Conference Agenda

15th ROOMVENT Conference

Session

IEQ1: Indoor Environment Quality (IEQ) 1

Session Chair: **Kwok Wai Tham**

Presentations

Thermal environment in room with ventilation design based on air quality and different cooling methods

Weixin Zhao[2], Mariya Bivolarova[1], Benjamin Jensen[1], Detelin Markov[3], Arsen Melikov[1]

[1]Technical University of Denmark, Denmark; [2]Aalto University, Finland; [3]Technical University of Sofia, Bulgaria;

Requirements for room thermal environment and air quality are specified in the present standards. Indoor air quality is expressed as the required level of ventilation or CO_2 concentrations. Mass balanced method and method based on person and building component are used for determination of the supplied airflow rate (CEN CR 1752, DS/EN15251). Room thermal environment can be achieved by use of combined (convective and radiant) or only convective systems. In this study we compared the room thermal environment obtained with the two methods for determination of the ventilation rate and two systems, combined system and convective system alone. Physical measurements were performed in a simulated meeting room with eight occupants: seven persons and one thermal manikin. Heated window was used to imitate solar load. The ventilation air was supplied from ceiling slot diffusers. Based on the mass balanced method CO_2 concentration of 1000 ppm at the exhaust was obtained with supply flow rate of 57.4 L/s. CO2 concentration of 1000 ppm was also obtained when the supply flow rate of 68.1 L/s was calculated based on the method of person and building component (category B of DS/EN15251). The supply air temperature and the chilled ceiling temperature were adjusted to maintain room temperature of 26 °C. Room temperature of 26 °C was also maintained at air supply flow rate of 142 L/s without use of the chilled ceiling. The results reveal that most uniform thermal environment in the occupied zone was achieved at 68.1 L/s and best mixing (i.e. best air quality) was achieved at 142 L/s. The draught discomfort was lowest at 54.7 L/s. The thermal manikin measurements revealed better overall thermal comfort at 68.1 L/s. These results, together with consideration on cost and complexity of the used system as well as energy use, may help ventilation design.

Large Eddy Simulation of the airflow generated by an idealized human footstep

Amir BOULBAIR[1,2], Ahmed BENABED[3], Bart JANSSENS[2], Karim LIMAM[1], Walter BOSSCHAERTS[2]

[1]Laboratoire des Sciences de l'Ingénieur pour l'Environnement (LaSIE) - UMR CNRS 7356, France; [2]Royal Military Academy, Rue Hobbema 8, Brussels, Belgium; [3]Departement of Mechanical and Environmental Engineering, ESTACA, Paris-Saclay Campus, 78180, Montigny-le-Bretonneux, France;

One of the main sources of pollution in indoor environments is the human walking-induced particle resuspension. The main cause of this phenomenon is the aerodynamic disturbances generated in the vicinity of the foot during the footstep. This generated airflow can cause detachment of deposited particles and their transport in the indoor environment. In this work, a three-dimensional numerical model of a moving idealized foot was generated using the ANSYS-CFX CFD package. The idealized foot was a rectangular plate, which rotates downward and upward with a constant velocity. A Large Eddy Simulation (LES) technique was used for simulating the unsteady airflow field around and under the idealized foot. The Immersed Solid Method (ISM) was used to incorporate the idealized foot. Results were compared with laser doppler anemometry (LDV) measurements and good agreement was found.

Study on installation criteria of ventilation opening in common corridor of apartment houses where gas water heaters are installed

Kaoru Nishizawa

Tokyo University of Science, Japan;

Japan has a bathing culture and home water heaters are popular. If a gas water heater is installed in the shared corridor of an apartment house, the exhaust gas may stagnate and flow into the room through the vent. For this reason, the standard of the installation position of equipment is set, but this limits the construction plan. In this study, we investigated the possibility of relaxing the conditions that can be installed, such as ventilation openings, through full-scale model experiments and CFD analysis. We also propose and verify a simple evaluation method using CFD steady state analysis, as well as experiments with varying output.

Ventilation openings are installed on the ceiling and wall surfaces in the corridor model, and the concentration of CO_2 inflow air into the room sucked in the duct is measured. The measurement conditions were based on the hot water supply mode method simulating the peak hot water usage time considering actual usage conditions, and the average CO_2 concentration for 1 hour was measured.

Also, with this test method, gas consumption fluctuates over time, so there is no established method to predict 60-minute CO_2 concentration using CFD steady state analysis. To solve this problem, the ultimate goal of this study is to establish effective and versatile prediction formulas for many test methods and model shapes. In addition, we will examine cases where experiments have not been conducted using CFD analysis and examine methods for improvement by changing the form of the adapter in cases where exhaust stagnation has been observed.

