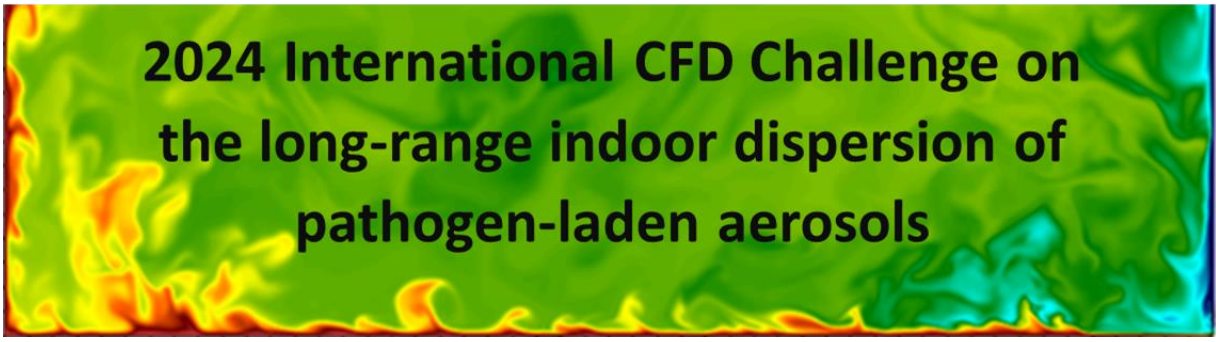
ANNOUNCEMENT



CHALLENGE PRESENTATION

The COVID-19 pandemic has evidenced the need for a better understanding of the flow physics behind the airborne transmission of infectious diseases. To analyze the indoor dispersion and transmission of pathogens via the aerosols spewed into the air by an infected person who sneezes, coughs, talks or sings, it is crucial to elucidate the dynamics of the long-range, long-term turbulent transport of the aerosol cloud. As a natural continuation of the past successful 2022 CFD Challenge on violent expiratory events (Pallares et al. 2023), focused on the short-term, short-range aerosol dispersion, we propose a new Challenge to determine the ability of the current CFD tools to predict the long-term dispersion of aerosol clouds in an idealized room with a large-scale enclosed turbulent natural convection flow. The Challenge is open to all the CFD community.

The aim of this challenge is to assess the validity of using a computationally affordable turbulence model to reproduce both the hydrodynamics and the dispersion of the aerosol cloud in an idealized indoor environment. Existing DNS of the flow at high Rayleigh number in an idealized room-sized enclosure will be used as a benchmark for this assessment.

Simulations can be performed using any of the turbulence modelling techniques available (RANS, LES, hybrid techniques…) and any numerical or discretization scheme for simulating fluid flows (finite element, finite volume, spectral methods, Lattice-Boltzmann…). The dispersion of the aerosol should be simulated using the one-way coupling hypothesis and with a Lagrangian approach, to be directly comparable with the reference DNS data.

SPECIFIC GOALS

This challenge is aimed at determining the accuracy of different combinations of numerical method and turbulence modelling to reproduce predictions of DNS of the turbulent natural convection flow and the unsteady particle dispersion in a three-dimensional room-sized cubical enclosure.

DELIVERABLES

To exchange the data files, we will use a dedicated Google Drive folder. Upon acceptance, a unique link for file sharing will be sent to each team. Accepted files must be either ASCII and vtk to facilitate post-processing in Paraview. (http://www.paraview.org). Please make sure that you can use Google Drive and export your results in any of the file formats readable with Paraview before considering participating in this challenge. The list of readable file formats can be found at https://www.paraview.org/Wiki/ParaView/Users_Guide/List_of_readers.

All submitted data must be stored in ASCII files (except for large Paraview files that can be stored in binary format if needed).

Further information and details will be provided in the instruction document that will be sent to participants upon registration by October 16, 2023.

Details about the code used for the simulation, the mesh, the spatial and temporal discretizations, the turbulence and numerical models and information about the particle tracking method will also be required.

Each team's results will be kept and presented anonymously.

IMPORTANT DATES

October 2 2023: Challenge Announcement.

October 16 2023: Challenge starts.

May 1 2024: Deadline for results submission.

June 2024: Virtual workshop. All teams are expected to participate and present and discuss their results and conclusions.

Fall 2024: Paper submission. The outcomes of the challenge will be presented in a single paper led by the organizers and co-authored by all challenge participants. The maximum number of co-authors per team will be limited to two for teams submitting, at least, a single simulation of the flow and the particle dispersion and to a maximum of three for teams submitting at least two different simulations for both, the flow and the particle dispersion, for example, with different turbulence models.

ORGANIZERS AND CONTACT

The Challenge is jointly organized by University Rovira i Virgili (Tarragona, Spain) (Jordi Pallares, Alexandre Fabregat and Salvatore Cito) and Università degli Studi di Udine (Udine, Italy) (Cristian Marchioli). Please address your questions to jordi.pallares@urv.cat

PARTICIPATION

To participate in the 2024 International CFD Challenge on the long-range indoor dispersion of pathogen-laden aerosols, please send an email with the **names**, **email addresses** and **affiliations** of the members of your team to jordi.pallares@urv.cat before October 16, 2023. After being enrolled, each team will be granted access to the Google Drive account and will receive the instructions to participate in the Challenge.

REFERENCES

Pallares J. et al. (2023). Numerical simulations of the flow and aerosol dispersion in a violent expiratory event: Outcomes of the "2022 International Computational Fluid Dynamics Challenge on violent expiratory events". Physics of Fluids, 35(4), 045106. https://doi.org/10.1063/5.0143795