ROOMVENT 2011

The 12th International Conference on Air Distribution in Rooms

Trondheim, Norway

June 19-22, 2011

Book of Abstracts

NTNU Norwegian University of Science and Technology
SINTEF
NORSK VVS ENERGI- OG MILJØTEKNISK FORENING
Norwegian Assosiation of HVAC Engineers – NORVAC

Contents

Contents

Program, Overview of sessions

Technical and poster session program			
Monday, June 20, 09:00 to 10:30			
Opening session Keynote speeker: Peter V. Nielsen Keynote speeker: Qingyan Chen			
Monday, June 20, 11:00 to 12:30			
Technical session 1	*Technical session 2*	*Technical session 3*	*Technical session 4*
Case Studies	*Demand controlled ventilation*	*Evaluation, control or measurements of indoor air quality*	*Aircraft cabins* *Evaluation, control or measurements of indoor thermal environment* *Modeling and visualization*
Room R1	*Room R2*	*Room R3*	*Room R10*
Monday, June 20, 13:30 to 15:00			
Technical session 5	*Technical session 6*	*Technical session 7*	*Work shop session 1*
Modeling and visualization	*Natural, hybrid and mechanical ventilation*	*Ventilation for low energy, passive houses/zero emissions buildings*	*Ventilation of heavy industry*
Room R1	*Room R2*	*Room R3*	*Room R10*
Monday, June 20, 15:30 to 17:00			
Work shop session 2 *Ventilation in Zero Emission Buildings*	*Technical session 8*	*Technical session 9*	*Technical session 10*
	Evaluation, control or measurements of indoor air quality	*Evaluation, control or measurements of indoor thermal environment*	*New technologies for heat recovery system*
	Ventilation strategies for large rooms in historic build.	*New technologies for heating and cooling or ventilation/AC*	*Codes and regulations*
Room R1	*Room R2*	*Room R3*	*Room R10*

Program

Tuesday, June 21, 09:00 to 10:30			
Technical session 11	Technical session 12	Technical session 13	Technical session 14
Case Studies	Evaluation, control or measurements of indoor air quality	New technologies for heat recovery system Modeling and visualization	Ventilation strategies for large rooms in historic build. Modeling and visualization
Room R1	Room R2	Room R3	Room R10

Tuesday, June 21, 11:00 to 12:30			
Technical session 15	Technical session 16	Technical session 17	Technical session 18
Modeling and visualization Case Studies	Evaluation, control or measurements of indoor thermal environment	New technologies for heating and cooling or ventilation/AC	Natural, hybrid and mechanical ventilation
Room R1	Room R2	Room R3	Room R10

Tuesday, June 21,13:30 to 15:00			
Technical session 19	Technical session 20	Technical session 21	Work Shop 3
Modeling and visualization	Evaluation, control or measurements of indoor thermal environment	New technologies for heating and cooling or ventilation/AC	Advanced methods for air distribution in spaces
Room R1	Room R2	Room R3	Room R10

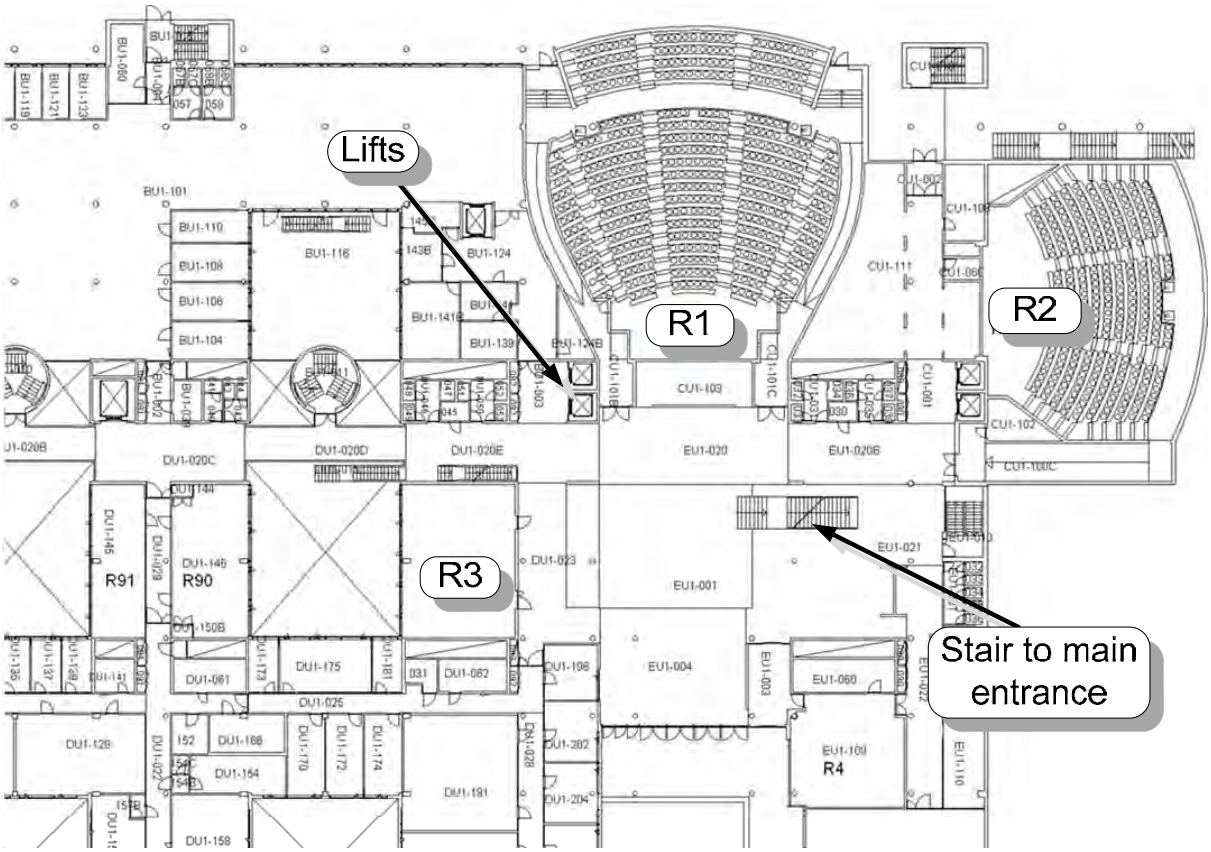
Tuesday, June 21,15:30 to 17:00			
Work Shop 5 Occupant's behavior related to ventilation energy use – EA/ECBCS Annex 53	Poster session 1	Poster session 2	Work Shop 4 Air motions and air exchange in Historical Buildings
Room R1	Room R2	Room R3	Room R10

Wednesday, June 22, 09:00 to 10:30			
Technical session 22	Technical session 23	Technical session 24	Technical session 25
Modeling and visualization	Evaluation, control or measurements of indoor thermal environment	New technologies for heating and cooling or ventilation/AC	Ventilation strategies preventing airborne infections Modeling and visualization
Room R1	Room R2	Room R3	Room R10

Wednesday, June 22, 10:45 to 12:15			
Technical session 26	Technical session 27	Work shop session 6 Short reports from the workshops	
Natural, hybrid and mechanical ventilation	Modeling and visualization		
Ventilation for low energy, passive houses/zero emissions buildings	Case Studies		
	New technologies for heating and cooling or ventilation/AC		
Room R1	Room R2	Room R3	

Wednesday, June 22, 12:30 to 13:00
Closing session

Where to go



Level U1

Level 5

The easiest way to get to R10 on the fifth level is to take the lift that is close to R1.

Technical session 1 Case studies

Paper No 296

RETROFIT OF HVAC RESIDENTIAL BUILDING USING VAV SYSTEM, INTEGRATED WITH THERMAL STORAGE SYSTEM

Qing Zhang[1], Ahmed Cherif Megri[2], Jay Puckett[3]
[1,2,3]Civil and Architectural Engineering Department, University of Wyoming, USA

With the cost of electricity constantly rising, energy conservation opportunities become ever more attractive. Green building grabs all the headlines, but retrofitting existing buildings is where the real savings are for most buildings.

This paper describes a proposed energy-efficient retrofit of a heating, ventilating, and air-conditioning system for a residential building from constant volume to variable air volume using TRNSYS program. The existing system consists of constant air volume heating/cooling and ventilating units. The proposed replacement consists of VAV system integrated with thermal storage system with supplemental solar thermal collectors. Fans with variable frequency drives have been used. Geothermal heat pump (GHP) system has been integrated to the whole system.

A parametric study has been performed to quantify the energy saving and to select the most appropriate and efficient configuration. As well, energy saving has been quantified for over a long time period.

Paper No 76

PERFORMANCE OF DISPLACEMENT VENTILATION IN A CANADIAN SCHOOL: A FIELD STUDY

Boualem Ouazia[1], Michel Tardif[2], Alexandra Thompson[1], Iain Macdonald[1], Daniel Booth[1]
[1]National Research Council Canada, Ottawa, Canada. [2]Natural Resources Canada, Ottawa, Canada.

Stratified ventilation systems use a fundamentally different approach to space conditioning than the systems found in the majority of non-residential buildings, which currently use a fully mixed and dilution approach to ventilation. In a stratified system air is introduced to a space close to floor level at a lower temperature than the desired set point. The air is then heated by occupants and equipment and the upward movement of the warming air removes contaminants from the breathing zone and exhausts them at ceiling level. This creates a non-uniform environment in terms of temperature and pollutant distribution, but acceptable conditions at the occupied zone. Previous research has shown that this type of system works well for regions where buildings require year-round cooling, however there are a growing number of buildings using this approach in Canada, where buildings require heating during winter months. This paper presents results from a field study conducted in a school equipped with a displacement ventilation system. The results show that the measured contaminant removal efficiency is better than that predicted in previous studies for heating conditions. In addition, key predictors of thermal comfort (the measured vertical air temperature difference and draft rating at ankle and head height) are also generally within limits set by ASHRAE standards. However, control issues as well as localized thermal discomfort were identified as negatively affecting overall system performance.

