is presented next.

Table 2. Boundary conditions

Surface	Condition	Value	Unit
Inlet A	Inlet Volume Flow	3380,2	Cubic meter per hour

Inlet B	Inlet Volume Flow	509,7	Cubic meter per hour
Outlet C	Outlet Volume Flow	254,9	Cubic meter per hour
Outlet D	Outlet Volume Flow	3380,2	Cubic meter per hour
Outlet E	Outlet Volume Flow	509,7	Cubic meter per hour
Inlet F	Environmental Pressure	101325	Pascal absolute
Outlet G	Environmental Pressure	101325	Pascal absolute

Table 3. ACH Value Interpretation - Allen et al. (2020)

ACH	Indications
< 3	Low
3 - 4	Bare minimum
4 - 5	Good
5 - 6	Excellent
> 6	Ideal

4.2 Graphical Results

As it can be observed in Figure 4 on a plot view located 1.2 m above the floor at the doctor's face level, stagnant areas can be spotted from the color scale plot to the doctor's left-back and the patient's sides with configuration (a). Although the air distribution was not inadequate in the area between the doctor and the patient's faces with these settings, there was room for improvement in the general airflow between both people on that level plane. For this case, we got an ACH of 0.74, which is considered low according to Table 3.

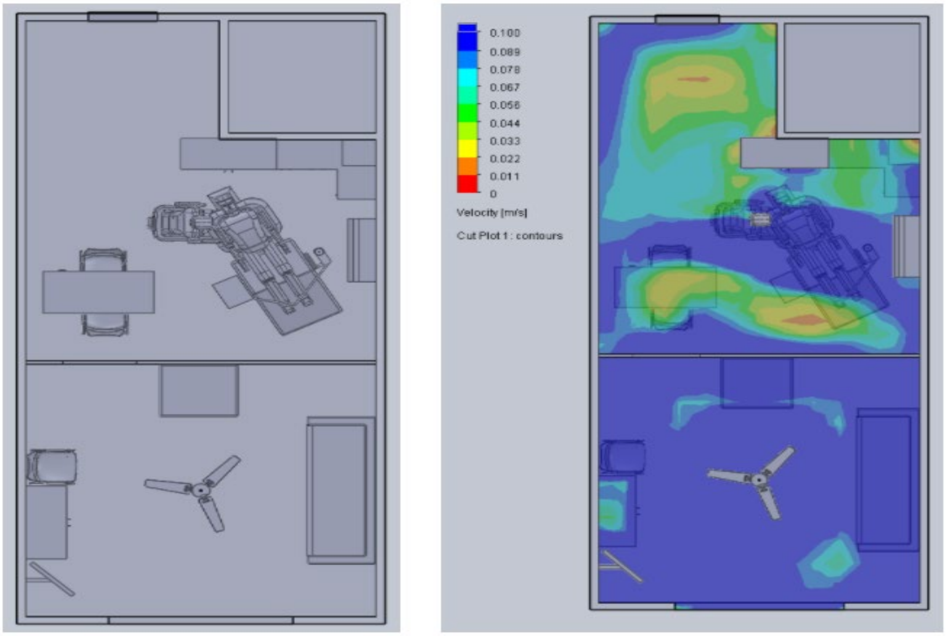

Figure 4. Cut view of the velocity plot 1.2 m above the floor with configuration (a)

When the airflow velocity profile was analyzed on the same level plane with configuration (b), the velocity profile showed high-velocity areas between both people but lower ones behind the doctor. This can be observed in Figure 5.

The air velocity distribution is observed to be better on this layout at said level, which is a favorable outcome compared to the previous case. For this case, we got an ACH of 4.11, which is considered good according to Table 3.

