CFD Analysis on Air Ventilation at a Manufacturing Plant as a Tool for Designing Machine Layout, a Case Study

Jorge Kurita and Mirna Limousin
Department of Industrial Engineering
Universidad Nacional de Asunción
San Lorenzo, Central 2160, PARAGUAY
jkurita@fiuna.edu.py, mlimousin@fiuna.edu.py

Nicolas Ferreira
Mechatronics Engineering Department
Universidad del Cono Sur de las Américas
Asunción, PARAGUAY
nferreira@alumno.ucsa.edu.py

Jose Ozuna
Plant Supervisor
FLUODER S. A.
Villeta, PARAGUAY
jozuna@fluoder.com.py

Abstract

The COVID-19 pandemic hit industry worldwide. One of the key factors to spread this virus is the airflow in an enclosed environment. Proper air ventilation should minimize, in a combination of the use of personal protective equipment, the spread of this airborne virus. Stagnant air locations in a plant are potential sources of contaminant concentration. Therefore, it is important to identify these hot spots first. Then, by moving machines, equipment, and people working areas, it will allow proper airflow in order to avoid the formation of stagnant points. This type of CFD analysis should set the benchmark for future plant machine layout standards under the context of this pandemic. The work presented here is a case study on a small chemical industry in Paraguay. Results of CFD analysis on air ventilation by changing the working area showed significant improvement in ventilation. This allowed industry management to reopen factories keeping its employees safe confidently.

Keywords
Covid-19, CFD, Ventilation, Layout and Stagnation

1. Introduction

Covid-19 pandemic hit hard worldwide in early 2020. As the virus spread reached every corner of the planet, the Paraguay industry sector was not exempt from this situation. Local government started a curfew in the second week of March, which forced the industry to stop production. Three months later, as a plan of an economic reactivation, the Paraguayan government resolved to reopen stores and factories gradually. This leads to mandated hygiene procedures to allow workers to be back at manufacturing plants. Among these measures is the mandatory use of face masks. Intensive research on virus spread mechanisms was conducted worldwide. One example is the work by Lu et al. (2020). In this publication, a case study on the Covid-19 outbreak associated with air conditioning in a restaurant in Guangzhou, China, was presented. This analysis showed how airflow paths were a critical influence of virus spread, and dining table layout configuration played an essential factor in this mechanism.

Another experimental research work, such as the one done by Ai et al. (2019), conducted measurements and evaluations on an airborne transmission between room occupants during short term events. Here, it was clear that person to person proximity significantly increases the chances of contagions.

Morawska et al. (2020) proposed an engineering level control to reduce the environmental risks for airborne transmission. It was noted that the indoor air cycle through open windows was recommended as part of a solution.

Inspired on a more applied work such as the one from Posner et al. (2003), Gan et al. (1994), Tian et al. (2006) where indoor airflow experimental research showed flow behavior when an object is a place between inlet and outlet, a solution to improve indoor airflow is proposed for a specific case in the local industry. This is a chemical industry called FLUODER S. A.

2. Literature Review

Air ventilation analysis is often applied to a closed environment where human activity is involved. To ensure proper airflow is presented either, guaranteeing a contaminant free human air intake or removing undesirable gas, a CFD analysis is performed. Several authors such as Abanto et al. (2004) have already worked on airflow modeling, in this case, a computer room facility. His findings on the influence of the shape of the ceiling diffuser on the air distribution in the room gave us an indication of how important the indoor environment configuration is for proper ventilation. Simulations on residential indoor air quality by Yang et al. (2014) showed certain areas of a room do not have adequate circulation; hence, a potential point of contaminant concentration is developed. A study on the impact of airflow profile in indoor air quality conducted by Sekhar et al. (2004) showed different gas concentrations in different room areas; this is CO2, CO, bacteria, fungi, etc.

Fundamental theoretical computation analysis, such as Direct Numerical Simulation (DNS) on the zero-equation turbulence model for indoor airflow, have been conducted by Chen et al. (1996). Earlier work on fundamentals of flow simulation such as Kulmala (1993), Awbi (1989) made the path for further developments in applied computational tools. For this analysis, commercial CFD software is used. Flow Simulation from SolidWorks was utilized on a regular PC at the faculty of engineering Universidad Nacional de Asuncion (UNA).

