Conference Agenda

15th ROOMVENT Conference

Session

NV2: Numerical ventilation models 2

Session Chair: **Marco Simonetti**

Presentations

Interacting Multiple Models for learning the air change rate and occupancy of a room from CO2 measurements

Simon Rouchier
Université Savoie Mont Blanc, France;

Accurate predictions of indoor air flow rates are a necessary condition for a reliable design of ventilative cooling solutions, but are also very uncertain. Combined heat and air flow modelling in buildings is a time-consuming process that requires the coupling of several simulation tools and is prone to user errors. Additionally, its outcome is very sensitive to non-controllable and uncertain conditions: occupant behaviour, windows opening schedule, weather… As a consequence, passive cooling designs based on direct modelling are not guaranteed to perform as well as intended and a maintenance period is recommended.

Instead of traditional modelling practices, which translate the knowledge of all physical phenomena of a system into equations, data-driven modelling refers to the use of measurements, rather than a priori knowledge, to build predictive models through statistical learning. As a first step towards data-driven air flow predictions, the present work uses CO2 concentration measurements to learn the air change rate and occupancy in a room. There are already numerous examples of statistical learning applied to estimating either the occupancy or the air change rate : the originality of the previous work is to simultaneously learn both, which are variable in time. In order to achieve this goal, Interactive Multiple Models, a type of Switching Linear Dynamic Systems, are implemented. They are a tool for inferring the parameters of a linear equation (here, the CO2 conservation equation) in which coefficients may take a finite number of values at every time step (abrupt variation of air change rate due a to window opening, occupants entering or leaving the room)…

Results show that this dual estimation is feasible, although estimates come with an uncertainty inherent to data analysis results. They are a first step towards data-driven modelling of natural and mechanical ventilation.

Fast fluid dynamics models with/without a semi-Lagrangian scheme for simulating indoor airflow

Wei Liu[1], Haowen Sun[2], Yu Xue[2]
[1]Division of Sustainable Buildings, Department of Civil and Architectural Engineering, KTH Royal Institute of Technology, Brinellvägen 23, Stockholm, 100 44, Sweden; [2]School of Civil Engineering, Dalian University of Technology (DUT), 2 Linggong Road, Dalian, 116024, China;

It is popular to use CFD (computational fluid dynamics) to predict indoor airflows and pollutants transport. However, CFD can be time consuming for modeling large-scale problems with a great amount of grid cells, such as industrial building. Then, FFD (fast fluid dynamics), which is able to accomplish efficient and accurate simulation of indoor airflow, would be a good option. FFD solves the advection term of the Navier–Stokes either by a semi-Lagrangian scheme or an implicit upwind scheme. The semi-Lagrangian scheme can be highly efficient, especially ran parallel using GPU (graphic processing unit). The 1st order semi-Lagrangian scheme adopts linear interpolation method that is simple and easy to use, but it is not conservative and would introduce significant numerical diffusions. In order to improve its accuracy, a high-order interpolation schemes such as high-order backward and forward sweep interpolating scheme not only reduces both dissipation and dispersion errors but also guarantees the convergence speed. An implicit upwind scheme is conservative and introduces minor numerical diffusion. But it may increase the computation time. Therefore, this study compares the performance of FFD with a semi-Lagrangian scheme using high-order interpolation scheme and that with implicit upwind schemes in terms of accuracy and efficiency. Both solvers were developed using the open source CFD tool OpenFOAM. The comparisons used one two-dimensional case and one three-dimensional case, with the experimental data from literature. The results showed that both schemes had similar performance in accuracy if an appropriate time step size was used. With large time step size, the semi-Lagrangian scheme using high-order interpolation scheme introduced noticeable numerical diffusion that reduced the prediction accuracy. Besides, the FFD with semi-Lagrangian scheme took more computing time than the FFD with implicit upwind schemes. This was because the identification of the departure cell in OpenFOAM took extra computing time.

Gross and integral parameters of air velocity distribution in rooms used to validate CFD calculation

Maria Hurnik[1], Nikolay Ivanov[2], Marina Zasimova[2], Zbigniew Popiolek[1]
[1]Department of Heating, Ventilation and Dust Removal Technology, Silesian University of Technology, Konarskiego 20, 44-100 Gliwice, Poland; [2]Department of Fluid Dynamics, Combustion and Heat Transfer, Peter the Great St.Petersburg Polytechnic University, 29 Polytechnicheskaya str., St.Petersburg, 195251, Russia;

The CFD calculation results are usually validated by a comparison with experimental data along some selected horizontal and vertical lines across the room or by presenting the maps of air distribution parameters at some cross-sections of the room. In this way, CFD results can be quantitatively compare at some points or regions of a room. In ventilated rooms, the jet zone and the occupied zone can be usually distinguished. The paper presents a method of gross/integral parameters determination used to validate CFD calculation in the jet zone, keeping in mind that accuracy of CFD modelling of the airflow in the jet region has significant impact on the results in the occupied zone. At various distances from the supply diffuser, the airflow field in the jet can be described using the maximum value of the mean longitudinal velocity component and the half width of the jet (see Fig. 1). By integrating the measured and computed velocity distributions, it is possible to determine the jet integral parameters: the volume flux and the momentum flux (see Fig. 2). The sidewall jet up to a certain distance from the opening can be considered as quasi-free jet and the model of free jet from point source of momentum can be used to describe the air velocity distribution. This model has three gross parameters: the velocity decay coefficient, the origin position (position of the source of momentum) and the momentum loss coefficient. The paper compares gross parameters evaluated from the computational and experimental data for a sidewall jet test and discusses possible reasons for discrepancies.

Fig.1. Jet half-width variation with the distance from the diffuser

Fig.2. Momentum flux variation with the distance from the diffuser

Fig.1. Jet half-width variation with the distance from the diffuser

Fig.2. Momentum flux variation with the distance from the diffuser

Coupled simulation study on the stack effect in shuttle elevator shafts of super high-rise buildings

Mengxiao Xie[1,2], Jian Wang[1,2], Chenyu Li[1,2], Jing Zhang[1]

[1]Tongji University; [2]Tongji Architectural Design (Group) Co.;

The strong stack effect in the shuttle elevator shafts of super high-rise buildings in winter will lead to many problems such as energy waste and safety risks. First of all, based on the theory of body heat source and surface heat source, this paper puts forward a mathematical model of air flow considering the convective heat transfer of elevator shaft wall in order to investigate the driving forces and its primary and secondary relationship. Theoretically, the formation of the thermal pressure in the shuttle elevator shafts under the condition of heating and air conditioning is affected by two kinds of heat sources. One is the body heat source of the hot air from the bottom lobby space and the other is the surface heat source of the shaft wall along the height direction. Secondly, according to the two heat source forces of the stack flow in the shuttle elevator shafts, the coupled simulation in a typical building under the heating condition in winter is conducted. The elevator shaft is the field area while the other spaces are the network areas. The coupling calculation is realized by using CONTAM. The comparison of theoretical and simulation results verifies that the body heat source is the main cause of the hot pressure, which is positively related to the temperature difference and the height of the elevator shaft. The surface heat source plays a secondary role in the cause of the hot pressure, which is negatively related to the air mass flow. Finally, through the coupling simulation results, it is found that the air flow in the shuttle elevator shaft is not only related to the pressure difference between the two sides of the elevator door, but also related to the opening mode and the relative opening areas of the elevator shaft. The opening and closing states of the upper and lower elevator doors have an important influence on the pressure difference in the shaft relative to the chamber and the air volume in the shaft. When each door of the upper and lower elevator doors is opened, the ventilation state produces the maximum series benefit and the air volume in the shaft is the maximum, which is the most unfavorable working condition for the elevator to close the door. Therefore, the coupling simulation method proposed in this paper can not only obtain the detailed parameter distribution at the shaft, but also simplify the actual building model and improve the simulation efficiency.

Controlling thermal comfort in a ventilated non isothermal enclosure using POD reduced bases

Mourad Oulghelou, Claudine Béghein, Cyrille Allery

LaSIE, UMR 7356 CNRS, University of La Rochelle, France;

This study aims at controlling thermal comfort in a two-dimensional enclosure under a mixed convection regime by modifying the air velocity and or the temperature at inlet. For that purpose an optimization problem must be solved. If a conventional optimization method is used, a large number of computations of the Navier-Stokes equations coupled with the energy conservation equation is required. This may require a large amount of computing time (several days) and computer storage.

In this work, the optimization problem was solved with a genetic algorithm. To decrease the computing time of the fitness function, reduced order models (ROM) were used. The ROM approach consists first in obtaining a reduced basis, that approximates the flow solution with a small number of vectors, by Proper Orthogonal Decomposition (POD). Projecting then the flow equations onto this basis with the Galerkin method enables to obtain a small set of ordinary differential equations, that can be solved very quickly.

However, this basis is valid only when the control parameters do not vary much. Such an approach cannot be used in an optimization loop where control parameters vary.

To circumvent this difficulty, CFD computations were first carried out for different values of the control parameters (temperature and air velocity at inlet). For each of these control parameters, a POD was then performed to provide the bases of the velocity and temperature fields. Next, for each member of the population built in the genetic algorithm, an interpolation of this reduced basis was applied for evaluating the fitness function.

This method yields good results in a quasi real time. In addition, another advantage of this method, in comparison with zonal approaches for instance, is that it provides the temperature and velocity fields for each grid node of the computational domain.