Validation and parameterization of turbulence models used in indoor environments for risk assessment of SARS-CoV-2

Validation and parametrization of turbulence and transport models used in computational fluid dynamic studies of aerosol transport in indoor environments

***D. Feldmann, C. Kühn, M. Avila**, Center of Applied Space Technology and Microgravity*

In Short

- Characterization and parameterization of turbulence models in indoor environments
- Validation of CFD simulation based on experimental data sets
- Modelling of Langrangian aerosol transport
- Risk assessment on the propagation of SARS-CoV-2 in indoor environments

The SARS-CoV-2 pandemic sparked the demand for indoor environmental risk assessment as research shows compelling evidence for its predominantly transmission in indoor environments[1,2]. At this computational fluid dynamics (CFD) analysis pose a commonly used tool in the research of many authors, due to its inexpensive and easy setup when compared to experimental analysis. However, transport dynamics in indoor environments are notoriously complex and therefore not straightforward to asses[3]. Regions of laminar air flow coexist next to turbulent regions. Breathing individuals and unsteady room ventilation generate high-velocity jets, but also much slower thermal plumes, resulting in fluid flows which span huge ranges of time and length scales. To capture effects caused by turbulence in such environments many authors utilize computational inexpensive turbulence models based on Reynolds-Averaged-Navier-Stokes (RANS) formulation or its unsteady variant (URANS). However, for the complex flow emerging in these environments, these models are not sufficiently validated. Hence in this project we aim to verify and parameterize a number of commonly used URANS models for this use case.
In turn to validate the simulations realized in this project we utilize an high-quality experimental data set courtesy of DLR Göttingen. This setup consists of an generic room measuring a volume of 12m^3. In its centre a manikin heated to body-temperature is seated, which is capable of realistically breathing through mechanic attached to his mouth (see figure 1). To capture the flow inside the room it is seeded with helium-filled soap-bubbles, which are natural buoyant in air, thus following the emerging flow structures. By aid of a 6-camera-system every point in the room is recorded for a time frame of 50s. From these recordings particle locations were acquired in full 3D and their trajectory is reconstructed with the Shake-The-Box algorithm[4]. Additional interpolation and smoothing steps and physical regularization via the Navier-Stokes equation result in continuous functions, describing velocity, acceleration and pressure in three-dimensional space.
CFD simulations in this project are preformed with help of the open-source numeric software library OpenFOAM (OF). With our numerical model we follow the setup of the experiment, where the mesh is of the same size as the experiment. Using the OF tool *snappyHexMesh* an STL file of the manikin is cut into the mesh, adding appropriate boundary layers in the process. The flow is governed by incompressible Navier-Stokes equations, as well as an energy equation accounting for thermal transport. Buoyancy in this natural convecting flow is accounted for by the Boussinesq approximation. As in the experiment we plan on simulating two breathing patter: first a light breathing manikin, where the volumetric flow rate at the mouth follows a sinusoidal progression with frequency of 0.25Hz and a tidal volume of 0.6l, and second a heavy breathing manikin where the frequency is 0.33Hz and the tidal volume is 1.1l. For the validation we plan on using the commonly used RANS models k-ϵ, k-ω, their blend k-ω-SST, as well as RNGk-ϵ. Additionally we also plan on using more expensive Reynolds-stress equation models to better account for anisotropies expected around exhaled jets and thermal plumes.
Afterwards a selected set of turbulence models is

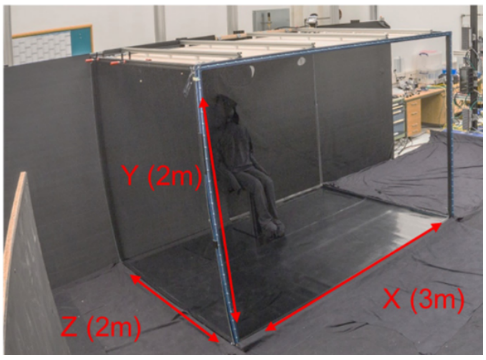

Figure 1: *Experimental setup at the DLR in Göttingen.*

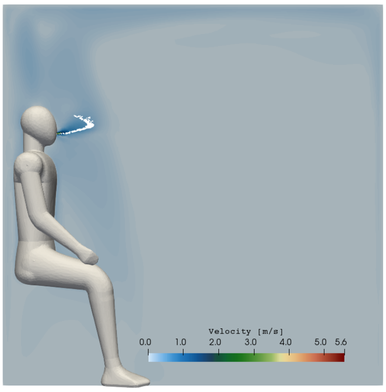

Figure 2: *Simulation showing the velocity in the centre plane, where the breathing manikin is located. White dots in the plume are Lagrangian aerosol particles.*

fine-tuned by modifying the sets of model coefficients so that the data obtained in the real world experiment is most accurately represented. This giving us the opportunity to express recommendation for future numerical analysis of indoor environments using RANS or URANS methods.
Finally using the optimized turbulence models and OF's advanced particle tracking capabilities, we perform simulations including an additional passive Lagrangian phase. In this process aerosol particle following the size distribution of 10's to 100's of micro meters[5] and are exhaled from the manikins mouth (see figure 2). Following the propagation of the particles and build-up of different concentrations this allows us perform risk assessment for the generic setup in this project.

WWW

https://zarm.uni-bremen.de

More Information

[1] Q. J. Leclerc, N. M. Fuller, L. E. Knight, S. Funk, G. M. Knight *Wellcome Open Research* **5**, p.83 (2020). doi:10.12688/wellcomeopenres.15889.2

[2] H. Qian, T. Miao, L. Liu, X. Zheng, D. Luo, Y. Li *Indoor Air* (2020). doi:10.1111/ina.12766

[3] R. Mittal, R. Ni, J.-H. Seo *Journal of Fluid Mechanics* **894** (2020). doi:10.1017/jfm.2020.330

[4] D. Schanz, S. Gesemann, A. Schröder *Exp. Fluids* **57** (2016). doi:10.1007/s00348-016-2157-1

[5] T. Dbouk, D. Drikakis *Phys. Fluids* **32** (2020). doi:10.1063/5.0011960

Project Partners

DLR Göttingen (https://www.dlr.de): D. Schanz & A. Schröder

Funding

DFG project number: AV 120/7-1