CWE-8
June 7-11, 2026
London, ON, Canada

Exploring the impact of Reynolds-Averaged Navier-Stokes solvers on the indoor modelling of a breathing jet

Nadine Hobeika[a], Philomena M. BLuyssen[b], Clara García-Sánchez[c]

[a]Delft University of Technology, Delft, the Netherlands, N.Hobeika@tudelft.nl
[b]Delft University of Technology, Delft, the Netherlands, P.M.Bluyssen@tudelft.nl
[c]Delft University of Technology, Delft, the Netherlands, C.Garcia-Sanchez@tudelft.nl

Summary

To support performance-based ventilation-based design, accurate yet efficient numerical simulations are important tools for architects and building engineers. Steady-state Eulerian-Eulerian Reynolds-Averaged Navier-Stokes (RANS) simulations are commonly used for their capabilities to accurately model the mean flow behaviour while reducing computational cost. Indoor air quality studies have typically used three different physics assumptions to model airflows: isothermal and incompressible flow, thermal and incompressible flow, and thermal and compressible flow. However, studies employ different physics assumptions with different solvers to model the same respiratory event, making it challenging for architects and building engineers to select the appropriate physics configuration for their application. Therefore, this study aims to compare these three solvers to support the simplified inclusion of breathing individuals in ventilation-based design assessments. To validate the modelled breathing jet, we acquire velocity magnitude data along the breathing jet of a manikin head in a controlled test chamber.

Keywords: *Validation, CFD, measurements, manikin, breathing jet*

1 INTRODUCTION

Numerically assessing indoor ventilation performance involves a trade-off between computational cost and accuracy. Among the numerical methods used for indoor ventilation, computational fluid dynamics (CFD) simulations remain a popular tool for assessing the impacts of different ventilation systems and room layouts on indoor air quality (IAQ) (Hobeika et al., 2023). This is becoming relevant for architects and building engineers, as building authorities are increasingly adopting ventilation-performance regulations (Leprince et al., 2023). These practitioners require accurate yet fast tools that can support the design process. Many studies have investigated steady-state Eulerian-Eulerian Reynolds-Averaged Navier-Stokes (RANS) simulations, which employ turbulence models that capture the essential effects of turbulence on mean flow behaviour at low computational cost (Zhang et al., 2007). Three types of flow physics assumptions are commonly used to study ventilation and dispersion indoors: isothermal incompressible flow (van Hooff et al., 2013), thermal incompressible flow (Angelova et al., 2021), and thermal compressible flow (Muthusamy et al., 2021). Although each of these flows assumes different physical properties about the indoor flow, they can be used interchangeably, making it challenging for architects and building engineers to select the appropriate assumptions for their design assessment. Therefore, this study aims to investigate these three flow types to efficiently model breathing jet decay, thereby developing a simplified yet representative approach for incorporating breathing individuals into ventilation simulations during design assessment.

2 METHODS

To validate the isothermal incompressible solver, we conducted measurements in one of the test chambers at the SenseLab of the Delft University of Technology (Bluyssen et al., 2018).

2.1 Test chamber set up

The test chamber is a 3.73 m x 2.22 m x 2.12 m box with one inlet located on the bottom of the back wall and an overflow outlet at the top of the opposite wall (See Figure 1). We also placed a manikin head on a tripod and connected it to a ventilator to reproduce human exhaling cycles. We used three hot-sphere anemometers mounted on a line-following robot to measure the air velocity magnitude, with one probe (p2) placed 1.1 m above the floor, at the height of the mouth centre. The other two probes were placed 5 cm lower (p1) and 5 cm higher (p3) than p2 (see Figure 3, top left).

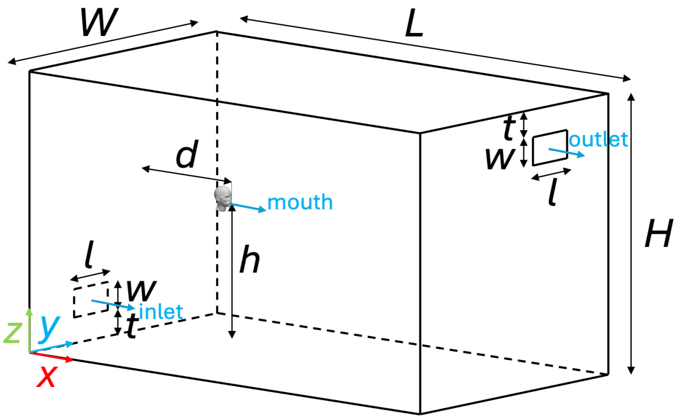

Figure 1: Test chamber sketch including all the dimensions considered within the simulation.