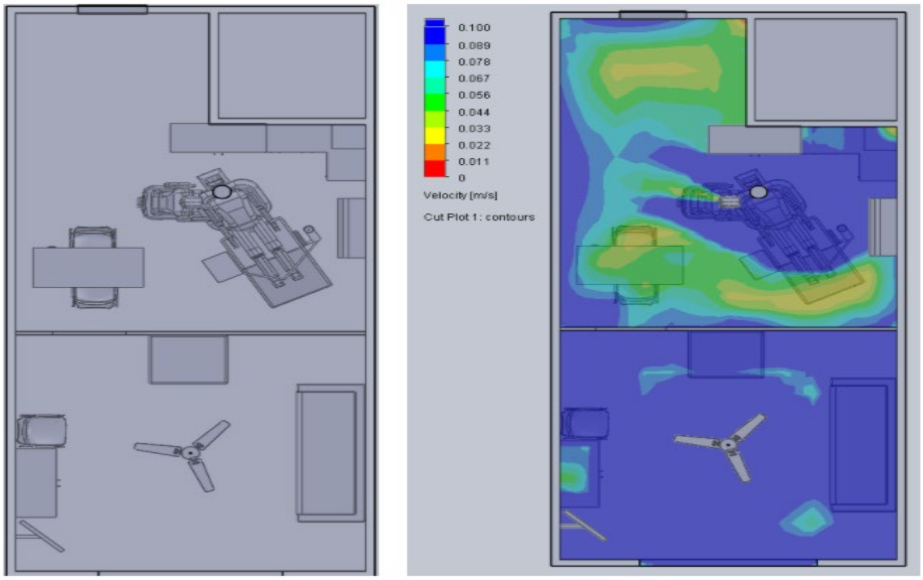

Figure 5. Cut view of the velocity plot 1.2 m above the floor with configuration (b)

Subsequently, the airflow velocity profile was analyzed on the same level plane with configuration (c). An overall better flow distribution can be observed in the room between the doctor and patient and behind them. This can be observed in Figure 6. For this case, we got an ACH of 4.16, which is considered good according to Table 3.

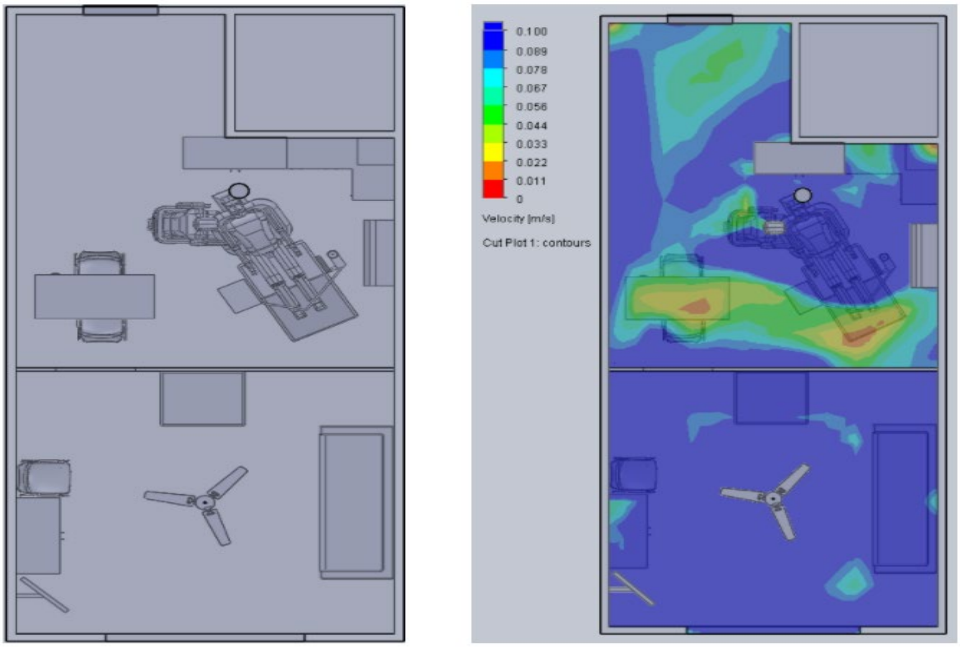

Figure 6. Cut view of the velocity plot 1.2 m above the floor with configuration (c)

Lastly, the airflow velocity profile was analyzed on the same level plane with configuration (d). The velocity profile showed a better air distribution from the back of the doctor to outlet G, but the detriment of the air distribution between doctor and patient. This can be observed in Figure 7. Overall, this configuration does not seem favorable compared to the two previous ones. The ACH value obtained for this case is 4.23, considered good according to Table 3.