3. Model and Mesh Configuration

In order to create the digital model for this analysis, we have rebuilt it from actual in situ measurements. Initially, we intended to use a 3D scanner; however, due to the low-resolution output, we have decided not to go through this approach. Figure 1 is showing the actual room picture compared to the CAD model. For this analysis, commercial software from Dassault Systemes and STAR CCM + was used. The mesh configuration was set 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 3mm approximately. This was computed from the mass conservation equation (i) and boundary layer equations (ii) and (iii).

Figure 1: Actual room picture vs CAD model picture

$$\int_{CV} \frac{\partial \rho}{\partial t} d\mathcal{V} + \sum_{i} (\rho_i A_i V_i)_{out} - \sum_{i} (\rho_i A_i V_i)_{in} = 0$$
… (i)

Where ρ is the air density at 101325 Pascal and 20 Celsius, work fluid is considered incompressible; therefore, density is constant. Cross-sectional areas A_i are measured at inlet and outlet surfaces. Velocities V_i will be calculated from mass conservation equation (i). The time derivative component will be neglected since we are analyzing at a steady-state condition.

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

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

Depending on the flow regime, i.e., laminar or turbulent, layer thickness δ can be calculated from equations (ii) and (iii), respectively. In this work laminar flow is considered when $10^3 < Re_x < 10^6$. Turbulent flow is considered when $10^6 < Re_x$. Reynolds number is defined as:

$$Re_x = \frac{\rho V_i x}{\mu}$$
… (iv)

Again, as mentioned before, we assume incompressible fluid and constant dynamic viscosity μ. Local x values are taken from the most extended dimensions in inlet or outlet geometry.
Then, boundary layer thickness at critical locations were calculated. This can be observed in Table 1.

Table 1. Boundary layer thickness values

Surface	**Area (A_i) in mm^2**	**Re_x**	**Thickness (δ) in mm**
Door 1	16462	4539	3
Door 2	16462	4539	3
Exhaust 1	31416	2379	4.1

Exhaust 2	125664	595	8.2

This calculation gave us the minimum element size we have to implement at all computational domains.
After proceeding with these calculations, CFD simulation mesh configuration was set up to include a minimal number of elements at the critical computational domain. Figure 2 is showing this computational domain mesh after calculating the minimum element size.

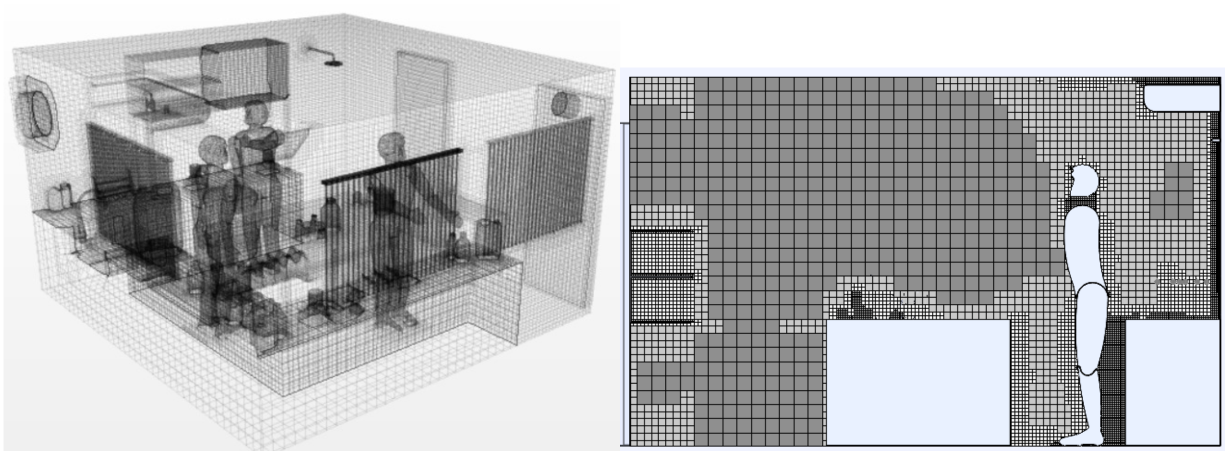

Figure 2: A quick global view of meshed computational domain.