NUMERICAL AND EXPERIMENTAL EVALUATION OF VENTILATION IN LABORATORIES: A CASE STUDY

Paper No 32

Glenn Reynders, Dirk Saelens
Building Physics Section, Department of Civil Engineering, K.U.Leuven, Belgium.

Ventilation is a key performance requirement in laboratory design as it has to guarantee a safe and comfortable indoor environment. Current standards and guidelines on laboratory ventilation often impose high ventilation rates, increasing the energy need for ventilation, the environmental impact and the energy costs, at many large research facilities. This research focuses on the intra-zonal airflow in a standard laboratory set-up. The airflow and ventilation efficiency is computed with Computational Fluid Dynamics (CFD) and an extensive in-situ experimental case, in which different ventilation strategies are evaluated, has been conducted. The results indicate that the current design standards, which impose a minimal number of air changes per hour, cannot guarantee an optimal, energy efficient design. An optimal design starts from a comprehensive risk analysis. The CFD-simulations and experimental study show that an optimal design should not only be based on a minimal ventilation rate but also has to include an analysis of the impact of the location and type of the ventilation inlet and outlet, the room geometry and ideally the influence of occupants and laboratory appliances. Therefore, it can be concluded that a reformulation of the requirements for laboratory ventilation is appropriate, which in practice will lead to an increasing complexity of the ventilation design process. Although they require a good comprehension and implementation of correct physical properties, CFD-simulations are expected to become an interesting and even mandatory design tool for future, energy efficient laboratory ventilation.

Paper No 297

IMPROVEMENT OF ENERGY DEMAND PREDICTION USING A NEW APPROACH

Yao Yu[1], Ahmed C. Megri[2]
[1,2]Civil and Architectural Engineering Department, University of Wyoming, USA

HVAC systems are different in term of cost, comfort and performance. The behavior of HVAC system in each single room or building is unique and depends on many factors, including the system itself, the locations of diffusers and thermostats, the inside design conditions, the outdoor weather conditions, the control scheme associated with the HVAC system and so on. Similarly, the building energy demand prediction is also unique and associated to the HVAC system used, building envelop characteristics and weather conditions. This work investigates the use of a new approach to improve the energy demand computation quality and accuracy by the use of an integrated zonal/multizone model. Several case studies have been carried out, in which several thermal inertia have been used. Finally, thermal comfort indices, such as PMV and PPD have been used to predict the thermal comfort level within the two-zones building.

Paper No 40

ENERGY EFFICIENT VENTILATION BASED ON INDEPENDENT TEMPERATURE CONTROL OF EACH SUPPLY AIR INLET

Xiaoliang Shao, Xiaojun Ma, Xuan Jiang, Xianting Li
Department of Building Science, School of Architecture, Tsinghua University, China

Usually, it is only a few persons occupy a certain local area in a room for most of the time. In this case it is enough to just satisfy the thermal comfort of this local area rather than the whole space.In this paper, the energy saving potential by independent temperature control of each supply inlet is numerically studied based on a typical meeting room. 4 cases, i.e. 4 persons, 12 persons, 20 persons and 24 persons are set up to explore the effect under different scenarios. It is concluded that:

(1) Compared with the uniform temperature control method, the energy saving can be realized at a certain degree with the satisfaction of occupied area by using the method of independent temperature control;

(2) As the number of occupants increases, the difference of averaged supply air temperature between the independent control and uniform control schemes will be decreased and the energy saving potential will decrease; (3) For a fixed scenario, the different energy saving can be achieved under different schemes of supply air temperatures, where a certain optimization method is needed to determine the optimal supply air temperature. The preliminary analysis is supposed to provide reference to the creation of energy efficient environment by non-uniform air distribution.

Technical session 2 Demand controlled ventilation

Paper No 173

"PERFORMANCE" PROJECT: LARGE SCALE MONITORING OF OCCUPIED DWELLINGS EQUIPPED WITH DCV (DEMAND CONTROLLED VENTILATION) IN FRANCE

Jean-Luc Savin[1], Anne-Marie Bernard[2]
[1]Aereco S.A., France. [2]ALLIE AIR, France.

While energy performance of heat recovery systems does not need to be proved anymore in very cold climates, unbalanced ventilation solutions using DCV (Demand Controlled Ventilation) terminals can be considered as a relevant and efficient alternative in moderate, even cold climates. Their affordable cost and low maintenance need make these solutions particularly suitable for a mass dissemination, condition for reducing the impact of the building sector on the greenhouse effect. Since the birth of humidity controlled ventilation in France in 1984, numerous monitoring have proved its energy efficiency as well as its performance in term of IAQ. But never this technology was tested in such a scale as in "Performance" project. During two years from 2007, 31 occupied dwellings dispatched on two building sites in France have been monitored. Every minute, parameters such as pressure, airflow, temperatures, humidity and CO_2 were recorded in all the rooms of the dwellings. Results have demonstrated the efficiency of the used DCV system: IAQ was improved in all the rooms, especially in occupied bedrooms during night. CO_2 concentrations stayed at a low rate, and the condensation risk has been considerably reduced with the use of humidity controlled air inlets. On the energy side, the yearly average airflows have been reduced of 30% of the regulatory constant airflow, although most of these dwellings were over-occupied. Extrapolations to standard occupations have shown 50 to 55% of energy savings on airflows. This project has demonstrated the potential of DCV, which could be the next step for the optimization of heat recovery, in a mix solution.

Paper No 142

CONVERTION FROM CAV TO VAV – A KEY TO UPGRADE VENTILATION AND REACH ENERGY TARGETS IN THE EXISTING BUILDING STOCK

Mads Mysen[1,2], Peter Schild[2], Finn Drangsholt[1], Bjørn Tore Larsen[3]
[1]Oslo University College, Oslo, Norway. [2]SINTEF, Oslo, Norway. [3]Vekst Foundation, Oslo, Norway.

Demand controlled ventilation (DCV) can considerably reduce the ventilation airflow rate and energy use for fans, heating and cooling compared to constant air volume (CAV) ventilation. Based on the new EPBD Recast directive from the EU Parliament, there is a potentially enormous upcoming marked for converting from CAV to efficient DCV in existing commercial buildings.

However, several DCV-solutions are not very suitable for upgrading applications because they require redesign of airflow paths or introduction of several new components that are difficult to integrate into existing CAV-systems. A normal consequence is that ventilation upgrading is postponed, or the existing ventilation system is completely discarded and replaced. Such complete replacement of existing systems is probably not environmentally friendly, and has considerable additional costs like loss of estate rental income during the rebuilding period.

This paper presents DCV-systems that seem particularly promising for upgrading ventilation in existing commercial buildings. At the moment, DCV with variable supply air diffusers seems generally most suitable for upgrading from CAV to VAV, but project-specific requirements and circumstances must be emphasized before the final choice in each project is taken.

Paper No 159

SUPPLY AND EXHAUST VENTILATION SYSTEM WITH HEAT RECOVERY IN COMPARISON TO A DEMAND-BASED (MOISTURE-CONTROLLED) EXHAUST VENTILATION SYSTEM

M. Krus, D. Rösler, A. Holm
Fraunhofer-Institut for Building Physics, 83626 Valley, Germany

As a countermeasure to global warming, the energy demand of buildings is to be reduced by specific measures, for example thermal insulation or intelligent ventilation systems. A demand-based (moisture-controlled) exhaust ventilation system is assessed in comparison to a supply and exhaust ventilation system with heat recovery by means of computational investigations. This assessment of different ventilation systems is performed by means of the newly developed hygrothermal indoor climate simulation model WUFI®-Plus. By implementing the individual ventilation systems the energy demand, especially the primary energy consumption on the basis of applying various fuels, as well as the effects on the indoor climate and the C02 content of the indoor air are calculated and compared. Moreover, air change rates are investigated resulting from the use of a demand-based exhaust ventilation system. The calculations are based on a model apartment with a ground floor of 75 m2 and an assumed 3-person household. These investigations comprise 3 different climates in Germany (cold, medium and hot climate).

Despite the high heat recovery coefficient of the supply and exhaust ventilation system an only slightly higher energy use occurred for the demand-based exhaust ventilation system. If regenerative energy sources such as wood are used, primary energy consumption of the demand-based exhaust ventilation system is even lower in comparison to the supply and exhaust ventilation system with heat recovery. With demand-based exhaust ventilation system, the C02 concentration of the indoor air remains permanently below 1200 ppm.

Paper No 55

ENERGY REDUCTION AND IMPROVED AIR QUALITY USING MOS GAS SENSOR TECHNOLOGY FOR DCV

Simone Herberger[1], Heiko Ulmer[1]
[1]AppliedSensor GmbH, Reutlingen, Germany.

Energy efficient ventilation relating to good indoor air quality (IAQ) is a major task for building performance according to the requirements set by the Energy Performance of Buildings Directive (EPBD) in 2010. Applying demand-controlled ventilation (DCV) in buildings, using sensors for IAQ control which offer variable airflow rates adapted to the actual load conditions in buildings, will provide healthy and comfortable conditions while minimizing the energy consumption. The use of sensors for IAQ control is still rare because most state-of-the-art sensing technologies are insufficient for adequate IAQ control or suffer from inaccuracy and drift problems. Hardly any data on performance and energy efficiency of metal oxide semiconductor (MOS) gas sensor controlled ventilation can be found in literature.

