Conference Agenda

15th ROOMVENT Conference

Session

NV1: Numerical ventilation models 1

Session Chair: **Koji Sakai**

Presentations

Reproduction of the thermal plume above commercial cooking equipment using computational fluid dynamics analysis (part 1) Validation of the RANS Turbulence Model for an isothermal jet

Yuki SHIMANUKI[1], Takashi KURABUCHI[2], Hayato KIYOSUKE[2], Yoshihiro TORIUMI[3], Lee SIHWAN[4], Yasuhisa ASAWA[1]

[1]TOKYO GAS CO., LTD.; [2]Tokyo University of Science; [3]Tokyo Denki University; [4]Shinshu University;

Validation of the Reynolds-averaged Navier–Stokes turbulence model for an isothermal jet was conducted to study the reproduction of thermal plumes above commercial cooking equipment. The standard k-ε model (SKE) and realizable k-ε model (RKE) were evaluated. The calculation fields were set based on previous studies. The results show that the full width at half maximum of velocity simulated with the RKE was similar to measurement results in the self-preserving. The results also suggest that the RKE is better than the SKE in the case of a commercial kitchen. The results also suggest a valid grid size for practical studies.

Buildings indoor environment monitoring: a hybrid approach for coupling real and virtual sensors

Gabriele Ottino, Davide Manzione

DOFWARE S.r.l., Italy;

In the IoT era, sensors are the direct way for sampling physical fields inside closed environments. Nevertheless they are affected by some limitations: they are often representative of the physical field at single sampling points, not giving an overall insight; their amount increases when a detailed representation both in time and space is required, entailing high infrastructure costs due to the large memory space handling and heavy maintenance activities. In the present work, a hybrid approach is proposed for monitoring the environment conditions inside closed rooms continuously in time and space: sensors are made to collaborate with physical-numerical model simulations. The contributions of the two approaches integrate mutually. The numerical models provide comprehensive information of the air behaviour inside the room by discretizing the time and spatial domains with the desired accuracy, and then extending the insight with respect to what the sensor data offer. On the other side, the sensors provide real field data, which allows to cover gaps in the modeling approach due to lack of information about, for example, window frame leakages, wall material decay throughout the years, and external atmospheric conditions.

In the present work, the sensor data are coupled with dynamic numerical models. A co-simulation platform is implemented where numerical models are run by providing external sensor data as boundary conditions at specified time intervals: in the time interval between two consecutive sensor samplings, the numerical simulation runs autonomously; when the sensor data are provided, the model boundary conditions are updated, and the simulation restarts from the last computed field. The numerical model is built up by joining the lumped approach implemented in Modelica language with the spatial behaviour description provided by a dedicated 3D treatment: the resulting model must be faster than real-time for keeping up with sensor data. Each computing unit is encapsulated inside an interface not having to deal with their specific implementation. The co-simulation platform is then built, by making the units communicate both mutually and with the field inputs at well-defined time intervals. An illustrative example is provided dealing with the airflow and thermal monitoring of a standard office.

Exploring the Pre-Cooling and Heating Performance of an Earth-to-Air Heat Exchanger System across Seasons in Warm and Cold Japanese Regions

Yusaku Motoyama[1], Yasuyuki Shiraishi[1], Kento Tomoda[1], Kenichi Hasegawa[2]

[1]The University of Kitakyusyu, Japan; [2]The University of Akita Prefecture, Japan;