4. Boundary Conditions

The fluid considered here is air at standard conditions. This is the temperature at 20 Celsius and pressure at 101325 Pascal. Humidity and other environmental parameters are not taken into consideration for this study. There are four openings in the computational domain. Two are outlets located at the exhaust fans. The other two are inlets from the existing clearances at the two doors. The rest of the boundary surfaces are considered adiabatic walls with no roughness. These surfaces included human bodies, furniture, instruments, air conditioning units, and curtains. As for the physical conditions, all inlets are set to be at environment pressure, i. e., 101325 Pascal (Pa). The two outlet conditions are 60 cubic feet per minute (CFM), the actual exhaust fan technical specification. Table 2 summarizes these boundary conditions. Figure 3 shows the surface locations.

Table 2. Boundary conditions

Surface	Condition	Value	Unit
Door 1	Environment pressure	101325	Pascal absolute (Pa)
Door 2	Environment pressure	101325	Pascal absolute (Pa)
Exhaust 1	Volume Flow Rate	60	Cubic Foot per Minute (CFM)
Exhaust 2	Volume Flow Rate	60	Cubic Foot per Minute (CFM)

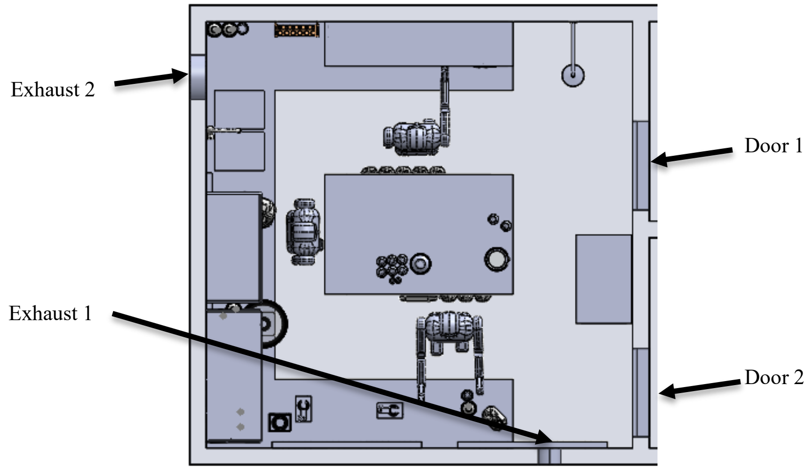

Figure 3: Top view of the room with surface descriptions.

5. Results and Discussions

As it can be observed in Figure 4, at the plot view located from 0.1 m above the floor, the stagnant areas are clearly spotted from the color scale plot.