Performance of a micromachined MOS gas sensor module for IAQ control in an office has been compared with further commercial sensor devices for DCV. Energy demand and resulting IAQ in the office for natural, time-controlled and demand-controlled ventilation has been investigated. The study accentuates the need for DCV and proves the functionality of the developed sensor module. Up to 70% less power consumption of the fan compared to time-controlled ventilation and IAQ conditions corresponding to IAQ level II of EN 15251 have been achieved for DCV.

Technical session 2, Theme: Demand controlled ventilation, Room R2, Monday 2011-06-20, 11:00 to 12:30

Paper No 138

A NOVEL DEMANDED-CONTROL VENTILATION STRATEGY FOR CO_2 CONTROL AND ENERGY SAVING IN MECHANICALLY-VENTILATED BUILDINGS

Tao Lu[1], Xiaoshu Lu[1,2,] Martti Viljanen[1]
[1]Aalto University School of Science and Technology, Espoo, Finland. [2]Finnish Institute of Occupational Health, Helsinki, Finland.

CO_2-based demanded-control ventilation (DCV) is one of most important control strategies concerning energy saving. However, although conventional CO_2-based DCV strategies, such as proportional and exponential controls, can ensure that minimum requirement of outdoor air by industry standard can be met for buildings/spaces, they are operated under the assumption of equilibrium condition which seldom occurs in practice and therefore there is still much space to improve on conventional strategies in terms of energy saving. In this paper, a novel and dynamic control strategy was developed for CO_2 control and energy saving in a kind of public buildings where their open hours are scheduled. The strategy utilized the schedule by setting the base ventilation rate for unoccupied periods and calculating ventilation rate dynamically at each occupied period by solving CO_2 mass balance equation so that indoor CO_2 concentration was kept near CO_2 set point at the occupied period. The utilization of schedule and consideration of dynamic effect in indoor CO_2 concentration brought much efficiency to energy saving for the new strategy. Industry standard (ASHRAE Standard 62-2007) about the minimum requirement of outdoor air was also followed in the new strategy. Experimental simulation was conducted over a sport training center. Results show that new strategy can save +34% of ventilation energy related to ventilation air compared to proportional control. The new strategy is simple, dynamic, flexible and efficient.

Paper No 107

PERSONALIZED VENTILATION AND CONDITIONING: INTEGRAL DESIGN STRATEGIES FOR NEAR ZERO IMPACT BUILDINGS

Wim Zeiler, Gert Boxem, Geert Filippini
Technische Universiteit Eindhoven, The Netherlands

A healthy and comfortable indoor environment should provide a pleasant and sustainable workplace for people in (office) buildings. The quality of the (thermal) indoor environment is important because it has a significant impact on the comfort experience, health and productivity of employees. The objective of this paper is to describe the design process for an individual microclimate to improve thermal comfort and reduce energy consumption for Near Zero Impact Buildings.

Technical session 3 Evaluation, control or measurements of indoor air quality

Paper No 78

IMPROVEMENT OF INDOOR AIR QUALITY BY USE OF AIR CLEANERS FOR ROOMS WITH DISPLACEMENT VENTILATION

Bing Gu[1], Joerg Schmid[2]
[1]University of Stuttgart, Institute of Building Energetics, Germany. [2]HLK Stuttgart GmbH, Germany.

A former investigation for the application of air cleaners in combination with mixing ventilation in rooms demonstrates: the application of air cleaners in rooms can effectively improve the indoor air quality and at the same time the supply of outside air can be reduced. In comparison with the mixing ventilation the displacement ventilation takes advantage of contaminant transport from rooms because of the strong indoor thermal sources. This aspect of the displacement ventilation in combination with portable air cleaners for a small office room is investigated in this paper. Numerical methods (CFD) are used as a tool for the 3D full scale room model.

The results demonstrate that air cleaners in combination with displacement ventilation have a high potential for improvement of indoor air quality. The displacement ventilation can enforce the effect of air cleaners. The position of air cleaners has an impact on their effect.

Paper No 98

SENSORY TESTS FOR BUILDING MATERIALS – STATUS OF THE STANDARDISATION

Birgit Müller[1], Jana Panaskova[1], Oliver Jann[3], Wolfgang Horn[3], Dirk Müller[4],Wolfgang Plehn[5]
[1]Technische Universität Berlin, Hermann-Rietschel-Institut, Germany and Hochschule für Technik und Wirtschaft Berlin, Germany, [3]BAM Federal Institute for Materials Research and Testing, Germany, [4]RWTHA Aachen; E.ON Energy Research Center; Institute for Rational Use of Energy in Buildings, Germany, [5]Federal Environment Agency, Germany

To control the emission of pollutants from building materials, a few different voluntary labeling systems have been introduced in Europe. In Germany mandatory “Principles for the health assessment of construction products used in interiors” have been applied for the technical approval of floor coverings since 2004. All of these both voluntary and mandatory systems combine in principle measurements of volatile organic compounds according to ISO 16000-9 and sensory evaluations (new ISO 16000-28).

In this paper a project financed by the German Umweltbundesamt (Federal Environmental Agency) is presented. The project deals with the questions how to combine emission tests with sensory evaluation and what the best sensory evaluation procedure is to achieve valid and accurate results. Flooring and flooring adhesives in particular are investigated in this project. In the next step the criteria for labelling those products without an unpleasant odour will be determined. The current status of ISO 16000-28 and 30 is also presented, and sensory limits and the evaluation methods will be presented and discussed.

Technical session 3, Theme: Evaluation, control or measurements of indoor air quality, Room R3, Monday 2011-06-20, 11:00 to 12:30

Paper No 88

AN AIR POLLUTANTS ANALYSIS OF URBAN RAILWAY TRAIN CABINS USING A MULTIVARIATE STATISTICS

Eunyoung Park[1,2], Duckshin Park[2]
[1]University of Science and Technology, Daejeon, Korea. [2]Korea Railroad Research Institute, Uiwang, Korea.

The issue of air quality within train cabins is attracting considerable attention of late, as public railway transportation travel has expanded exponentially. In particular, operational agencies and passengers are increasingly concerned about the concentration of particulate matters and CO_2 affecting air quality of passenger cabins. This study has observed PM10 and CO_2 concentration in the cabin of urban railway trains and analyzed the factors affecting air quality using a multivariate statistical analysis. Measurements have been conducted at Seoul metropolitan subway 8 lines and an additional Bundang line during three seasons.

The results show that the mean concentration level of the PM10 and CO_2 inside subway cabins is at the range of 62.6 to 108.0 μg/㎥ and 907 to 2,008 ppm, respectively. The CO_2 level in specific sections during the rush hours has exceeded national guidelines, which requires designated train ventilation controls. The analysis of the influence factor correlation and regression implies that the CO_2 level is severely influenced by the number of passengers and the PM10 level is also correlated with the number of passengers. In particular, the PM10 level in the cabins indicates a positive correlation with the outdoor PM10 level. In addition, the PM concentration, the PM concentration has been highly affected by the number of passengers and the distance between stations.

Paper No 182

DEVELOPMENT OF AN INDOOR AIR QUALITY AUDIT APPROACH FOR UNIVERSITY LIBRARY BUILDINGS

Ehsan Asadi, Ana Isabel da C. Rodrigues, J. J. Costa and Manuel Gameiro da Silva
ADAI – LAETA, Department of Mechanical Engineering, University of Coimbra, Portugal.

In most developed countries, people are spending most of their time in various indoor environments, including libraries. This reality has contributed to the growing concern about the indoor environment and, therefore, the indoor air quality (IAQ). Good IAQ in libraries provides a comfortable and healthy environment for the students and staff, to learn and work in, respectively.

Hence, it is important to ascertain the IAQ status in the learning environment. This study is aimed to establish and demonstrate the comprehensive IAQ audit approach for libraries, based on Portugal national laws. In this paper, a number of case studies are used to demonstrate the application of the IAQ audit and evaluate its comprehensiveness and usefulness to the facility managers.

The audit was conducted in 5 libraries of a University Campus in Portugal. The systematic IAQ approach involves measurement of physical parameters (temperature, relative humidity and concentration of suspended particulate matter (PM10)), monitoring of the concentrations of selected chemical indicators (carbon dioxide, carbon monoxide, formaldehyde and total volatile organic compounds (TVOCs)), and measurements of biological indicators (bacteria and fungi concentrations). Besides, air exchange rate was measured by the concentration-decay method using disseminated CO_2 as the tracer gas. In addition, a questionnaire was completed by the staff in order to provide a subjective assessment of IAQ.

The comprehensive IAQ audit revealed four main problems in the libraries: (1) insufficient ventilation rate; (2) too high particle concentration in some places of the libraries; (3) Poor filtration effectiveness in all air handling units (AHUs).

Technical session 3, Theme: Evaluation, control or measurements of indoor air quality, Room R3, Monday 2011-06-20, 11:00 to 12:30

Paper No 275

EXPERIMENTAL STUDY ABOUT HOW THE THERMAL PLUME AFFECTS THE AIR QUALITY A PERSON BREATHES

Inés Olmedo[1], Peter V. Nielsen[2],Manuel Ruiz de Adana[1], Piotr Grzelecki[2], Rasmus L. Jensen[2]
[1]Department of Chemical Physics and Applied Thermodynamics, Córdoba University, Córdoba, Spain. [2]Department of Civil Engineering, Aalborg University, Aalborg, Denmark.

The Personal Micro Environment (PME) depends directly on the heat transfer in the surrounding environment. For the displacement ventilation systems the convective transport mechanism, which is found in the thermal plume around a person, influences the human exposure to pollutants.