地対空熱交換器システムでの屋外空気の予冷および加熱効果は、3次元非定常計算流体力学（CFD）解析によって長期にわたってシミュレートする必要があります。ただし、流れ場の計算負荷には限界があるため、従来のコンピューターでは非定常CFD解析の年間計算は困難です。次に、計算負荷を軽減するために流れ場を解析しない非定常CFD解析によるEAHEシステムの予冷および暖房性能の年間予測方法を提案および検証します。この方法は、事前にシミュレートされた流れ場を使用し、これらの流れ場を読み込むことで熱方程式を線形化することにより、CFD解析の計算負荷を軽減します。提案手法では、以下の効果が期待されます。1）事前分析による計算負荷の軽減。2）熱方程式の収束の改善。3）時間間隔の拡大。また、これまでのすべての研究において、検証は日本の九州（温暖地域）に限定されており、EAHEシステムの予冷および加熱効果に対する気候分類の違いの影響は検証されていません。本研究では、流れ場の計算負荷を低減した非定常CFD解析によりEAHEシステムの年間性能を予測する方法を提案し、温暖地域と寒冷地域の性能の違いと性能を示します。分析の結果から、冷房および暖房期間中の屋外温度と比較して、出口に向かう温度の偏差は小さくなりました。また、冷却期間中に温度が冷却側に、加熱期間中に加熱側に変化することも確認された。このことから、このモデルは土壌熱容量を用いてEAHEシステムの特性を再現できることが確認されました。夏の暖かい地域では、予冷の効果は比較的大きかったが、EAHEシステムの相対湿度は増加し、寒い地域よりも結露がよりスムーズに発生した。ただし、暖かい地域と寒い地域では、相対湿度が長時間80%を超えたため、両方の地域で結露が頻繁に発生したと考えられます。予冷の効果は比較的大きかったが、EAHEシステムの相対湿度が増加し、結露は寒冷地よりも流occurredに発生した。ただし、暖かい地域と寒い地域では、相対湿度が長時間80%を超えたため、両方の地域で結露が頻繁に発生したと考えられます。予冷の効果は比較的大きかったが、EAHEシステムの相対湿度が増加し、結露は寒冷地よりも流occurredに発生した。ただし、暖かい地域と寒い地域では、相対湿度が長時間80%を超えたため、両方の地域で結露が頻繁に発生したと考えられます。

一方、寒冷地の予熱量は暖地の予熱量よりも多いことが確認された。以上の結果から、導入された外気およびEAHEシステムの温度および湿度特性は、屋外の気候に応じて変化し、夏季は暖かく、冬は予冷および暖房効果が大きいことが示唆されました。寒い地域で。

Convective heat transfer coefficients in cavities: a review of correlations for modelling double skin facades

Elena Catto Lucchino[1], Olena Kalyanova Larsen[2], Francesco Goia[1]

[1]Department of Architecture and Technology, Faculty of Architecture and Design, Norwegian University of Science and Technology, NTNU, Norway; [2]Department of Civil Engineering, Aalborg University, AAU, Denmark;

Convection is an essential mechanism for heat transfer in double skin façades, which are building envelope systems characterised by a (naturally or mechanically) ventilated cavity. The study of this mechanism in the built environment is already challenging when convection is analysed at the room level, where each surface can be analysed independently. The application of the same approach to the study of the convective heat transfer in the double-skin façade cavity, however, is questionable given the potentially different range of dimensionless numbers characteristic for DSF performance. These differences are often due to dimensional aspects (cavities of small enclosed space), asymmetrical boundary conditions, a varying range of velocities, etc., which must be taken into account when choosing the proper correlation for the surfaces facing the ventilated gap of a double skin façade, i.e. glazing and shading system.

This paper gathers and analyses the correlations available in the literature, with a particular focus on studies related to the asymmetrically-heated channels, for different typologies and flow regimes. The set of correlations presented can be applied to the airflow under different regimes (natural convection, mixed ventilation and forced convection), different cavity dimensions (narrow or wide cavity), and that takes into account the presence (or the absence) of the shading device inside of the cavity.

A benchmark for room air distribution: The backward facing step flow

Peter Vilhelm Nielsen, Chen Zhang, Laura Annabelle Bugenings, Markus Schaffer

Aalborg University, Denmark;

A CFD workshop on a geometry that could be typical for deep buildings and tunnels took place in 2015 on the ISHVAC-COBEE 2015 conference. The used model is known as the backward facing step flow. 19 professional teams participated in the workshop and covered laminar flows, transitional flows, and fully developed turbulent flows. Members of the workshop made CFD based predictions of the air distribution without having measurements for comparison. These predictions were significantly diverse among different teams but were the results of the different turbulence models used, the inability to handle transitional flows at a sufficient level and of different software elements as discussed in the conference [1].

A benchmark test for the backward facing step flow has been performed in a small-scale model and the airflow pattern was measured by a Laser Doppler Anemometry (LDA). This study will address the possibilities to validate CFD models by the benchmark test results and will provide recommendations on selecting sub-models such as turbulence models or wall functions for CFD simulations for both 2D and 3D geometries.

As the results are now validated by known measurements it is possible to optimize these predictions and the influence of turbulence models, wall functions, and other important elements of the software are addressed in the discussion. This paper identifies the optimal software elements for the benchmark.

CFD MODELING AND SIMULATION OF SOLAR SHADING EFFECT ON EXTERNAL LOUVER

Akira Okamura, Sihwan Lee

Shinshu Univercity, Japan;

An external louver has become widely used to decrease the solar load in building by shading the sunlight. However, it is difficult to design effectively because there are various types with color, material, shape, and reflectivity, and the solar shading effect fluctuates depending on the time of day and the angle of solar radiation. The purpose of this study is to examine the solar shading effect of external louver quantitatively by using computational fluid dynamics (CFD). To achieve the purpose, a simple box model with a window on face for actual measurement, and the internal temperature and the window surface temperature of the test model was measured. Then, the test model was generated by CFD model and numerical analysis was performed, and it was confirmed that the calculation results agreed with actual finding. A vertical louver and horizontal louver were added to the CFD model, and the internal temperature and window surface temperature were compared with that without the louver. As the results, the solar shading effect according to the shape of the louver was obtained.

Investigation of combined ventilation systems through co-simulation

Markus Wirnsberger[1], Dominik Aimer[1], Harald Krause[1], Sahra Weiner[2], Arnav Pathak[2], Gunnar Grün[2]

[1]Technical University of Applied Science Rosenheim, Germany; [2]Fraunhofer IBP, Deutschland;

The housing construction sector is under enormous pressure to reduce costs, leading to a great need for effective solutions that meet all requirements for comfort, air quality, energy efficiency, acoustics and moisture protection.

The amendment to the German standard for the design of ventilation systems for apartments (DIN 1946-6 „Lüftung von Wohnungen") will in future not only describe the design of natural or fan-assisted systems, but also the design of combined (/ hybrid?) ventilation systems. Different fan-supported systems can be combined with each other or with natural ventilation. These combined system allow cost-effective user-independent moisture protection and hygienic air exchange in multi-story residential buildings. However, the combinations of various fan-assisted systems in particular have not yet been sufficiently investigated, which is why there are no recognized design guidelines. As a result, significant losses can occur in air quality (poor mixing, low volume flows), thermal comfort (draughts, noise pollution) or energy efficiency (imbalance in heat recovery). Consequently, planners will only use these systems to a limited extent, since both experience and suitable dimensioning specifications are lacking.

Therefore dimensioning and configuration solutions are required, which will be researched within the framework of the study presented here. For this purpose, differently combined ventilation solutions for residential construction are investigated using measurement technology and dynamic simulation. The metrological investigation in a real environment provides practical analysis values for the evaluation of the system solutions and for the validation of the assigned simulation. The simulation based on using the VEPZO model (VElocity Propagating ZOnal model, Norrefeldt 2013) offers the possibility to investigate the possible combinations of system components in different variations and usage scenarios. The results can directly be applied in residential construction with secured performance characteristics as well as statements on thermal comfort and air quality. This work summarizes the

results of the model creation and co-validation process. Co-validation means that the physical measurements conducted are tested for physical soundness using the physical model, while the physical model is validated against measurement data. This work summarizes the results of the model creation, simulative scenarios of combined ventilation systems and the associated co-validation process.

A Machine Learning-based Non-Intrusive ROM Approach for Indoor Environment Applications

Andrea Lario, Gabriele Ottino

Dofware S.r.l., Italy;

Recently Computational Fluid Dynamics (CFD) is being used increasingly in HVAC design due to its accuracy in predicting flow velocity and thermal fields inside closed environments. Once geometry, boundary and initial conditions are specified, designers are able to produce virtual snapshots containing all the quantities relevant to the evaluation of the internal air quality and to test solutions. The main drawback of CFD consists into its high computational cost which is limiting its application. Moreover energy saving and sustainability requirements are becoming more strict and, in order to match these requirements, goal-oriented design, sensitivity analysis, and optimal control are being heavily involved both in building design and management.