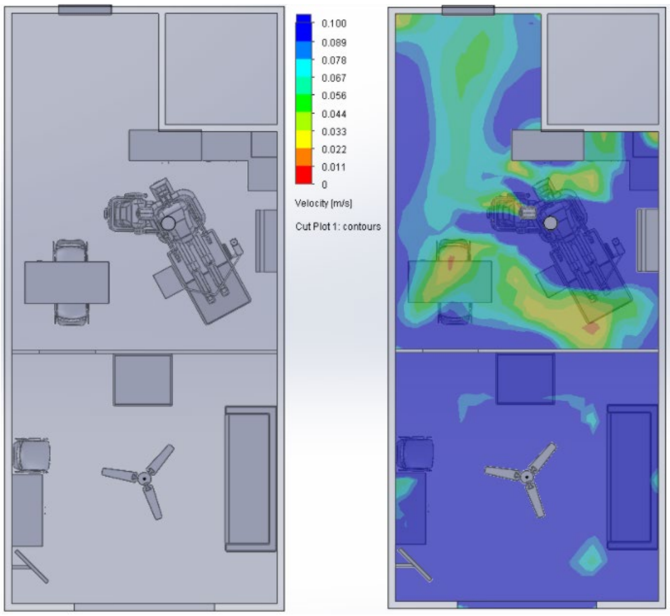

Figure 7. Cut view of the velocity plot 1.2 m above the floor with configuration (d)

5. Conclusion

By analyzing the velocity profile with four different configurations, we can conclude that the new layouts showed a better flow distribution and velocity profile overall and a much better ACH compared to the baseline.

Comparing all simulation cases for configurations (a), (b), (c), and (d), we can conclude that the use of an air exhaust system in a dental clinic is very beneficial from the ACH improvement point of view. The presented case (a) is the baseline case. This original layout was added for the following three simulation configurations.

Of all four configurations, the most beneficial configuration (c) was observed in Figure 8 since the layout showed a better flow distribution between doctor and patient. It delivered a better ACH value than configuration (b).

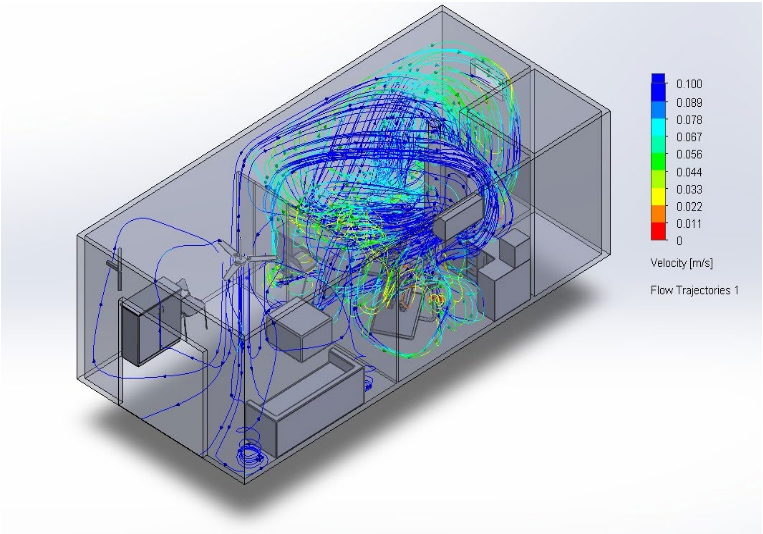

Figure 8. General view of the flow lines for case (c)

Equations

$$\int_{CV} \frac{\partial \rho}{\partial t} dV + \sum_i (\rho_i A_i V_i)_{out} - \sum_i (\rho_i A_i V_i)_{in} = 0 \quad (1)$$

$$\frac{Q_{in}}{4} = A_{out} V_{out} \quad (2)$$

$$\frac{\delta}{x} \approx \frac{5}{Re_x^{1/2}} \quad (3)$$

$$\frac{\delta}{x} \approx \frac{0.16}{Re_x^{1/7}} \quad (4)$$

$$Re_x = \frac{\rho V_i x}{\mu} \quad (5)$$

Acknowledgements

- Facultad de Ingeniería - Universidad Nacional de Asunción (FIUNA)
- Mechanical Engineering Department of FIUNA

References

Ai, Z., Hashimoto, K. and Melikov, A.K. Airborne transmission between room occupants during short-term events: measurement and evaluation. *Indoor air*, vol. 29 no. 4, pp.563-576, 2019.

Abanto, Juan, Daniel Barrero, Marcelo Reggio, and Benoît Ozell. Airflow modeling in a computer room. *Building and Environment* vol. 39, no. 12 pp.1393-1402, 2004.

Allen, J., Spengler, J., Jones, E., Cedeno-Laurent. 5-step guide to checking ventilation rates in classrooms. *Architectural R*ecords, vol. 1, p. 27, 2020.

Bernard, Peter S., and James M. Wallace. Turbulent flow: analysis, measurement, and prediction. *John Wiley & Sons*, 2002.

Gan, Guohui, and Hazim B. Awbi. Numerical simulation of the indoor environment. *Building and Environment* vol. 29, no. 4 pp.449-459, 1994.

Kehler, P., Chaves, C., García, A., Centurion, H., Escobar, A., Lopes, L., Aquino, S., Ferreira, N. and Kurita, J. Ventilation CFD Analysis at a Classroom as a Tool for Air Safety Verification Under COVID19 Context, a Case Study. *Proceedings of ASME International Mechanical Engineering Congress and Exposition* vol. 85666, p. V010T10A056. American Society of Mechanical Engineers, November 2021.

Kurita, J., Limousin, M., Ferreira, N. and Ozuna, J. CFD Analysis on Air Ventilation at a Manufacturing Plant as a Tool for Designing Machine Layout, a Case Study. *Proceedings of the 5th North American International Conference on Industrial Engineering and Operations Management*, Detroit, Michigan, USA, August 10, 2020.

Lee, Ya-Ling, Dachen Chu, Sin-Yi Chou, Hsiao-Yun Hu, Sheng-Jean Huang, and Yung-Feng Yen. "Dental care and infection-control procedures during the COVID-19 pandemic: The experience in Taipei City Hospital, Taiwan." *Journal of Dental Sciences* vol. 15, no. 3 pp.369-372, 2020.

Lin, Yue. Ventilation in the dental clinic: An effective measure to control droplets and aerosols during the coronavirus pandemic and beyond. *The Chinese journal of dental research: the official journal of the Scientific Section of the Chinese Stomatological Association (CSA)* vol. 23, no. 2 (2020): 105-107.

Lu, J., Gu, J., Li, K., Xu, C., Su, W., Lai, Z., Zhou, D., Yu, C., Xu, B. and Yang, Z. COVID-19 outbreak associated with air conditioning in restaurant, Guangzhou, China. *Emerging infectious diseases*, vol. 26, no. 7, p.1628, 2020.

Posner, J. D., C. R. Buchanan, and D_ Dunn-Rankin. Measurement and prediction of indoor air flow in a model room. *Energy and buildings* vol. 35, no. 5 pp.515-526, 2003.

Sekhar, S. C., and H. C. Willem. Impact of airflow profile on indoor air quality—a tropical study. *Building and Environment* vol. 39, no. 3 pp.255-266, 2004.

Yang, Li, and Miao Ye. CFD simulation research on residential indoor air quality. *Science of the Total Environment* vol. 472 pp.1137-1144, 2014.

Biography

Elías Espínola began attending technical school in 2016, earning the degree of electromechanical technician once he graduated in 2016. In 2017, he began his university career in order to obtain a Mechanical Engineering degree at the Universidad Nacional de Asunción. He was introduced to research in 2021 during the mandatory fluid mechanics course at university and began working in that field from that year and on. In 2022, he was chosen as one of the two candidates chosen by the university's research department for an internship program offered by Fermilab, a laboratory that specializes in high-energy particle physics located in Illinois and is expected to finish the program in December 2022. He is a founding member of the ASME Paraguay Student Branch and currently occupies the Vice Chair position.

Vivian González is an undergraduate student of Mechanical Engineering at the Universidad Nacional de Asunción. She became interested in research in 2021 while taking the degree's mandatory fluid mechanics' course. She has been a volunteer for IEEE affinity groups such as Power and Energy Society (PES) and Industrial Applications Society (IAS) since 2018. She was actively involved in research works at the university during 2022. She is a founding member of the ASME Paraguay Student Branch and is set as a presenter for a technical research paper in the 2022 International Mechanical Engineering Congress & Exposition (IMECE) in Columbus, Ohio.

Liz Esquivel is an undergraduate student of Mechanical Engineering at the Universidad Nacional de Asunción. Her interest in research piqued when she was invited to participate in a research project in 2020, and she has been working on that field since that year. She is currently a Teacher Assistant for an elective AP Fluid Mechanics course. She was chosen as a candidate for an internship program offered by Fermilab, a laboratory that specializes in high-energy particle physics located in Illinois and is expected to begin the program in September 2022 and finish it in December of the same year. She is a founding member of the ASME Paraguay Student Branch and currently occupies the position of treasurer.