The aim of this research is to increase the knowledge of how the thermal plume generated by a person affects the PME and therefore the concentration of contaminants in the inhalation area.

An experimental study in a displacement ventilation room was carried out. Experiments were developed in a full scale test chamber 4.10 m (length), 3.2 m (width), 2.7 m (height). The incoming air is distributed through a wall-mounted displacement diffuser. A breathing thermal manikin exhaling through the mouth and inhaling through the nose was used. A tracer gas, N2O, was used to simulate the gaseous substances, which might be considered as biological contaminants, exhaled by the manikin.

The manikin was operated in three different heat fluxes with a value of: 0W, 94 W and 120 W.

During the experiments six concentration probes were situated in the room. Three concentration tubes were fixed on the surface of the manikin at three different heights: hips, chest and nose (inhalation). The three other tubes were situated in a vertical line at 0.50 m from the manikin and at the same three heights.

The results show the highest concentration of contaminants around the manikin when the manikin heat load is fixed to 0W. However, the concentration is significantly reduced in the case with 120 W, especially in the breathing area.

Paper No 241

PRELIMINARY EVALUATION OF NANOPARTICLE TRANSFER ACROSS THE DYNAMICAL AIR BARRIER OF A MICROBIOLOGICAL SAFETY CABINET

Vincent Cesard[1,2], Emmanuel Belut1, Corinne Prevost[2]
[1]Institut National de Recherche et de Sécurité (INRS), Vandoeuvre, France.
[2]Institut de Radioprotection et de Sureté Nucléaire (IRSN), Gif-sur-Yvette, France.

This paper presents preliminary results regarding the containment efficiency of a Microbiological Safety Cabinet (MSC) with respect to nanoparticles. The device is studied in an ISO class 4 clean room: a reference nanoparticle source is placed inside, and nanoparticle number concentrations are measured with a CPC (Condensation particle counter) in several locations inside and outside the cabinet. Results show a noticeable transfer across the air barrier of the cabinet; extrapolation to normal working conditions raises the concerns about the use of such cabinets when highly hazardous nanoparticles are manipulated.

Technical session 4 Aircraft cabins

Paper No 43

RAPID CONSTRUCTION OF A DIGITAL GEOMETRIC MODEL OF AN AIRLINER CABIN FOR CFD SIMULATIONS

Jiangyue Chao[1], Xuan Mu[1], Wei Liu[1], Huajie Wu[2], Junjie Liu[1], Qingyan Chen[1,3]
[1]School of Environmental Science and Technology, Tianjin University, Tianjin, China
[2]Tianjin Research Institute for Internal Combustion Engines, Tianjin, China
[3]School of Mechanical Engineering, Purdue University, Indiana, USA

To provide a more comfortable and healthier cabin environment to passengers and crew, airflow in airliner aircraft cabins should be further improved. Computational Fluid Dynamics (CFD) is an advanced method for simulating the environment while a three-dimensional (3D) digital model of cabin geometry is essential for generating mesh for the CFD simulations. This paper used a laser tracking system and inverse engineering to generate a digital model of an MD-82 aircraft cabin. Due to the large volume and complicated internal structure of the cabin, this research scanned the whole cabin into five sections, which can be connected to form the global structure of the fuselage. The scanning was also performed in different stations to acquire local point clouds that represent detailed geometric features, such as doors, toilet cabins, air supply diffusers, air return outlets, gaspers, and seats, etc. Then a 3D digital model of the cabin geometry was generated by assembling all the geometric parts together. The scanning took about nine person-days, and the model construction one person-month. The measuring error by the system was very small but approximations and assumptions were needed to reduce the workload and data size. The geometric model can also be easily used to calculate the space volume.

Paper No 112

THERMOFLUID BOUNDARY CONDITIONS IN A COMMERCIAL AIRCRAFT FOR CFD SIMULATIONS

Jizhou Wen[1], Wei Liu[1], Xuan Mu[1], Chao-Hsin Lin[2], Junjie Liu[1], Qingyan Chen[1,3]
[1]School of Environmental Science and Engineering, Tianjin University, Tianjin, China.
[2]Environmental Control Systems, Boeing Commercial Airplanes, Washington, USA.
[3]School of Mechanical Engineering, Purdue University, Indiana, USA.

As more passengers are traveling by air, the aircraft cabin environment has gained greaterpublic attention. Numerical simulations by Computational Fluid Dynamics (CFD) have become popular for cabin environment research,but CFD needs to be validated. This validation requires not only the air distribution data in a cabin but also the associated accurate thermofluid boundary conditions. This study showshow to measurethe thermofluid boundary conditions for an aircraft cabin, such as the temperature of interior surfaces and supply air parameters. The experiment used a thermocouple system, a tracer-gas monitoring system, hot-sphere anemometers(HA), ultrasonic anemometers(UA), flow meters, etc.,in aretired but functional MD-82 aircraft cabin. It wasfound that both the constant and decay tracer-gas methods can be used to measure ventilation rate. The combination of HA and UA were necessary to measure the complicated air velocity and direction from the supply diffusers. To minimize the impact of the outside weather on the temperature of the interior surfaces, it is effective to insulate the cabin fuselage. Our effort has demonstrated that it is possible to providea set of stable and accurate thermofluid boundary conditionsof the MD82 aircraft cabin for CFD simulations of the cabin environment.

Technical session 4, Theme: Aircraft cabins (Evaluation, control or measurements of indoor thermal environment and Modeling and visualization), Room R10, Monday 2011-06-20, 11:00 to 12:30

Paper No 254

VENTILATION IS NOT THE SAME AS VENTILATION – THERMAL COMFORT IN THE AIRCRAFT CABIN

J. Winzen, C. Marggraf-Micheel
German Aerospace Center, Aviation and Space Psychology, Hamburg, Germany.

In the research project Comfortable and Silent Cabin+, thermal comfort was analyzed in an aircraft cabin mock-up of a Dornier 728. Two realistic climate scenarios were realized as test conditions: cruise flight and climb flight, with different temperatures and air flow rates each. Objective and subjective data concerning several climate parameters and comfort perceptions were assessed by means of physical measures and questionnaires. In sum, 280 subjects participated in four investigations. Due to a differentiated measurement design, objective data concerning the local air stream could be gathered for ten seats only.

Our results confirm that the air flow at different parts of the body and at different seats is characterized by different velocities – and perceived as different by the subjects. Altogether, the stronger the air flow felt in cruise flight, the less comfortable it is rated ($r = -.42$, $p < .01$, $N = 70$). In climb flight, more air flow tends to be more agreeable ($r = .16$, n. s., $N = 70$). The correlation between air velocity and subjective ratings is not as clear. Combining objective and subjective data seems to be promising when predicting thermal comfort: more than 70 % of variance can be explained in path models. Objective data can be complemented significantly by subjective perceptions when predicting thermal comfort and should be taken into account in cabin design.

Paper No 154

CFD MODELING OF INTERNATIONAL SPACE STATION AND VISITING SPACECRAFT VENTILATION: EVALUATION OF DESIGN SOLUTIONS FOR COMPLEX ON-ORBIT OPERATIONS

Chang H. Son[1], Evgueni M. Smirnov[2], Nikolay G. Ivanov[3], Denis S. Telnov[3]
[1] The Boeing Company, Houston, Texas 77059, USA [2] St. Petersburg State Polytechnic University, Russia, [3] New Technologies and Services, St. Petersburg, Russia.

This contribution covers the recent experience in CFD modeling of the International Space Station (ISS) and visiting spacecraft ventilation that was accumulated during the assembly, integration and maintenance of the ISS modules. Several examples of CFD techniques application to estimation of space exploration vehicle cabin environment are discussed. The modeling has been performed with the ANSYS FLUENT software. The Reynolds-Averaged Navier-Stokes (RANS) approach was employed. The modeling verification was performed by means of a comparison of the RANS data with the experimental data as well as with Large Eddy Simulation results obtained for a pressurized ISS module.

The CFD-studies presented are divided into four groups. The first group covers regular ISS ventilation examinations. The focus here is on flow characteristics and spatial/time variations of carbon dioxide concentration within the ISS cabin. The studies of the second group deal with Space Shuttle and other visiting vehicles ventilation. The tasks of the third group cover examination and/or improvement of ISS modules ventilation when the stowage cargo or new devices with protrusions into Environmental Control and Life Support keep-out zones are installed inside an ISS module. Finally, the fourth group tasks are aimed at evaluation of design solutions for off-nominal and other contingency operations. In general, the study gives evidence that when gradients within the ISS module become more important, CFD modeling should be employed to address complex problems related to the quality of the cabin air breathed by crewmembers.

Technical session 4, Theme: Aircraft cabins (Evaluation, control or measurements of indoor thermal environment and Modeling and visualization), Room R10, Monday 2011-06-20, 11:00 to 12:30

Paper No 123

THERMAL COMFORT SIMULATION IN MODERN AIRCRAFT COCKPITS

Paul Mathis[1], Rita Streblow[1], Dirk Müller[1], Gunnar Grün[2], Andreas Wick[3], Jean-Christophe Thalabart[3]

[1]Institute for Energy Efficient Buildings and Indoor Climate, E.ON Energy Research Center, RWTH Aachen University, Germany. [2]Fraunhofer Institute for Building Physics, Valley, Germany. [3]Airbus Operations GmbH, Hamburg, Germany.

The non-uniform thermal environment in an aircraft cockpit during standard cruise conditions is simulated for a specified ventilation configuration and three different boundary condition sets. The air change efficiency is increased by changing the wall boundary condition from adiabatic to a specified temperature value and additionally, by installing air outlets in the ceiling. Due to strongly inhomogeneous flow conditions, locally resolved assessment of thermal comfort is required. The thermoregulatory answer of the pilots and a 3rd occupant is simulated by a multi-compartment thermal comfort model providing local and overall thermal sensation and comfort values. Local cold discomfort at outward pointing shoulders and feet is identified for the pilots resulting in moderate overall thermal comfort. The 3rd occupant is not expected to suffer from poor thermal comfort.