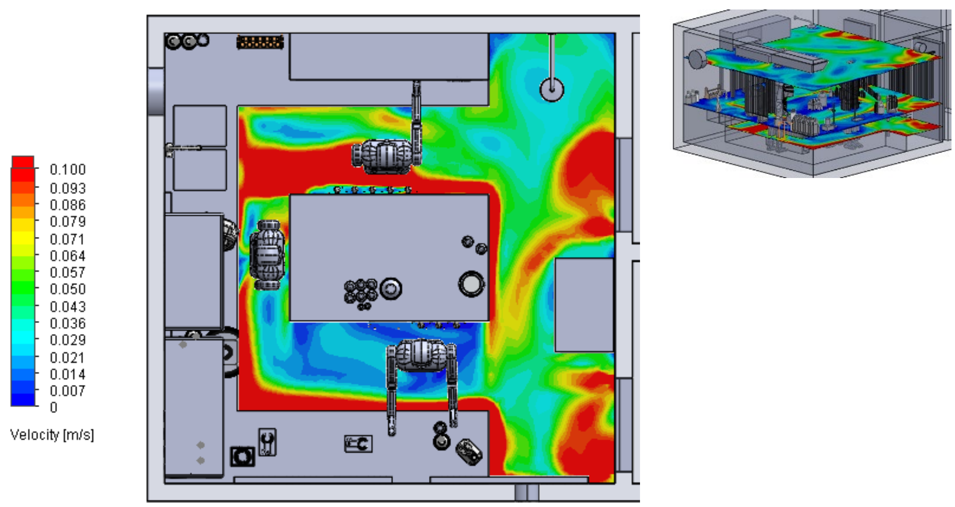

Figure 4: A cut view of the velocity plot from 0.1 m above floor.

When the airflow velocity profile is analyzed at 1 m above the floor, just 0.1 m above the lab tables, the velocity profile showed low-velocity areas over these surfaces. This behavior is expected since the viscous air forces over solid static surfaces will undoubtedly show this effect. This can be observed in Figure 5.

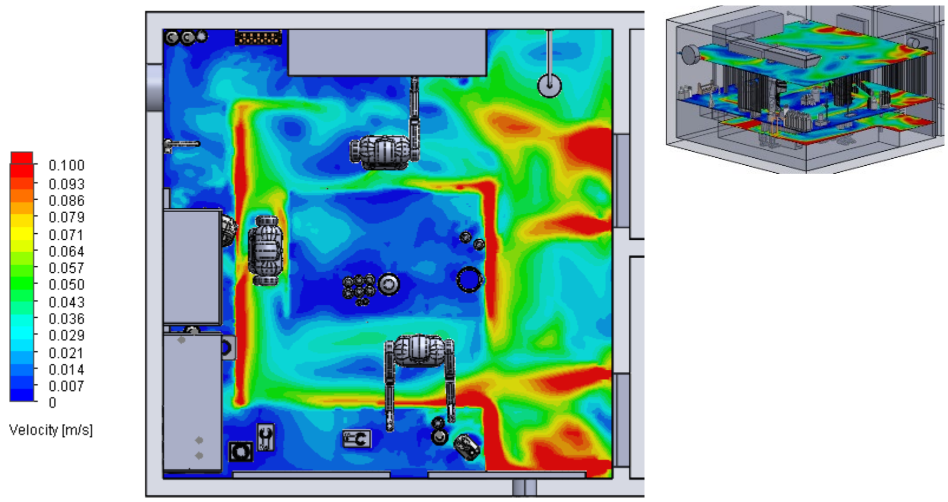

Figure 5: A cut view of the velocity plot from 1m above floor.

When the airflow velocity profile is analyzed at 1.7 m above the floor, roughly at the human head level, the velocity profile showed low-velocity areas between two adjacent individuals. See Figure 6. However, the absolute values are not as low as at the 1 m level.

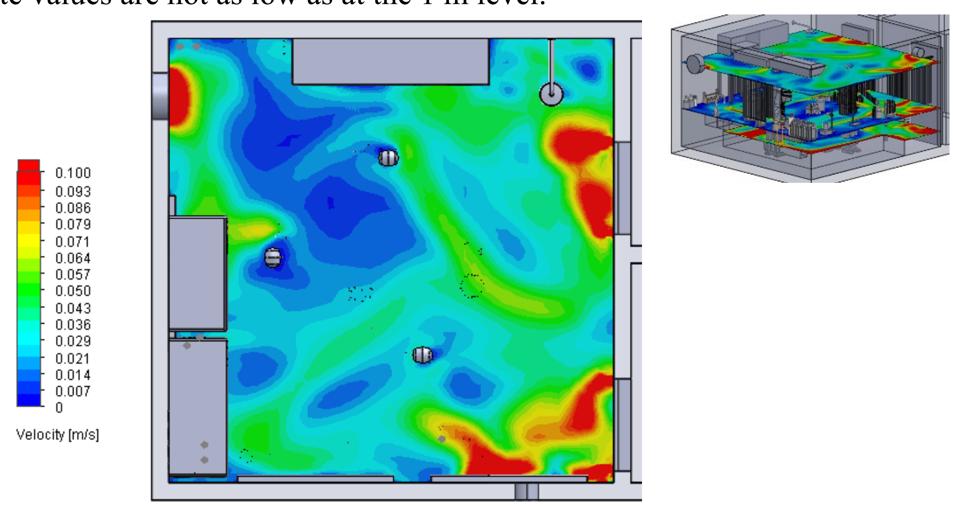

Figure 6: A cut view of the velocity plot from 1.7 m above floor.

6. Conclusions

By analyzing the velocity profile at different planes measured from the floor, we can conclude that the worst location where we can see low velocities and stagnant areas is at the table level. While at the other levels, we see reasonable fluid velocities. When the head level is analyzed, the low-velocity areas were observed between two consecutive persons. This confirmed the importance of social distancing. Finally, this layout proved to be better than the original, seen in Figure 7. Due to the table orientation about the exhaust fans, this configuration showed a much more low-velocity spot. Figure 8 is a cut plot of the velocity profile at 1 m levels from the floor.

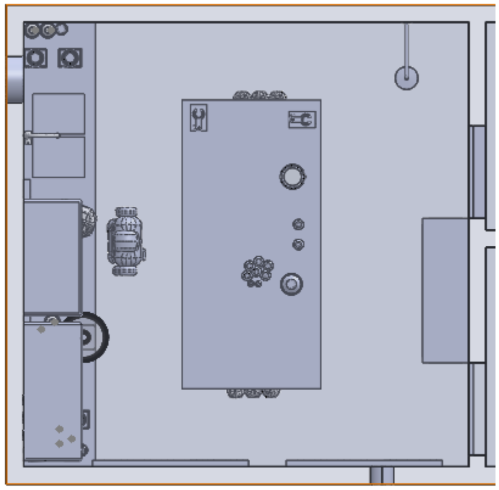

Figure 7: Top view of the room with original layout.

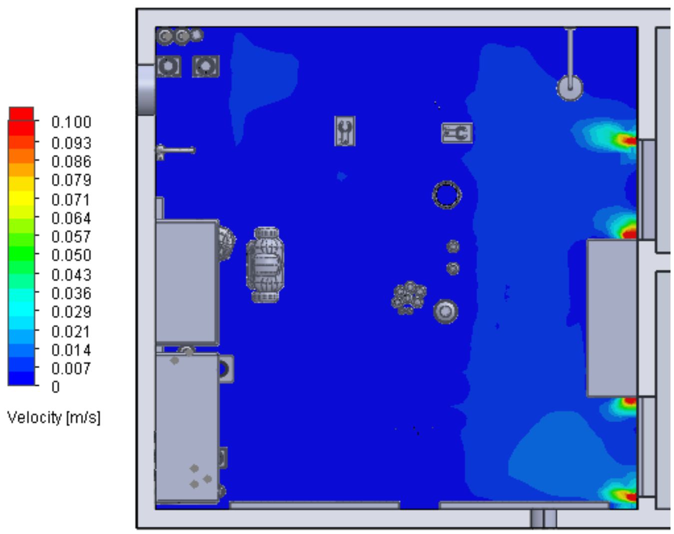

Figure 8: A cut view of the velocity plot from 1m above floor.

Acknowledgements

- FLUODER S.A.
- Faculty of Engineering at Universidad Nacional de Asuncion
- Universidad del Cono Sur de las Américas

References

Lu, J., Gu, J., Li, K., Xu, C., Su, W., Lai, Z., Yang, Z. *COVID-19 Outbreak Associated with Air Conditioning in Restaurant*, Guangzhou, China, 2020.

Ai, Z., Hashimoto, K., 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.

Morawska, L., Tang, J. W., Bahnfleth, W., Bluyssen, P. M., Boerstra, A., Buonanno, G.,Cao, J., Dancer, S., Floto, A., Franchimon, F., Haworth. C., Hogelin, J., Isaxon, C., Jimenez, J.L., Kurnitski, J., Li, Y., Loomnas, M., Marks, G., Yao, M., A. *How can airborne transmission of COVID-19 indoors be minimized?* Available: https://www.sciencedirect.com/science/article/pii/S0160412020317876, September 2020.