Establish an equation and perform a simple evaluation of experimental concentration C. In the established equation, the output with a small time ratio was approximated using different operation modes, and the change of the time ratio of each output was examined. For analysis, two types of codes, C and D, were examined. Code D was more reproducible at each output. Therefore, a case study was performed using code D. The case study examined changes in exhaust retention due to changes in screen width, but it was shown that the changes do not have a significant effect on exhaust retention. In addition, the change in the adapter form was examined after confirming that the experimental values and the analysis results generally corresponded. When the roof was attached to the adapter, the CO_2 concentration at the ventilation port was worse than the original form but improved when the outlet area was narrowed. Improvement was seen when the exhaust angle was 15 ° downward, but it worsened at 30 °. The latter is thought

to be caused by exhaust hitting the waist wall. In addition, when the corridor width is changed, different angles are considered to be optimal, so it is expected that there is a correlation between the corridor width and the exhaust angle.

As an ongoing work, the problem is that the predicted average concentration of the wall vents is lower than the experimental value, and studies are being conducted to establish parameters to solve it.

Particle resuspension & elevation from hard indoor surfaces due to vibrational and aerodynamic disturbances

Douaa Khoder Al Assaad[1], Kamel Ghali[1], Nesreene Ghaddar[1], Carine Habchi[2]

[1]American University of Beirut, Lebanon (Lebanese Republic); [2]Lebanese university, Roumieh, Lebanon;

Resuspension of particulate matter from hard indoor spaces causes adverse health effects as it re-releases settled particles on surfaces into the bulk air. Resuspension in indoor environments is usually caused by human activities generating a combination of aerodynamic and mechanical vibration disturbances. The aim of this investigation is to develop a mathematical model that can predict particle emission rates from hard indoor surfaces due to the concurrent action of aerodynamics and vibration removal forces. The developed model computed the decrease in adhesive forces due to vibrations as well as the lift and drag forces and moments due to the fluid flow. To calculate flow removal forces, the mathematical model was coupled with computational fluid dynamics (CFD) model, which can solve for the surface shear stresses. The model deals with particles embedded in the viscous sublayer. Thus, the rolling resuspension condition was defined to predict the resuspension rates. The resuspension model was validated by published literature resuspension data and good agreement was obtained.

Results showed that for an exposure of 300 seconds, particle resuspension exhibited a parabolic profile with a significant jump during the first 25 seconds of the disturbance before decreasing to background concentration values. Resuspension rates also increased when the surface was also vibrated along with the flow disturbance. For different indoor surfaces, it was found that resuspension fractions were the smallest for glass, followed by rubber, ceramic and wood. This was due to increasing roughness. When vibrations were applied to the surfaces in addition to the fluid flow, resuspension was enhanced for glass, rubber, marble and wood at different rates depending on their stiffness. It was also found that decreasing surface roughness reduced the effects of vibrations since it increases adhesive forces. It also increases surface stiffness and the rate of reduction of adhesive forces due to vibrations.

An analysis of infiltration characteristics of outdoor particulate matter in school buildings

Sowoo Park[1], Junemo Koo[2], Doosam Song[3]

[1]Dept. of Civil and Environmental System Eng., Graduate School Sungkyunkwan University, Suwon 16419, Republic of Korea; [2]Dept. of Mechanical Eng., Kyung Hee University, Yongin 446-701, South Korea; [3]School of Civil, Architectural Eng., and Landscape Architecture, Sungkyunkwan University, Suwon, 16419, Republic of Korea;

According to epidemiological studies, Particulate Matter (PM) is associated with many respiratory and cardiovascular diseases. Also, people typically spend more than 90% of their time indoors, so, exposure to ambient PM occurs mainly indoors. Even if the building is enclosed, outside air infiltrate into the room via various paths, and at this time, a high concentration of outdoor PM can be introduced together with the air. When PM penetrates through the gap, the penetration amount depends on the characteristics of the gap, which is affected by the particle size and the properties of the building. School buildings are one of the most vulnerable to high density PM. School buildings have low airtightness performance compared to typical residential buildings. In addition, classrooms are closed during class, and most rooms do not have a ventilation system or air purification system. Therefore, the PM can stay indoors for a long time, and it is difficult to be taken out to the outside. As a result, classroom PM concentrations may be higher than the recommended PM standard.

The purpose of this study is to calculate the penetration coefficient and deposition rate of PM in various buildings by measuring the change of indoor PM concentration according to changing outdoor environmental conditions. The indoor and outdoor PM number concentrations was measured using optical particle counter. Real-time air change rates were calculated using the carbon dioxide emitted by the students. Indoor and outdoor temperature, external wind speed, and wind direction were also measured to determine the influence of external environmental conditions.

As a result, the indoor PM concentration strongly followed the change of the external PM concentration. As the airtight performance was lowered, the intrusion coefficient of fine dust tended to increase. This is because the building with low airtightness was relatively easy to infiltrate PM due to the large gap between doors and windows. The smaller the particle size, the more the penetration coefficient tended to increase, because the larger the particle size, the more invasive particles were infiltrated into the building.

A methodological framework for socio-economic impacts assessment of ICT solutions to improve IEQ, health and well-being

Giulia Vergerio[1], Cristina Becchio[1], Marta Carla Bottero[2], Stefano Paolo Corgnati[1]

[1]Department of Energy (DENERG) - Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129, Torino (Italy); [2]Interuniversity Department of Regional and Urban Studies and Planning (DIST) - Politecnico di Torino, Viale Mattioli 39, 10125, Torino (Italy);

MOBISTYLE is a Horizon2020 European project that views the application at different demo cases level of some personalized ICT solutions (Mobile App, Game and Dashboard) to drive behavioural changes in buildings occupants, leveraging on the three issues of energy, indoor environmental quality (IEQ) and health. The MOBISTYLE approach is based on the provision of personalized feedback and on the deployment of a tailored awareness campaign to increase people perception about how their behaviour can influence energy consumptions and IEQ in buildings, but also about the impacts it has on their health and well-being. To support the project, specific standing monitoring systems were installed in 5 different demo cases across Europe (Italy, Slovenia, Poland, Denmark and Netherlands).

Within the project two needs have been identified: i) assessing the effectiveness of the deployment of MOBISTYLE solutions from both a private (owner/occupant) and a macro-economic (whole society) point of view; ii) making the potential benefits tangible and understandable for people to push a behavioral change. For both purposes, the Cost-Benefit Analysis (CBA) application turns to be useful.

Indeed, to assess the convenience of MOBISTYLE solutions deployment at the demo cases level, a net financial cash-flow, counting for costs and energy savings, could be performed. However, a financial analysis is not exhaustive of the possible benefits that the application of MOBISTYLE approach could generate, especially in such a project where a specific relevance is given to the health issue as a leverage to motivate a medium- and

long-term change in people habits in the way they are using buildings. Moreover, the impacts monetization required by the CBA methodology turns to be convenient in terms of communication of the potential benefits, since the translation in money makes the impacts more understandable and tangible.

Therefore, the aim of this paper is to present a methodology according to which a well- established economic evaluation tool, namely the Cost-Benefit Analysis (CBA), have been adjusted to the MOBISTYLE project to assess its socio-economic performance at the level of the single pilots. In face of the innovation brought by the project in adopting an interdisciplinary approach, building a bridge between energy and IEQ expertise and disciplines closer to medicine and anthropology, a special focus is given to the definition, quantification and monetization of the impacts that the project can have in terms of well-being and health, which are assessed in one of the pilot thanks to the data gathered from monitoring systems and to models borrowed from literature.

The whole picture of the methodological framework, as well as some preliminary results on one demo case, are presented and discussed within this paper.

CFD analysis on the indoor pollutant transmission with diffuse ceiling ventilation under cooling and heating operation

Pei Peng[1,2], **Chen Zhang**[2], **Michal Zbigniew Pomianowski**[2], **Guangcai Gong**[1]

[1]Hunan University; [2]Aalborg University;

Diffuse ceiling ventilation (DCV) has drawn much attention in many years due to its high energy efficiency and good thermal comfort, and now it is finding its way to be more broadly used. The ventilation effectiveness of DCV in term of air quality is not well documented in the literature. A numerical model is developed and validated to investigate the pollutant transmission in an office room with the DCV system. The ventilation performances of DCV are simulated under cooling and heating operation, and the comparisons are made by changing different air change rates, outlet positions and pollutant sources (e.g. from human with heat load or point source without heat load).

The results show that the airflow in the room with DCV is more complicated in heating conditions than in cooling conditions. The warm air supplied from the diffuse ceiling is difficult to enter the occupied zone due to the low momentum and low density, and the uprising thermal plumes from heat sources can create a reverse flow to the plenum. The comparisons of different pollutant sources indicate that the N_2O is hard to dispread in the space when the pollutant source is from the point source without heat load, thus the indoor average N_2O concentration is lower. Moreover, the outlet position has an effect on indoor airflow pattern and N_2O distribution in the space, and the lower outlet position is better for cooling conditions, while the upper outlet position is better for heating conditions. The CRE of cooling conditions is around 1, which implicates a good mixing of air in the room with the DCV system. While in heating conditions, the value of CRE will decrease due to the weakened indoor natural convection, and the values are around 0.78-0.95.

HUMAN WALKING INDUCED PM10, PM2.5, PM1, AND PM0.1 RESUSPENSION

Ahmed Benabed[1], **Amir Boulbair**[2,3]

[1]ESTACA'Lab, Departement of Mechanical and Environmental Engineering, France; [2]University of La Rochelle, France; [3]Royal Military Academy, Belgium;

This work investigated the human walking-induced particle resuspension in a full-scale 2.5×2.5×2.5 m^3 wooden chamber. The mass concentrations of PM10, PM2.5, PM1, and PM0.1 were monitored using a MiniWRAS Grimm counter. The floor was covered with a tufted synthetic carpet. Relative humidity RH was controlled and maintained at 40% during all experiments. The floor was seeded with neutralized alumina dust using an ejector nozzle. Using the mass-based balance equation and the well-mixed condition, resuspension rate coefficients were estimated. Results of the present work show that human walking significantly increases the indoor PM10, PM2.5, PM1, and PM0.1 concentrations. The average estimated PM10, PM2.5, PM1 and PM0.1 resuspension rate coefficients were $(2.5 \pm 0.6) \times 10^{-1}$, $(1.9 \pm 0.5) \times 10^{-2}$, $(6.5 \pm 0.3) \times 10^{-3}$ and $(4.3 \pm 0.3) \times 10^{-3}$ respectively.

Experimental study of polydisperse particle deposition in a small-scale test chamber

Ahmed Benabed[1], **Amir Boulbair**[2,3], **Marc Abadie**[2]

[1]ESTACA'Lab, Departement of Mechanical and Environmental Engineering, France; [2]University of La Rochelle, France; [3]Royal Military Academy, Belgium;

The aim of this work was to study polydisperse particle deposition in an enclosure. Experiments were carried out inside a 50×50×50 cm chamber with indoor walls covered with aluminum. Two polydisperse powders were tested: alumina (3.9 g/cm^3) and orgasol (1.5 g/cm^3). Particle concentration monitoring was carried out using a Grimm MiniWRAS counter. This later combines two measuring instruments: an aerosol spectrometer for particles larger than 0.253 µm and the so-called Nano-sizer for particles smaller than 0.253 µm. Deposition rate coefficients were estimated for particles with different sizes using the particle number balance equation and the hypothesis of well-mixed conditions. The estimated deposition rates for particles smaller than 0.07 µm and larger than 1.5 µm are in good agreement with the model of Lai and Nazaroff (2000). For particles larger than 1.5 µm, deposition rate coefficients of alumina aerosol were higher than those of orgasol aerosol. This result confirms the importance of considering the density of particles constituting the aerosol in the theoretical prediction of the deposition rate coefficients.

Study on Thermal Environment Evaluation Using the Ground Source Heat Utilization System in Ordinary Classroom

Rei Murata, **Ryoichi Kuwahara**

Yamaguchi University, Japan;

CO2濃度の上昇に伴い地球温暖化が進んでおり、世界の平均気温は上昇しています。全国平均気温の上昇、PM2.5などの大気環境の悪化、熱射病対策から学生の健康面を守る必要があります。日本では、全国の小中学校が17万教室の空調設備の設置を完了することを決定しました。エアコン

を導入することで、子どもたちの健康管理を改善し、快適な学習環境を確保することができます。しかし、空調設備は大量の電力を必要とし、ランニングコストの上昇や冷却時のエネルギー使用が懸念されています。そのため、エネルギーを削減する方法の1つとして、再生可能エネルギーの使用を検討しました。この研究は、地中熱利用システムにも焦点を当てました。地中熱利用システムは、地下約6mの深さに埋められた地下集熱管の地下温度を利用しています。夏には、気温は一年中影響を受けません。夏は外気を冷やし、冬は外気を暖めます。その後、建物に供給されます。これにより、消費電力を抑えることができます。また、地中熱利用システムは、床吹き空調システムと組み合わせて使用することができます。床吹き空調は、住宅地の空調を効率的に行い、省エネに貢献します。本研究の目的は、CFDシミュレーションにより、地中熱利用システムを用いた床吹き空調システムを用いた教室の室内環境とエネルギー消費効果を評価することである。本研究では、地中熱利用システムと床吹きを備えた教室をモデル化し、入力条件を変えて室内熱環境と気流環境シミュレーションを行った。まず、測定値に基づいて対象教室を再現します。シミュレーション結果の室温は27.0度で、27.5度の測定結果とほぼ同じでした。この結果から、シミュレーションの妥当性が確認されました。第二に、地中熱の影響を利用した場合の室内熱環境と地中熱の影響を無視した場合の室内熱環境を比較し、地中熱利用システムの熱環境影響を検討する。その結果、地中熱利用システムを利用することで、室内を快適な温度に保つことができることが確認されました。第三に、地中熱利用システムを使用した場合のランニングコストと従来の天井パッケージ空調システムを使用した場合のランニングコストを比較し、地中熱ヒート利用システムの省エネ効果を検討します。その結果、地中熱利用システムを採用することで、エネルギー消費量を30%以上削減できることが確認されました。