2.2 CFD simulation set up

To compare the three physical assumptions, we simulate the breathing jet and contaminant dispersion using three different solvers in OpenFOAM® ESI 2312 (ESI-OpenCFD, 2023). Table 1 shows the three OpenFOAM® solvers we customised to model contaminant dispersion as a passive scalar. For the inlet at the mouth of the manikin head, we average the measurements we conducted at the mouth of the manikin in the test chamber and set the velocity at the mouth inlet to 0.55 m/s. To perform the simulations, we used a mesh with 1.3 million grid cells, after performing a grid convergence analysis (Celik et al., 2008; Sorensen and Nielsen, 2003).

Table 1: Properties of the different solvers used in this study.

Solver	(In)compressible	Temperature modelled
passiveSimpleFoam (pSF)	Incompressible	No
passiveBuoyantBoussinesqSimpleFoam (pBBSF)	Incompressible	Yes
passiveBuoyantSimpleFoam (pBSF)	Compressible	Yes

3 RESULTS

3.1 Velocity magnitude contour plots for all solvers

Figure 2 introduces velocity magnitude contour plots at the centre of the SenseLab chamber. As can be seen from the plots, the characteristic flow around the room is different, especially when comparing the non-thermal solver with the solvers that include temperature effects. The extension of the breath jet also varies depending on the airflow around it, extending wider for the thermal solver that assumes the Boussinesq hypothesis compared to the compressible solver.

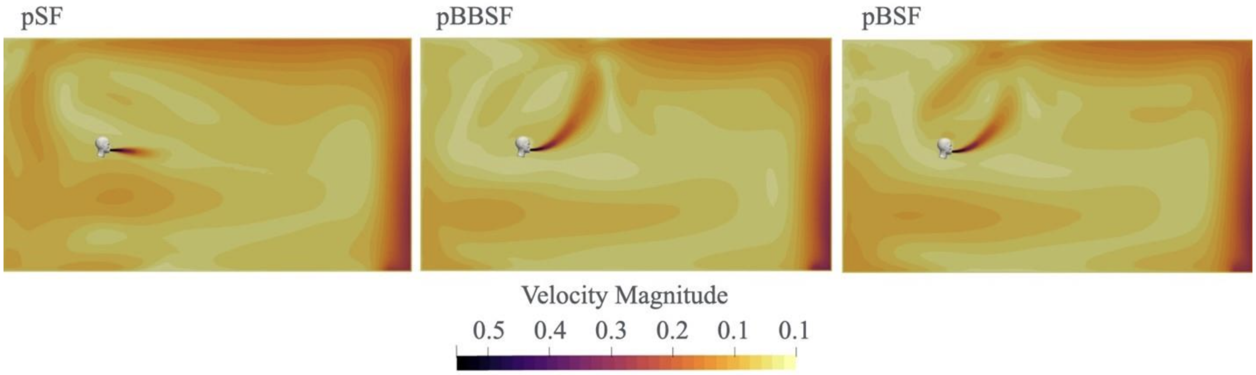

Figure 2: Air velocity magnitude contours for the three solvers considered under ventilation conditions.

3.2 Velocity magnitude comparison for pSF solver and experiments

To validate the pSF solver, we compare our results with our experimental data in the test chamber. Qualitatively, the CFD simulations reproduce the general mean flow structures observed in the experimental data: a central jet core and lateral spreading regions that decay away from the core, both laterally and longitudinally, from the mouth (See Figure 3).