Posner, J. D., Buchanan, C. R., Dunn-Rankin, D. *Measurement and Prediction of Indoor Air Flow in a Model Room*. Energy and buildings, vol.35, no.5, pp. 515-526, 2003.

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

Tian, Z. F., Tu, J. Y., Yeoh, G. H.,Yuen, R. K. K. *On the Numerical Study of Contaminant Particle Concentration in Indoor Airflow*. Building and Environment, vol.*41*, no.11, pp. 1504-1514, 2006.

Abanto, J., Barrero, D., Reggio, M., and Ozell, B., *Airflow Modelling in a Computer Room*. Building and Environment, vol.39, no.12, pp. 1393-1402, 2004.

Yang, L., Ye, M. *CFD Simulation Research on Residential Indoor Air Quality*. Science of the Total Environment, vol. 472, pp. 1137-1144, 2014.

Sekhar, S. C., Willem, H. C. *Impact of Airflow Profile on Indoor Air Quality - a Tropical Study*. Building and Environment, vol.39, no.3, pp. 255-266, 2004.

Chen, Q., Xu, W. *A Zero-Equation Turbulence Model for Indoor Airflow Simulation*. Energy and buildings, vol.28, no.2, pp. 137-144 1998.

Kulmala, I. *Numerical Calculation of Air Flow Fields Generated by Exhaust Openings*. The Annals of Occupational Hygiene, vol. 37, no 5, pp. 451-468, 1993.

Awbi, H. B. *Application of Computational Fluid Dynamics in Room Ventilation*. Building and environment, vol. 24, no 1, pp. 73-84, 1989.

Biographies

Jorge Kurita is a Research Professor at the Universidad Nacional de Asuncion (UNA) in Paraguay. In 2004 Dr. Kurita was granted the Fulbright scholarship to attend a graduate program on Mechanical Engineering at Michigan Technological University. From 2011 Dr. Kurita worked as a Senior Development Engineer, in the competitive automotive industry, at Filtran LLC, located in Des Plaines, Illinois. His experience as an experimental researcher helped Filtran to develop special testing techniques never implemented before on filtration systems. Also, Dr. Kurita worked in the Computer-Aided Engineering (CAE) group, contributing to improving simulation techniques to help develop state of the art filtration systems such as the SmartMedia™ technology. One outstanding evidence from this successful work is Filtran's achievement of the Global Special Award, for the "Contribution to Quality Enhancement by Development of the SmartMedia™ Strainer" from Jatco Ltd., a Japanese company leading in the supply of transmission systems to automakers worldwide. Another recognition Filtran LLC received was about on his contribution to the development of the GM 6T70; this is, the transmission filter for the Chevrolet Traverse. This product, as well as the GM X39F, gave Filtran LLC the 2015 GM Supplier Quality Excellence Award.

Mirna Limousin. She is an industrial engineering undergraduate student from the Faculty of Engineering at the Universidad Nacional de Asunción. Ms. Limousin worked as an assistant in Security and Health at Lean Management Paraguay. Currently, she is a volunteer member of the IEEE Industry Applications Society of the Universidad Nacional de Asuncion Student Brach.

Nicolas Ferreira. Ferreira is an undergraduate student at the Mechatronics Department in Universidad del Cono Sur de las Americas. He has worked as a research assistant for 3K Engineering consulting firm in Asuncion, Paraguay. His skills in the use of simulation software were useful during his internship at the Paraguay Space Agency.

Jose Ozuna. He is a Chemical Engineering graduate from Universidad Nacional de Asunción (UNA). Also, Mr. Ozuna is a Civil Engineering graduate from UNA. He holds an MBA from Escuela de Administración de Negocios (EDAN). From 1982 to 1985, Mr. Ozuna started working as an intern at MEGA INGENIERIA S. R. L. Later, he joined FUODER S. A. as the project manager, and from 1991 to date he was appointed as Plant Manager. Also, from 1997, he is affiliated to CALDETEC INGENIERIA S. R. L. as a project manager.