In view of these considerations, a high number of expensive, parameter-dependent simulations is needed for gaining reliable information. New techniques are then fundamental which are able to provide still accurate solutions at a reduced computational cost. Recently new approaches have been proposed for speeding up fluid dynamic simulations: among others, the fast fluid dynamics (FFD) and reduced order models (ROM), the former applied mainly to enclosed environments analysis, the latter to a wider range of cases. The results are encouraging, as faster-than-real-time solvers have been developed and tested with a modest reduction in accuracy. Driven by this trend, in the present paper a Non-Intrusive ROM (NIROM) is proposed. It joins the features of two techniques: the Singular Value Decomposition (SVD) on one side, which is employed for extracting the most energetic modes from a CFD set of simulations, thus reducing its dimensionality. On the other side neural-networks are employed in order to build a relation, through the computation of proper weights, between modes, parameterized boundary conditions and the corresponding flow field solutions. The result is a tool able to run faster than real time even on large meshes, whose accuracy is tested against literature benchmarks. The main load is ascribed to the CFD simulation database generation and processing: once the modes have been computed and the neural network trained, it is possible to explore several configurations in a short time with limited computational resources. A typical application of this tool could be in support to an HVAC control system: the controller could analyse several settings by relying on a case-tailored tool able to evaluate the behavior of the internal air quality when subjected to different operating conditions. Other usages such as dealing with indoor environment layout design are made straightforward by simply changing the database features.

Indoor airflow simulation via Deep Neural Network on the basis of CFD database

Qi Zhou, Ryozo Ooka

The University of Tokyo, Japan;

There is an urgent need for fast and accurate prediction of indoor environments with regard to inverse design, ventilation online control, coupling simulation, etc. Deep neural networks (DNNs) have been confirmed to have advanced modelling abilities, high-accuracy prediction capabilities, and high-speed computation powers. To confirm the feasibility of DNN for indoor airflow prediction, in this study, a DNN is constructed, in which the input are the boundary conditions of a two-dimensional room, and indoor velocity (vector) and temperature distributions of this room are simulated simultaneously. Computational fluid dynamics (CFD) simulations are conducted to create a database for training, validating and testing the DNN. Two data preprocessing methods, i.e., nondimensionalization and normalization, are considered, and the results are compared to those with no preprocessing. The results show that predictions via DNN with nondimensionalization or normalization preprocessing are consistent with CFD simulations on both the training dataset and testing dataset. The mean errors of the velocity and temperature predictions on the testing dataset are less than 10% and 6%, respectively, which are much lower than those of DNN predictions with no preprocessing. The data preprocessing is thus suggested for both input and output data for DNN prediction of indoor airflow. With regard to calculation time, the CFD simulation requires at least 1000 s for each case, whereas the DNN prediction requires no more than 0.1 s. DNN has great potential for practical applications of fast and accurate indoor environment prediction.

Numerical investigation of the ventilation effectiveness for different exhaust positions and contaminated locations for isothermal mixing ventilation

Gerrid Brockmann, Benjamin Zielke, Martin Kriegel

Technische Universität Berlin, Germany;

In the literature the exhaust openings for common mixing ventilation rarely attract any attention. It is known that: „air movement in the vicinity of an exhaust opening shows the characteristic of a potential flow … Exhaust openings have therefore, no great distance effect.“ Further the REHVA Mixing Ventilation Guidebook says: „However, the arrangement of the exhaust diffuser has impact on the ventilation efficiency“. But there are no numbers given to quantify the impact or how to determine the optimum position for an exhaust opening.

The present numerical investigation demonstrates the need for future guidelines. It starts by solving the optimum position of the exhaust air depending on the position of the contamination source for two different isothermal ventilation cases (slot and swirl diffuser). The room to be examined has a ground floor area of 4 by 5 meters and a height of 3 meters and contains no obstacles. Different exhaust air positions are chosen on the walls and the ceiling. Different positions of contamination sources in the room are considered by using different passive scalars. Since the contamination source positions are assumed to be known, the evaluation is based on the contamination removal effectiveness. In addition, the flow is visualized in order to better understand the relationship between the exhaust air position and the contamination source position.