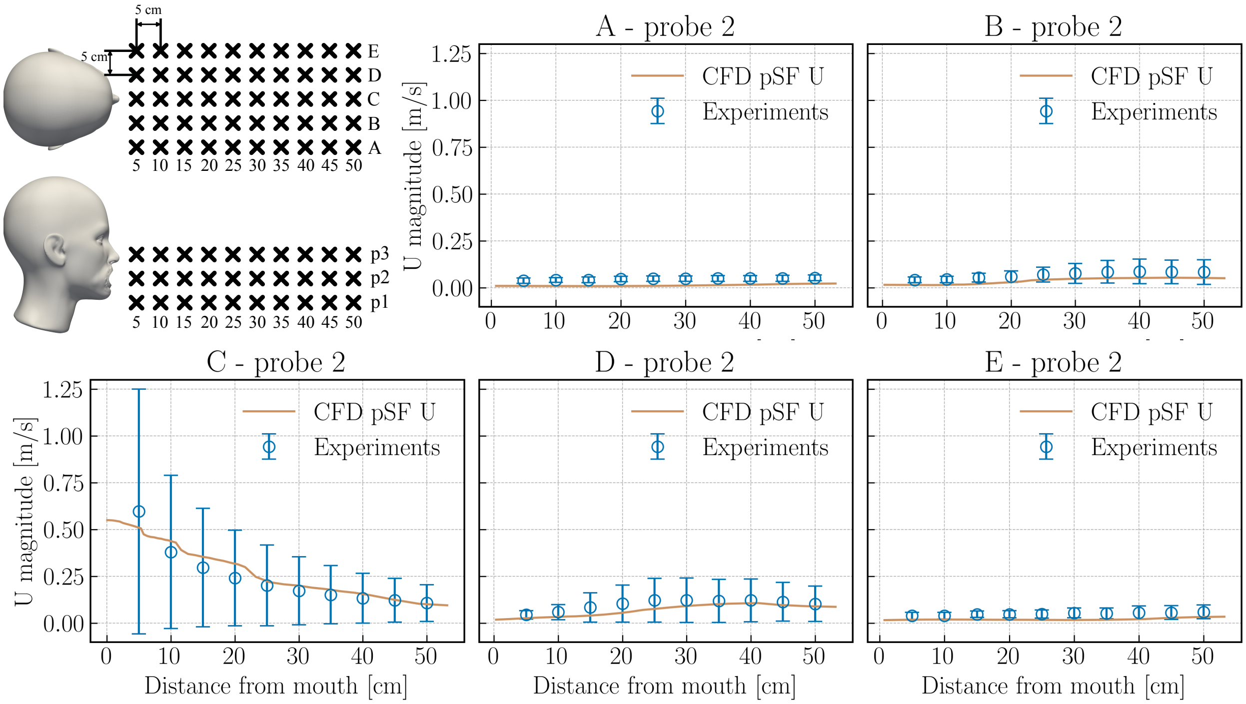

Figure 3: Air velocity magnitude near the mouth of the manikin compared between experimental measurements and CFD simulations.

However, where the air velocity magnitude drops below 0.05 m/s, there is almost no agreement between the experimental data and CFD results. The CFD underestimates the jet's span width, consistent with RANS limitations in accurately modelling the jet's wake compared to its centreline. Another issue with indoor airflow is the difficulty in accurately measuring and simulating near-zero velocities. The quantitative comparison yields an R^2 value of 0.83, indicating overall good agreement, despite underprediction in the decay regions. Consequently, the pSF solver is validated.

4 CONCLUSIONS AND FUTURE WORK

As it can be observed from the results, although the three solvers have been used interchangeably, there is a difference in the characteristic airflow patterns and the results they provide. Future work will involve validating the two thermal solvers against the closest literature references.

ACKNOWLEDGEMENTS

This work utilised the DelftBlue supercomputer's computational resources provided by the Delft High-Performance Computing Centre and the Dutch national e-infrastructure, with the support of the SURF Cooperative grant no. EINF-6125 from the Dutch National Council (NWO).

REFERENCES

R. Angelova, S. Mijorski, D. Markov, P. Stankov, R. Velichkova, and I. Simova, 2021. Numerical modelling of the draught rate in a mechanically ventilated climate chamber. In *Journal of Physics: Conference Series*, volume 1730, page 012095. IOP Publishing. doi:10.1088/1742-6596/1730/1/012095.

P. M. Bluyssen, F. van Zeist, S. Kurvers, M. Tenpierik, S. Pont, B. Wolters, L. van Hulst, and D. Meertins, 2018. The creation of SenseLab: A laboratory for testing and experiencing single and combinations of indoor environmental conditions. *Intelligent Buildings International*, 10(1):5–18. doi:10.1080/17508975.2017.1330187.

I. B. Celik, U. Ghia, P. J. Roache, and C. J. Freitas, 2008. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering - Transactions of the ASME*, 130(7).

ESI-OpenCFD, 2023. *OpenFOAM® v2312 - Open source CFD toolbox*. Release of OpenFOAM v2312, December 2023.

N. Hobeika, C. García-Sánchez, and P. M. Bluyssen, 2023. Assessing indoor air quality and ventilation to limit aerosol dispersion—Literature review. *Buildings*, 13(3):742. doi:10.3390/buildings13030742.

V. Leprince, B. Poirier, and G. Guyot, 2023. How to create a performance-based regulation on ventilation – the French Experience. In *Proceedings of the 43rd AIVC – 11th TightVent – 9th Venticool Conference: Ventilation, IEQ and Health in Sustainable Buildings*, Copenhagen, Denmark.

J. Muthusamy, S. Haq, S. Akhtar, M. A. Alzoubi, T. Shamim, and J. Alvarado, 2021. Implication of coughing dynamics on safe social distancing in an indoor environment - A numerical perspective. *Building and Environment*, 206:108280. doi:10.1016/j.buildenv.2021.108280.

D. Sorensen and P. V. Nielsen, 2003. Quality control of computational fluid dynamics in indoor environments. *Indoor Air*, 13(1).

T. van Hooff, B. Blocken, and G. Van Heijst, 2013. On the suitability of steady RANS CFD for forced mixing ventilation at transitional slot Reynolds numbers. *Indoor Air*, 23(3):236–249. doi:10.1111/ina.12010.

Z. Zhang, W. Zhang, Z. J. Zhai, and Q. Y. Chen, 2007. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2—Comparison with experimental data from literature. *HVAC&R Research*, 13(6):871–886. doi:10.1080/10789669.2007.10391460.