NUMERICAL STUDIES AND CFD SIMULATION OF NATURAL VENTILATION ON A TYPICAL SCHOOLROOM: EVALUATION OF TURBULENCE MODELS AND COMPARISON OF VENTILATION MECHANISMS.

E. Lencastre[1]*, M.S.C.A. Brito[2], S.I. Sousa[2], M. Tomé[3], RJ. Santos[1]

1: Laboratory of Separation and Reaction Engineering - Laboratory of Catalysis and Materials (LSRE-LCM), Faculty of Engineering ,University of Porto, Rua Dr. Roberto Frias, Porto, 4200-465, Portugal.
e-mail: erlencastre@fe.up.pt, rsantos@fe.up.pt

2: LEPABE — Laboratory for Process Engineering, Environment, Biotechnology and Energy, Faculty of Engineering, University of Porto, Rua Dr. Roberto Frias, Porto, 4200-465, Portugal

2: proMetheus, Instituto Politécnico de Viana do Castelo, Rua da Escola Industrial e Comercial de Nun'Alvares, 4900-347, Viana do Castelo, Portugal.

Keywords: Indoor Ventilation**,** Natural Ventilation, Indoor air quality, Computational Fluid Dynamic simulation, Turbulence models

Abstract

Indoor Air Quality has an enormous impact on our daily lives since it significantly affects our health, well-being, and comfort. Since, in the last few decades, the time that people spend indoors has significantly increased the interest in understanding this type of environment, namely regarding indoor air quality, has been increasingly studied
In this work, CFD simulations were used to study indoor natural ventilation in primary classrooms. The influence of the simulation setup, namely boundary conditions and turbulence models are analyzed as well as the effect of some ventilation parameters. The efficiency of natural ventilation is compared to mechanical ventilation using a Corsi-Rosenthal box type filter. The ventilation power was studied using tracer simulations, using CO_2 has the tracer gas.

The turbulence models studied were the standard k-ε, realisable k-ε, RNG, SST k-ω as well as LES and SBES. It was found that the RANS models produce similar atmospheric flows, however the main difference reported was in the flow through the openings. Comparing with the result produced by LES and SBES the SST $k-\omega$ was considered to be the most accurate of the RANS models. Figure 1 shows surface streamlines traced near and inside the room at planes $y = 1{,}5$ m and $z = 25$ m. It was also found that the wind direction has a significant role in the ventilation capacity, reaching a maximum value of total outflow at approximately $\theta \approx 25°$. From the tracer simulations, it was found that the indoor air was renewed by two mechanisms: convection and turbulent mixing, being the later the more significant. For the conditions simulated, the pollutant removal by mechanical ventilation was much less effective. However, natural ventilation will depend in plethora of factors that reduce its viability to be a reliable ventilation mechanism along, let's say, one year of different atmospheric conditions.

DOI: https://doi.org/10.57910/ipvc-prometheus-3788ccs2023
ISBN: 978-989-9141-05-6

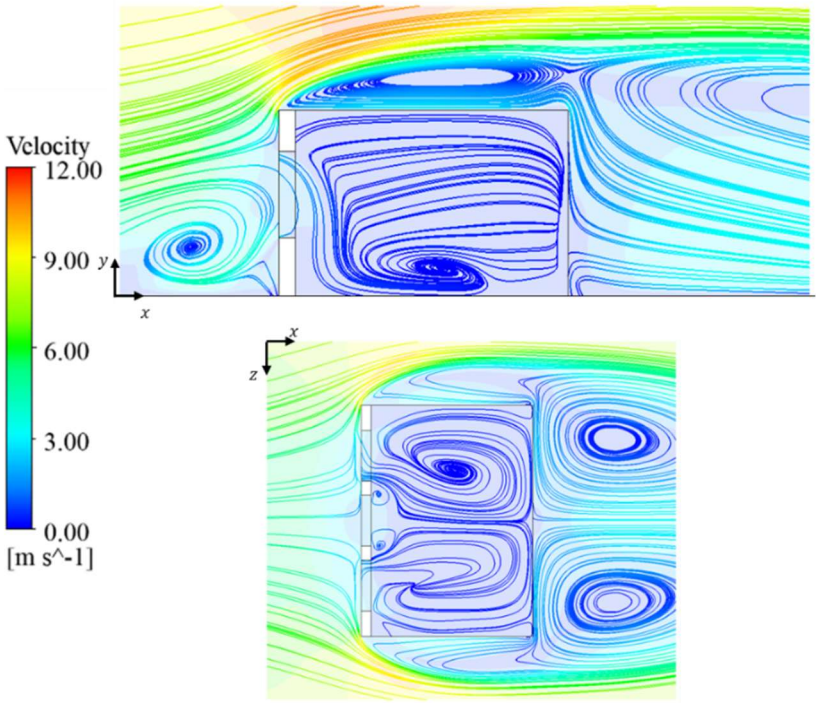

Figure 1 – Surface streamlines near and inside the room at planes y=1,5 m (bottom) and z=25 m (top)

Finally, a new approach for these types of problems was studied, it is a halfway between the coupled and decoupled approach. It consists in first running a simulation in a larger domain, simulating both indoor and exterior air flow and using it to interpolate the boundary profiles for a new smaller domain. More specifically, the first domain included indoor air and external atmosphere and then CFD data generated at the openings (windows) were used to seed the inlet/outlet conditions to simulate only the internal domain of the school room. This new approach reduced the computation cost of the simulations approximately 75%, while maintaining fairly good accuracy.

Acknowledgments

This work was financially supported by LA/P/0045/2020 (ALiCE), UIDB/00511/2020 and UIDB/50020/2020 and UIDP/50020/2020 (LSRE-LCM) funded by national funds through FCT/MCTES (PIDDAC); Project 2SMART - engineered Smart materials for Smart citizens, with reference NORTE-01-0145-FEDER-000054, supported by Norte Portugal Regional Operational Programme (NORTE 2020), under the PORTUGAL 2020 Partnership Agreement, through the European Regional Development Fund (ERDF).

References

Richards, P. J. and S. E. Norris (2011). "Appropriate boundary conditions for computational wind engineering models revisited." Journal of Wind Engineering and Industrial Aerodynamics **99**(4): 257-266. https://doi.org/10.1016/j.jweia.2010.12.008

Franke, J et al. (2004). "Recommendations on the use of CFD in wind engineering." Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics.

Warren, P. R. and L. Parkins (2013). Single-sided Ventilation through open windows. Link.

Gough, Hannah L. et al. (2020) Evaluating single-sided natural ventilation models against full-scale idealised measurements: impact of wind direction and turbulence. Building and Environment, 170. 106556. doi: https://doi.org/10.1016/j.buildenv.2019.106556 .