Paper No 158

COMPUTATIONAL ANALYSIS OF INTEGRATED PARAMETRIC MULTI-SEGMENTED HUMAN MANIKIN MODEL IN AIRCRAFT CABINS

Sandeep Rao Bolineni[1], Christoph van Treeck[1], Daniel Wölki1, Sebastian Stratbücker[1], Stephan Fiedler[1], Andreas Holm[1]

[1]Fraunhofer Institute for Building Physics (IBP), Holzkirchen, Germany.

The topic of this study is to adapt the Computational Fluid Dynamics (CFD) approach used for the thermal analysis of a parametric multi-segmented human manikin model in aircraft cabins. Considering the local thermodynamics in a non uniform environment is a very important aspect while predicting the thermal comfort of passengers in aircraft cabins. The purpose of this study is to develop and demonstrate a parametric multi-segmented human manikin model that predicts the response of the human body to varying environmental conditions, as this is relevant for the analysis of thermal comfort aspects in the aircraft cabins. The objective of this research is to obtain convective heat transfer coefficients for various environmental conditions, where the interface modelling of the seat climate and the manikin is of critical importance. In this regard a parametric thermal manikin model with 48 body segments has been developed based on the settings of a thermo-physiological model. Additionally transforming body parts according to an ergonomic model which relates topological dependencies is also allowed. For linking computational codes and engineering models, a developed middleware framework has been applied as well as a parametric geometric cabin model.

Technical session 5 Modelling and visualization

Paper No 250

COMPUTATIONAL STUDY OF PERSONAL VENTILATION WITH MULTIPLE OCCUPANTS AND VARIOUS CONFIGURATIONS

Jackie Russo[1], H. Ezzat Khalifa[1]
[1]Syracuse University, Syracuse NY, USA

For this work, three computational simulations were conducted involving the practical implementation of Personal Ventilation (PV). First, PV was compared to a conventional ventilation system in an office space with four cubicles and four occupants to evaluate the quality of air away from the air terminal. It was found that PV improves air quality for a person seated in front of the system without drastically changing the air quality in locations where the space may be periodically occupied. Next, a two cubicle office space was simulated to assess cross contamination between occupants when only one PV system is in use. From this work it was found that cross contamination was not significantly increased between occupants when contrasted against a simulation without PV. Finally, to maximize the practicality of PV in the indoor environment a single office simulation was conducted. Variations in the single office setup included different sensible and beneficial configurations of PV and it was found that these placements can improve air quality for an occupant without sacrificing special demands.

Paper No 131

MODEL FOR ASYMMETRIC COLLISION OF TWO ATTACHED PLANE JETS

Hannu Koskela[1], Pekka Saarinen[1], Risto Kosonen[2], Mika Ruponen[2]
[1]Finnish Institute of Occupational Health, Turku, Finland. [2]Oy Halton Group Ltd, Helsinki, Finland.

A collision of two attached plane jets at the ceiling is a common flow phenomenon in air distribution. The flow after the collision typically turns downwards and enters the occupied zone. The flow pattern and velocity distribution of the downward flow are of great importance for the thermal conditions in the occupied zone below the collision point. For design purposes of air distribution, a model is needed for predicting the position and direction of the downward jet as well as the velocity level in the occupied zone.

This work discusses the colliding jets from two ceiling mounted active chilled beams. Chilled beams are common devices for air distribution and cooling especially in office buildings. Two chilled beams were installed in a laboratory test room and the flow patterns were measured and visualized. Several test conditions were measured with different ratios of volume and momentum flows from the two units. The measurement results were earlier reported in Roomvent 2009 for the maximum velocity in the occupied zone. Estimates were also given for the collision point based on maximum jet velocity.

In the present study, the test conditions were simulated with CFD. The collision point and the direction of the downward flow were determined from these results. Comparisons were made with the experimental results. The location of the collision point was found to depend on the jet velocities in the boundary layer close to the ceiling surface. The direction of the downward flow depends on ratio of the momentum flow rates of the two jets. Based on the results, a simplified model was developed for the direction and position of the downward flow.

Paper No 120

HOW TO REPLICATE COMPUTATIONALLY A HUMAN PLUME WITH A SIMPLE GEOMETRY, AND HOW NOT TO

Maria-Alejandra Menchaca-B., Leon R Glicksman
Massachusetts Institute of Technology, Cambridge, MA

Buoyant plumes created above humans are amongst the most important elements in room airflow studies, and replicating them accurately is of critical importance. Considerable effort has been invested on creating numerical or real life thermal manikins that replicate the exact thermal and airflow properties of a human. However, the accuracy of these manikins comes at a high monetary and/or computational cost, and becomes a limitation when running simulations with multiple occupants. Consequently, occupants in large room airflow simulations are commonly represented by heated rectangular boxes or cylinders. While these shapes are certainly simpler to create than a thermal manikin, there has not been a detailed analysis of how well they emulate a real human plume. In this paper we study the airflow and heat transfer characteristics of several simple geometries using Computational Fluid Dynamics, and compare them to numerical results of a more complex humanoid geometry found in the literature, as well as to experimental data of velocity profiles above live humans. Six geometries are analyzed: a square box, a rectangular box, a simple cylinder, two different cylinders with a round top, and a humanoid shape. Plumes were modeled in uniform and stratified room environments. We find that the plumes generated above geometries with a round top match experimental results, while flat top geometries (cylinder and square box in particular) fail to do so. We also see a considerable effect of geometry height on the plume development, in stratified conditions. Finally, results show that, in uniform environments, the centerline velocity of plumes above different heated geometries with the same heat output converges only several meters above the heat source. We conclude that it is possible to replicate the flow above a human plume using round top cylinders. The use of flat top geometries is discouraged, unless the room height is considerably higher than the elevation of the occupant level and the air is fully mixed.

Paper No 96

MODELLING OF COLLIDING ISOTHERMAL JETS FROM TWO CEILING DIFFUSERS

Marjo Lahtimo[1], Hans Wigö[2]
[1]Fläkt Woods Oy, Akaa, Finland, [2]University of Gävle, Gävle, Sweden

The objective of the study was to create a simple CFD (Computational Fluid Dynamics) model of colliding isothermal jets from two identical ceiling diffusers facing each other and to study the influence of the distance between the diffusers on the resulting downward jet. The modelling was done using a commercial CFD (Computational Fluid Dynamics) code. Air flows were visualised with smoke in order to analyse the collision point and flow behaviour after the collision. Velocity measurements were carried out to obtain velocity profiles for the supply air diffusers to be used as initial conditions in the CFD simulations. In addition, measurements were made to determine velocity magnitudes in the collided jets at the occupied zone. Different turbulence models were used in the simulations and compared to the visualised and measured data. As a result a valid CFD model describing the behaviour of the two colliding isothermal jets was created. It was concluded that the CFD models adequately described the phenomena and provided useful data to beused in further studies.

Paper No 100

COMPARING THE EFFICIENCY OF THREE TURBULENCE MODELS WITH EXPERIMENTS CONDUCTED FOR A TWO DIMESIONAL WALL-JET IN A VENTILATED ROOM

[1]Taghi Karimipanah, [1]Mathias Cehlin
[1]Academy of Technology and Environment, Faculty of Engineering and Sustainable Development, University of Gävle, Gävle, Sweden

Turbulent wall jets belong to an important group of shear flows and differ from the free jets because of wall damping effects. When jet left the slot opening, the potential core is consumed at the point where boundary layer growth on the surface meets the shear layer expansion on the free boundary. The flow downstream of the core then becomes fully developed. When designing room ventilation. Practically, a wall jet discharging along the ceiling generates high velocity region restricted to the ceiling therefore freeing the occupied zone from draught. Under normal design conditions, the jet remains attached to the ceiling until the opposite wall is reached where it is then deflected downwards in to the occupied zone.

In order to the complexity of flow pattern and large size of volume in a ventilated enclosure it is always a difficult task to predict the flow parameters accurately.

This investigation deals with comparing an experimental data from well-designed ventilated room and CFD simulations using three commercially existed turbulence models. In a ventilated room it is known that there will be a transfer of momentum from the jet to the moving ambient and the jet undergoes repeated deflections at the room corners. There are recirculation bubbles at the corners and middle of the room which are not captured well by using simple two-equation models. The short-comings of the widely-used standard k-ε model to accurately predict the turbulence levels and streamline curvatures in a ventilated room imply that the more suitable but commercially acceptable models should be used.

This paper focuses on the airflow pattern by experimental and numerical simulation in a full-scale test room. The results show that higher order turbulence models are capable to capture the general flow pattern more correctly than the simple standard k-ε model.

Paper No 141

A CO-SIMULATION FRAMEWORK FOR SCALE-ADAPTIVE COUPLING BETWEEN HETEROGENEOUS COMPUTATIONAL CODES

Sebastian Stratbücker[1], Christoph van Treeck[1], Sandeep Rao Bolineni[1], Daniel Wölki[1], Andreas Holm[1]
[1]Fraunhofer Institute for Building Physics (IBP), Holzkirchen, Germany.

We present the scale-adaptive coupling between heterogeneous computational codes and experimental facilities for human-centred indoor thermal performance analysis. Currently, the analysis comprises human thermoregulation, convective and radiative heat transfer between a virtual manikin model and the environment as well as models for thermal sensation and comfort perception. The developed software framework presented in this paper enables the modular adaptation of existing and future simulation software and experimental devices for common integration into a uniform co-simulation environment. A middleware layer provides the runtime infrastructure which facilitates loose coupling of numerical sub-models on distributed computing platforms. It provides services to visualize intermediate results and to interact with the simulation during computation. Multiple scales are addressed by the simulation framework as the level of detail can be adapted, depending on the trade-off between accuracy and computational costs. Built-in numerical smoothing improves the quality and the stability of the simulation results. The architecture further supports the use of parallel computing resources like clusters or graphics hardware to increase the overall simulation performance.

Technical session 6 Natural, hybrid and mechanical ventilation

Paper No 49

COLLIDING ISOTHERMAL CEILING JETS AS A METHOD TO GENERATE VELOCITY VARIATIONS IN THE OCCUPIED ZONE

H Wigö[1], M Lahtimo[2]
[1]University of Gävle, Sweden, [2] Fläkt Woods Oy, Finland.

A new approach to create thermal comfort in classrooms without using mechanical cooling has been developed at the University of Gävle. The method is based on enhanced convective cooling where a jet blows from the ceiling down in to the occupied zone. To avoid draft problems the jet blows at short period of time. The first technical solution to generate these velocity variations was to let the supply air enter the room either through high velocity diffusers, mounted at the ceiling, or through low velocity diffusers. This paper presents another method to create these velocity variations. Two standard ceiling diffusers, each generating a jet attached to the ceiling, were placed opposite at different distances. Where the two jets did collide a new jet blowing downwards occurred. By changing the distribution of the supply air between the two diffusers, the position where the downward jet did enter the occupied zone could be controlled. The initial result shows that it is possible to control the downward jet within the distance between the two ceiling diffusers and it is also possible to distribute the supply air so that there is no downward jet and hence, low velocity in the entire occupied zone. The study will continue with other diffuser types and configurations and non-isothermal supply air.

Paper No 72

NIGHT PURGING A TWO-STOREY ATRIUM BUILDING BY NATURAL VENTILATION – A MATHEMATICAL MODEL

P. M. Lynch, G. R. Hunt
Imperial College London, London, UK.

The night ventilation of buildings can be incorporated as part of a passive ventilation strategy, the aim being to purge the building of warm air that has accumulated during the day and, thereby, to cool the building fabric in order that it may act as a heat sink during the following day. While several studies have shown night ventilation to be effective in improving thermal comfort and reducing cooling demand during the day, no guidance on how to design for an effective passive night purge has been available. The aim of the research described herein has been to gain insight into this behaviour through the development of a simplified mathematical model of the night purging of a two-storey atrium building by passive displacement ventilation. The model captures how the pressure differences produced by warm air that has accumulated in the storeys and atrium drive a ventilating flow, in which warm air from the storeys discharges into the atrium as turbulent thermal plumes. The model predicts the depth and temperature of the layers of warm air in each storey and the atrium as functions of time.

Our predictions reveal complex behaviours which may be categorised into three distinct purging regimes - each distinguished by the chronology of a number of key events during the purge. We define measures of the efficiency of a purge based on the volume of (cool) night air which is unavoidably mixed with warm air in the atrium prior to completion of a purge. Even for the most efficient purges, this volume is significant, and may exceed 40% of the initial volume of warm air to be purged.

Paper No 187

MEASUREMENT OF PRESSURE AND VELOCITY DISTRIBUTION AROUND CROSS-VENTILATED BUILDING AND ACCURACY STUDY OF CFD ANALYSIS

Tomohiro Kobayashi 1, Hisashi Kotani 2, Toshio Yamanaka 2, Kazunobu Sagara [2], Yoshihisa Momoi [2]
[1] Ritsumeikan University, Kusatsu, Japan.[2]Osaka University, Suita, Japan.

The final goal of this work is to establish an improved prediction method of wind-induced ventilation rate when large openings are provided with a room in series, where conventional prediction method based on the so-called Orifice equation cannot work well. This is because the conventional method cannot consider that the dynamic pressure could remain after passing through an opening, and the improved prediction method should be established based on actual phenomenon inside the stream tube. Since this needs the analyses using CFD prediction, this paper presents the accuracy of CFD, especially flow around the building, by comparing them with experimental results. Studied model has a rectangular configuration and it is assumed to include nine one-room residences where openings of the central room are opened. In analysing energy loss inside the flow path, both static pressure and velocity are important, and these quantities are measured by using five-hole pitot tube and PIV system in a wind tunnel. For the measurement using five-hole pitot tube, three calibration curves regarding wind angle, velocity magnitude, static pressure are first created. Based on these calibration curves, static pressure and velocity are simultaneously measured around the model. Meanwhile, velocity distribution was also measured by 2-D PIV because each experimental method has both strength and weakness. Those experimental results are to be compared with CFD results using Standard k-e Model, Reynolds Stress Model, and Large Eddy Simulation to study accuracy of the calculation. At the same time, the scope of each experimental technique is also studied. It is finally shown that Large Eddy Simulation has sufficient accuracy to predict pressure and velocity distribution around the building model.

Paper No 119

AN EXPERIMENTAL AND NUMERICAL ANALYSIS OF FORCED MIXING VENTILATION FOR A TRANSITIONAL SLOT REYNOLDS NUMBER

Twan van Hooff[1,2], Bert Blocken[1], Thijs Defraeye[2,3], Jan Carmeliet[4,5], GertJan van Heijst[6]
[1]Building Physics and Systems, Eindhoven University of Technology, Eindhoven,The Netherlands.[2]Laboratory of Building Physics, Katholieke Universiteit Leuven, Leuven, Belgium. [3]BIOSYST-MeBioS, Katholieke Universiteit Leuven, Leuven, Belgium. [4]Chair of Building Physics, Swiss Federal Institute of Technology Zürich (ETH), Zürich, Switzerland. [5]Laboratory for Building Science and Technology, Swiss Federal Laboratories for Materials Testing and Research (Empa), Dübendorf, Switzerland. [6]Fluid Dynamics Laboratory, Eindhoven University of Technology, Eindhoven, The Netherlands.

Forced mixing ventilation is a commonly used ventilation principle in which air is forced into the upper part of the room at relatively high speed. Attachment of the wall jet to the ceiling, also known as the 'Coanda effect', is used to ensure that the air does not enter the occupant zone too early, thus preventing discomfort for the room occupants. Most mixing ventilation studies in the past have been conducted for wall jets with slot Reynolds numbers that are considered to be in the turbulent regime, while transitional flows can be present for lower slot Reynolds numbers. Previous studies have indicated possible deficiencies of the commonly used Reynolds-Averaged Navier-Stokes (RANS) equations in combination with a turbulence model to provide closure, when applied for transitional flows.

This paper presents the first results of an experimental and numerical study of forced mixing ventilation at transitional slot Reynolds numbers, with focus on assessing the accuracy of steady RANS in combination with four frequently used turbulence models in Computational Fluid Dynamics (CFD) for indoor airflow. Visualisations with fluorescent dye seem to indicate that for a slot height (h/L) of 0.1 the flow is transitional for slot Reynolds numbers from about 800 to 2,900. 3D steadystate

RANS simulations in combination with four turbulence models have been compared with timeaveraged

PIV measurements, for h/L = 0.1 and a slot Reynolds number of 1,500. In general, the SST k-ω model shows the best agreement with the PIV measurements, whereas the standard, realizable and RNG k-ε models show rather large deviations, at least in this particular case. Further validation studies should be made to analyse the accuracy of the CFD simulations for other slot heights and other slot Reynolds numbers.

Paper No 130

EXPERIMENTAL AND CFD STUDIES OF NATURAL VENTILATION IN EXTERNAL RESPIRATION DOUBLE-SKIN FACADE

Cheng Li, Zhen Zeng, Xiaofeng Li
Department of building science, Tsinghua University, Beijing, China

Double-skin facade (DSF) is becoming widely used in commercial buildings. To study the natural ventilation performance of external respiration DSF, a complete series of field experiments were performed in a DSF test cell at Tsinghua University of China. Temperature distributions and ventilation rates were measured under different solar altitude angles and outdoor wind environments. The computational fluid dynamic (CFD) method was also used to study the ventilation performance, and showed good agreement with experimental results. Porous media was applied in CFD modeling to reduce the number of meshes and proved to have good computation accuracy. Results of experiments and CFD modeling in three different cases were analyzed. Laws of temperature distributions and airflow motions in different conditions were summarized and several rules for CFD modeling were promoted.

Technical session 7 Ventilation for low energy, passive houses/zero emissions buildings and Codes and regulations

Paper No 105

ALLOCATION OF MULTI-ZONE SPACE FOR BETTER VENTILATION EFFECT

Fu Xiao, Shengwei Wang, Yaping Lei
Department of Building Services Engineering, The Hong Kong Polytechnic University,Hung Hom, Hong Kong, China

Due to uneven thermal loads and occupancy densities, outdoor air ventilation in multi-zone VAV air conditioning systems is rarely satisfactory. When a multi-zone space is severed by one air handling unit, the fresh air fractions in the air delivered to multiple zones are the same. However, the required fresh air fraction of each room is usually different, which is determined by the zone cooling load and occupant number. As a result, energy waste and poor indoor air quality occur with overventilation and under-ventilation. A significant amount of research has been conducted on optimal control to improve multi-zone ventilation effect. However, no ultimate solution to this problem was reached yet. This paper addresses the multi-zone ventilation problem from a new angle other than active control of ventilation systems. Considerate space allocation can alleviate the unevenness of cooling loads and occupancy densities, and therefore minimize the differences among the fresh air fractions required by different zones.

This paper compares two space allocation schemes for a typical multi-zone office in terms of energy consumption and indoor air quality. A VAV air handling unit serves the office. CO_2-based demand controlled ventilation strategy is adopted in both schemes to determine the total fresh air flow rate dynamically. Simulation tests are performed on the platform of EnergyPlus. The simulation results showed that appropriate space allocation could greatly improve the indoor air quality without increasing the total fresh air flow rate and energy consumption.

Paper No 318

DOUBLE-SKIN FACADE TECHNOLOGY FOR ENERGY-EFFICIENT COMMERCIAL BUILDING REFURBISHMENT IN NORWAY

Matthias Haase[1,2], Tore Wigenstad[2]
[1]Norwegian University of Science and Technology, Trondheim, Norway.
[2]SINTEF Building and Infrastructure, Trondheim, Norway.

This work is part of the project “Low Energy Commercial Buildings” (leco) run by SINTEF Building and Infrastructure. The project’s principal objectives are to develop more energy-efficient buildings The project includes analyses of building envelopes applied in different kinds of climates, different uses, and different construction methods.
Energy-efficient building refurbishment in Norway is becoming a main focus. With the help of dynamic computer simulations of energy and indoor environment for a case building in Norway the impact of an additional ventilated glass facade on energy use and indoor environment was analyzed. A focus was put on a comparison of energy use and thermal comfort levels of various cases. Here, it became obvious that airflow in the double skin façade needs to be integrated into the existing ventilation system.
Main parameters to study were:

- different construction standards (air tightness, thermal bridges, and facade design) and its energy use implications
- simulation robustness in dependence of different assumptions (thermal bridges in and air tightness of the existing building)
- airflow control strategies and its energy use implications
- comfort criteria and energy issues (thermal vs. visual comfort vs. heating cooling demand)

The results show that significant efforts are needed in order to establish ventilated double-skin facades in Norwegian buildings as high energy efficient solutions. In particular, significant improvements of construction details regarding insulation levels and air tightness of the envelope are needed. The best solution provides a heating energy use reduction of 59% but the importance of level of details became obvious. Also, a clear ventilation strategy has to be found.

The design of energy robust, energy efficient, and comfortable buildings was possible with advanced building simulation. The strategy developed for improving building performance with an additional ventilated glass layer is an important step towards a more sustainable building stock in Norway.

Paper No 247

METHODOLOGY FOR ASSESSING VENTILATION SYSTEMS IN LOW ENERGY BUILDINGS

Juslin Koffi, Francis Allard
University of La Rochelle, France.

The ventilation of “low-energy buildings” leads to specific requirements for which researchers should bring appropriate solutions in order to both guaranteeing lower energy consumption and good indoor air quality. In fact, the energy consumption due to ventilation, mainly the necessity of heating the fresh air during winter, as well as the consumption of ventilation auxiliaries can represent a great part of the total energy consumption of these buildings. In the context of a French project on the field evoked above, some simulations are carried out for assessing the performances of different innovative ventilation strategies in four low-energy buildings: i.e. a single-family house, an apartment in a residential building, an open-office building and a school building. The project aims to find out the suitable ventilation system for each type of building; the final objective is to define a reference system for each type of building and then establish equivalence principle for the other systems.

This paper deals with the first results in the apartment in a multifamily building: a balanced demanded-controlled ventilation (DCV) system based on occupancy is compared to standard balanced ventilation system. In the studied cases, the energy consumption is in adequacy with the energy requirements. The results also tend to show a better indoor air quality when using presence DCV as far as the air supply is adapted to the fresh air need.

Paper No 261

COMPARISON OF VIRTUAL SPHERE AND CFD TRANSIENT MODELING OF NIGHT COOLING OF BUILDINGS WITH NATURAL VENTILATION

Marco Simonetti[1], Gian Vincenzo Fracastoro[1], Marco Perino[1]
[1]Polytechnic of Turin, Turin, Italy

In this work a typical case study of night cooling through natural ventilation of buildings has been simulated, using a detailed transient conjugated heat transfer CFD (Computational Fluid Dynamic) model, and compared with two simplified transient models, one with and one without the “virtual sphere” building mass description. The simulations concern the discharge phase over a single night. Natural air flow rates are calculated in different ranges of initial air temperature differences between building mass and outdoor air. Radiation heat transfer is considered.

The comparison with virtual sphere model has revealed unacceptable sagreement, while the simplified model with “normal” mass description gave values fairly similar to CFD simulations.

Technical session 7, Theme: Ventilation for low energy, passive houses/zero emissions buildings and Codes and regulations, Room R3, Monday 2011-06-20, 13:30 to 15:00

Paper No 319

INFILTRATION MODELS AND HEAT LOSS IMPLICATIONS IN OFFICE BUILDINGS

Matthias Haase[1,2], Tore Wigenstad[2]
[1]Norwegian University of Science and Technology, Trondheim, Norway. [2]SINTEF Building and Infrastructure, Trondheim, Norway.

The project's principal objectives are to develop more energy-efficient building envelope assemblies and new methods for the design of building envelopes in harsh climates, resulting in more accurate and geographically dependent design guidelines. The project includes analyses of building envelopes applied in different kinds of climates, different uses, and different construction methods.

The total energy consumption is simulated for three different construction standards in different locations in Norway and the results are compared. First, energy budgets according to Norwegian regulations (TEK 2007) were calculated. Then, the energy saving potential of air tightness of the building envelope was evaluated. Here, different calculation methods for estimating energy use due to infiltration were applied, including simplified models and detailed advanced calculations taking site factors, building specific factors, and other climate conditions into account.

The results show that significant energy savings are possible with stricter air tightness of the envelope. The infiltration and related heat losses (and cooling demands) depend on the model used to predict this. Some models simplify infiltration and thus might give wrong suggestions for a building design. A careful consideration of climate related parameter is needed particularly for super-efficient envelope systems which can further enhance energy robustness, energy efficiency and comfort of the future building stock in Norway.

Air tightness of the building envelope is an important factor in energy efficient building design in Norway. However, traditional simplified methods that calculate infiltration rates related energy consumption can mislead in an early design stage by over or under estimating heating demand by up to 25%. Detailed calculations of site specific conditions are essential in order to optimize thermal building envelope design in cold climates like Norway.

Paper No 33

AIR CHANGE RATE MEASUREMENTS UNDER NATURAL PRESSURE GRADIENT USING TRACER GAS METHOD

Matthieu Labat[1,2], Géraldine Garnier[1], Monika Woloszyn[2], Jean-Jacques Roux[2]
[1]CSTB - Centre Scientifique et Technique du Batiment, 24 rue Joseph Fourier, 38400 Saint Martin d'Heres
[2]CETHIL, Universite de Lyon, CNRS, INSA-Lyon, CETHIL, UMR5008, F-69621, Villeurbanne, France, Universite Lyon 1, F-69622, France

In order to quantify air change rate in a wooden frame house, tracer gas technique using carbon dioxide has been selected to add information to classical air-tightness measurements. Moreover, a general correlation between outdoor conditions and air change rate has also been aimed, using inverse method.

Measurements were done from autumn to winter and are ranging from 3 to 18 m³/h. The correlation showed very good results, leading to 4.4% accuracy in air change rate calculations, and allowing quick identification of the influent parameters.

Technical session 8 Evaluation, control or measurements of indoor air quality

Paper No 101

INDOOR AIR QUALITY AND ENERGY EFFICIENCY IN MULTI-APARTMENT BUILDINGS BEFORE AND AFTER RENOVATION

Ilze Dimdiņa[1], Ēriks Krūmiņš[2], Arturs Lešinskis[1,2], Laimdota Šnīdere[3], Viktors Zagorskis[1]
[1]Riga Technical University, Riga, Latvia.
[2]Latvia University of Agriculture, Jelgava, Latvia.
[3]Ltd Kaimiņiem.lv, Riga, Latvia.

Previous experience shows that renovation of ventilation system is not a typical component of renovation of multi-apartment buildings (MABs) in Latvia. As a consequence, rooms are not ventilated enough and indoor air quality (IAQ) is reduced. This, in turn, leads to progression of the so called sick building syndrome (SBS). IAQ measures in 13 unrenovated MABs shows that CO2 concentration in 19 of the examined 30 apartments was ≥1000, indicating insufficient ventilation [2].

We present an ongoing project which aims at monitoring IAQ in MABs before and after renovation. We collect measures of indoor air microclimate parameters in 8 (eventually 12) apartments of 5 MABs in Riga, during the heating season of 2010/2011. The measures are collected in two types of standard construction buildings, the so called project type 464 and project type 602, in three and two buildings of each type, where one is renovated and the other is not renovated. Now in 6 apartments of three type 464 buildings (on the first, middle and top floor) and in 2 apartments of two type 602 building (top floor), sensors of air temperature, relative humidity, CO2 concentration level, and air flow velocity are located to find the average value of IAQ parameters, and to make calculations of indoor air exchange in each apartment. At the same time, outdoor air temperature and relative humidity are measured. The total number of measured parameters is 55 (eventually planned till 98). The measurements are made every minute, and the paper presents data for the duration of 1 to 3 months. The results of the measures make it possible to find and to compare IAQ parameters and energy efficiency in MABs before and after the renovation.

Paper No 246

VENTILATION FILTER EFFICIENCY FOR BIRCH POLLEN ALLERGENS – EXPERIMENTAL DATA FROM ONE POLLEN SEASON

[1]Magnus Mattsson, [1]Lars T Andersson, [2]Kerstin Alm Kübler, [2]Agneta Ekebom, [3]Anders Jansson, [4]Leif Holmquist and [4]Olof Vesterberg,
[1]University of Gävle, Sweden
[2]Swedish Museum of Natural History, Sweden
[3]Stockholm University, Sweden
[4]Consultant at University of Gävle, Sweden

In an experimental study, ventilation filters of high quality (F7 and F9) were tested regarding their efficiency in collecting birch pollen allergens in outdoor air. The birch pollen grain concentration in outdoor air was measured at the same time as pollen allergen and particle number concentrations were measured before and after the tested ventilation filters, thus enabling collection efficiency calculations. Simultaneously, the size distribution of birch pollen allergens was measured in outdoor air using a cascade impactor. The study confirms previous indications that pollen allergens occur in outdoor air in particles much smaller than pollen grains, and can penetrate ventilation filters to a larger extent than might be expected. This entails that although the high quality filters collect most of outdoor air pollen allergens, a significant exposure dose to these allergens can occur in the indoor environment, especially when considering the fact most people stay much more indoors than outdoors.

The study also confirms previous similar indications attained with *grass* pollen allergens, in that the allergenic particles tend to penetrate ventilation filters to a greater extent than other airborne particles.

Paper No 125

IAQ EXPOSURE OF SLEEPING OCCUPANTS UNDER DIFFERENT RESIDENTIAL VENTILATION CONFIGURATIONS

Jelle Laverge[1], Arnold Janssens[1]
[1]Faculty of Engineering and Architecture, Ghent University, Gent, Belgium.

Ever tried coming up with a witty question at a conference when your baby child kept you up all night? Of all things done at home, sleeping may have the largest impact on your economic potential. On average, people spend about 1/3rd of their lifetime in their bedroom. Most assessment schemes for residential ventilation only look at the total exposure in all spaces within the dwelling. Physiological and sensory response of sleeping persons to IAQ parameters differs from that in the normal wake state, for which the commonly used performance indicators are developed. Therefore, the exposure within the dwelling should be broken down into exposure in living and bedroom areas for performance assessment.

This paper assesses the exposure of the occupants of a statistically representative dwelling to human bio-effluents and to humidity under 4 different residential ventilation system configurations with multi-zone simulations. Monte Carlo techniques make it possible to assess the sensitivity of the results. The adopted approach varies the most influential boundary conditions such as wind pressure coefficients and occupancy.

The results demonstrate that the expected exposure to unacceptable air quality in the bedrooms is up to 16 times higher than that in the rest oft he dwelling. The results will be used as boundary conditions for experimental investigations on the physiological response to IAQ exposure.

Paper No 236

IMPACT OF HEATING SYSTEM ON AIR VELOCITIES IN A MEDIEVAL STONE CHURCH

[1]Magnus Mattsson, [2]Tor Broström, [1]Elisabet Linden, [1]Svante Lindström, [1]Mats Sandberg
[1]University of Gävle, Academy of Technology and Environment, Sweden
[2]Gotland University, Department of Culture, Energy and Environment, Visby, Sweden

The air flow pattern and magnitude of air velocities in churches and other historic buildings are of interest since they influence the deposition rate of airborne particles on surfaces, and hence affect soiling of valuable artifacts of different kinds. Increased air movements might also cause enhanced sooting from candles and it influences thermal comfort of people. The type of installed indoor heating units is likely to be important here since these usually induce substantial air movements through natural or forced convection. In an experimental field study, two different heating systems were compared regarding their effect on room air velocities in a medieval stone church: air-to-air heat pumps with indoor fan convectors vs. a combination of bench heaters and radiators. Hot-sphere anemometers were used to record air velocities in the near-zone of the heat pumps and their surroundings, and 3-D sonic anemometers were used to measure downdraught air velocities at the surfaces of a wall and a window. Smoke was used to visualize air flow patterns.

It was found that the heat pumps caused strong buoyant air jets that rose to the ceiling, but that the air velocities were rather low outside of these jets. The bench heaters caused buoyant plumes, which also seemed to attain rather high air velocities and reach the ceiling. As regards downdraught along wall and window, no significant difference between the two heating systems could be seen, although there was a tendency towards slightly higher air velocities at these surfaces when the heat pumps were in use. Since the air flow pattern at the surfaces appeared similar, also the particle deposition mechanisms and soiling rate can be expected to be similar.

Technical session 9 Evaluation, control or measurements of indoor thermal environment and New technologies for heating and cooling or ventilation/AC

Paper No 167

THERMAL COMFORT EVALUATIONS IN UNDER-FLOOR AIR DISTRIBUTION SYSTEM WITH DIFFERENT BOUNDARY CONDITIONS

Yuanda Cheng, Jianlei Niu
The Polytechnic University of Hong Kong, Hong Kong, China

Compared with traditional mixing ventilation, the under-floor air distribution (UFAD) system may achieves better performance in energy saving and thermal comfort and has been developed rapidly in recent decades. As one of the most important devices, the swirling diffuser has been studied for years because of the great impact on the performance of UFAD system. This paper mainly focused on the influences of throwing angle on the capability of swirling diffusers from the view point of thermal comfort. Four different throwing angles were simulated and the results indicated that the throwing angle has significant effect on both the local and whole thermal comfort. Meanwhile, as a convenient method to evaluate thermal comfort in stratified environment, the ISO 14505 index is worthy to be further developed. Also, three thermal regulation modes for the detailed manikin have been investigated. The comparison results revealed that the impact of the regulation principle on the equivalent temperature was diminished when clothing resistance was considered in the simulation.

Paper No 169

EXPERIMENTAL STUDY ON THE MAXIMUM VELOCITY DECAY OF ATTACHED PLANE JET WITH DIFFERENT TURBULENCE INTENSITIES

Guangyu Cao[1], Mika Ruponen[2]
[1]VTT Technical Research Centre of Finland, Espoo, Finland.
[2]Halton Oy, Kausala, Finland.

Supplying air into rooms properly without causing a sensation of draught is a challenging task for designers and air diffuser producers. The insufficient understanding of how turbulence intensity will affect the jet flow results in using less effective ventilation system in room conditions. The objective of this experimental study is to identify the effect of airflow turbulence intensity on the airflow maximum jet velocity decay of an attached plane jet.

All the measurements were conducted in two test chambers where two different types of slot diffusers were used to produce attached air jet under isothermal conditions. The turbulence intensity of the jet discharging from the slot was approximately 1% and 20% in two test chambers, respectively. The results of this study reveal the maximum velocity decay of the attached plane jet will be influenced by both turbulence intensity and initial Reynolds number of the supplied air. Compared with the effect of Reynolds number on the velocity decay, however, it's still not unambiguous how much the turbulence intensity of the jet will affect the maximum jet velocity decay. More measurement results are needed to clarify the influence of the turbulence intensity on the maximum velocity decay. The measurement result will contribute to the development of more energy efficient ventilation strategy.

Technical session 9, Theme: Evaluation, control or measurements of indoor thermal environment and New technologies for heating and cooling or ventilation/AC , Room R3, Monday 2011-06-20, 15:30 to 17:00,

Paper No 106

ADVANCED CONTROL ON THE LEVEL OF INDIVIDUAL WORKPLACE

Wim Zeiler, Gert Boxem, Rinus van Houten
Technische Universiteit Eindhoven, Netherlands

Intelligent scheduling and control of the demand of energy can achieve a better use of sustainable sources like sun and wind. By splitting the thermal comfort in a less strict global part and a personally controlled part at each individual workplace, two goals are to be achieved: energy saving due to less strict global comfort requirements; and an improved thermal comfort at each workplace. To realize these goals an advanced control setup, based on a multi-agent system (MAS) has to be developed. This paper gives the outline for the development of the agent system at the individual workplace. The whole agent system is set up as an extension of the 'Intelligent grid' applications currently under development. The article provides a historical overview and a setup for experiments for 'optimal workplace control'.

Paper No 219

THE EFFECT OF HEAT LOAD ARRANGEMENT ON THE PERFORMANCE OF RADIANT PANELS

Panu Mustakallio[1], Arsen Melikov[2], Marcin Duszyk[2], Risto Kosonen[1]
[1]Oy Halton Group Ltd., Helsinki, Finland.
[2]International Centre for Indoor Environment and Energy, DTU Civil Engineering, Technical University of Denmark, Lyngby, Denmark.

The objective of this study was to identify the importance of heat load distribution on the cooling power of radiant panels. A full-scale test room was equipped with two top insulated ceiling radiant panels installed near the long walls. Radial multi-nozzle supply air diffuser was installed in the middle of the ceiling so that the supply air jet was flushing the radiant panels. The test room was constructed according EN-standards for accurate measurement of cooling power of the radiant unit. The cooling power of the panels was studied in two situations: 1) without supply air to compare two standardized testing methods for chilled beams in the case of radiant panels when the heat loads are conducted through test room walls or when using uniformly distributed heated dummies in the test room, and 2) with supply air in the similar cases as earlier and also in the case when the internal heat loads are located only in one end of the test room. The air temperatures and black ball temperatures were measured in the test room, and supply air jet pattern was visualized with smoke. Two standardized testing methods for the chilled beams gave nearly the same cooling output for the radiant panels. The most significant effect of heat loads on the performance of the radiant panels occurs when heat loads are located unevenly and their convection flow turns or weakens the supply air jet flushing the radiant panels. This should be taken into account in the design when supply air is used to enhance the operation of the radiant panel.

Paper No 179

A STUDY ON THE PERFORMANCE EVALUATION OF THE CAPILLARY TUBE RADIANT FLOOR COOLING SYSTEM USING GEOTHERMAL ENERGY

Jin-Ok, Kim[1], Sung-Kyun, Jung[1], Taeyeon, Kim[2], Seung-Bok, Leigh[2]
[1]Graduate Student, Department of Architecture, Yonsei University, Korea
[2]Professor, Department of Architecture, