Analysis of ventilation in Central corridor in UPJR building UPJR using CFD

CISNEROS-RIVAS, Maria Del Carmen, RAMIREZ-VICTORIA, Silvana Del Carmen, PRIETO-SANCHEZ, Juan Carlos and MURRIETA-LUNA, Edilberto

Abstract

To date, the parameters associated with the transmission of SARS-CoV-2 virus and its disease COVID- 19 have not been fully studied [1], [2], [3]. It is known that transmission occurs through dispersed droplets that are exhaled, which contain the virus and are suspended in the air, so characterizing them is complex since many parameters that influence hydrodynamics must be considered. In this study, a CFD analysis of the ventilation in a central corridor in the UD1 building at UPJR was performed.

Introduction

Physical measurements were made in the corridor, both of the dimensions, as well as of the air velocity, ambient temperature and pressure drop. CAD models were generated in SolidWork based on the original corridor (Figure 1 and Figure 2) and exported to ANSYS Meshing to be meshed and simulated in ANSYS Fluent.

Materials and methods

The first level of the building has 10 classrooms with capacity for 30 students, two computer centers, restrooms and a balcony in the central part. The air intake velocity measurements were made with an anemometer, obtaining that the air intake is only at the ends of the building. The doors and windows are used to supply air to the classrooms, so they are considered outlets in this study, as well as the balcony.

Figure 1 UPJR corridor

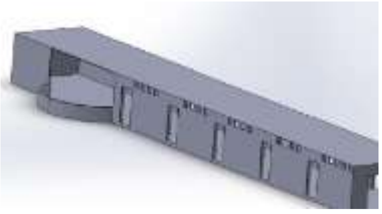

Figure 2 SolidWorks model

The generated mesh was of 69233 nodes and 348049 elements, in the form of hybrid tetrahedra, see Figure 3. Atmospheric air concentration was considered with 78.62 % N2, 20.84 % O2, 0.04 % CO2 and 0.5 % H2O. Table 1 shows the boundary and initial conditions considered for the simulation.

	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	
[illegible]	[illegible]	[illegible]	
[illegible]	2.30 x 2.05	velocity inlet	1.2
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	

Table 1 Case studies

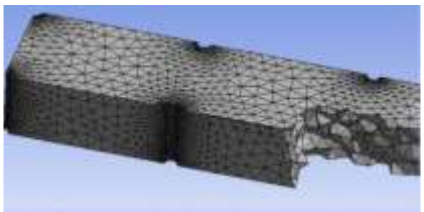

Figure 3. Meshed Model.

Results

Figure 4 shows the streamlines, in which it can be seen that the exit is through the access doors to the classrooms and through the central balcony; the windows show very little air exit. The average velocity inside the corridor was 0.8 m/s. The main air outlet is the balcony in the middle of the building. When analyzing the turbulence generated, it is more prevalent in the classroom doors, with the greatest effect of this phenomenon in the computer centers.

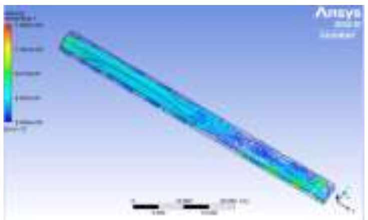

Figure 4 Power lines

The pressure contours in the corridor are shown in Figure 5. It is shown that the highest pressure drop occurs at the entrance to computer center 2, which is due to the fact that the access is in front of the side entrance door. The air movement is by natural convection.

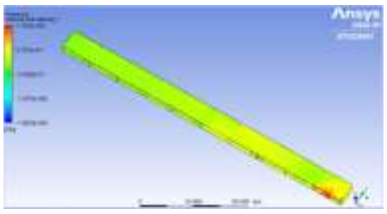

Figure 5. Pressure contours.

The lowest pressure drop occurs in the classrooms located near the main door, so it can be deduced that they are the least ventilated. This phenomenon is due to the fact that the entrances to the doors are perpendicular and the current lines generated are parallel to the doors.

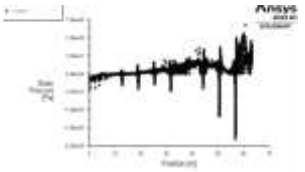

Figure 6. Pressure vs. Position.

A pressure versus position graph was generated (Figure 6). The analysis of the graph shows peaks, which occur at the access doors to the classrooms, with the computer centers and central classrooms having greater ventilation.

The velocity at the inlet of the gates averaged 1.2 m/s, which is the highest velocity reported. The graph in Figure 6 shows that the highest pressure drop occurs when the velocity is higher, with the graph showing a parabolic growth.

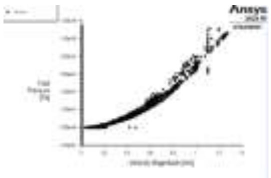

Figure 7. Pressure vs. Velocity

Conclusions

A model was developed to perform a hydrodynamic analysis by means of CFD, in the central corridor of the Polytechnic University of Juventino Rosas, in order to determine the ventilation inside the corridor, thus meeting one of the recommendations made by the Federal Government, which indicates that good ventilation should be provided inside closed buildings. The air distribution within the corridor is heterogeneous, so one could consider minimizing the dependence of ventilation on natural convection by increasing efficiency through forced convection.

A more rigorous analysis is required to consider factors that affect ventilation, such as wind direction, temperature and people inside the corridor.

References

[1] J.B. Casademont, Estudio Hidrodinámico por CFD del casco de una lancha motora (2014) 001010.
[2] G. Buonanno, L. Stabile, L. Morawska, Estimation of airborne viral emission: quanta emission rate of SARS-CoV-2 for infection risk assessment. Environ. Int. 141 (2020) 105794, https://doi.org/10.1016/j.envint.2020.105794.
[3] J. Komperda, A. Peyvan, D. Li, B. Kashir, A. L. Yarin, C. M. Megaridis, P. Mirbod, I. Paprotny, L. F. Cooper, S.Rowan,
[4] P. Shrestha, J. W. DeGraw, M. Zhang, X. Liu. (2021). Multizonal modeling of SARS-CoV-2 aerosol dispersion in a virtual office building. Building and Environment 206 (2021). 108347. https://doi.org/10.1016/j.buildenv.2021.108347.
[5] R. F. Ballestero-Tajadura, J. González-Pérez, J. M. Fernández-Oro y K. M. Argüelles-Diaz. Monografías de Mecánica de Fluidos: Técnicas numéricas de Mecánica de Fluidos. Editorial Gijón (2003). Universidad de Oviedo, España. g. https://doi.org/10.1063/5.0043934.[6] (J. E. Hall, Tratado de fisiología medica: Decimotercera edición (2016).

Contact: CISNEROS-RIVAS, Maria Del Carmen **E-mail: 319030037@upjr.edu.mx** **Project website: https://www.ecorfan.org**

DOI: 10.35429/S.2022.1.4.15.25

RENIECYT 1702902 CONACYT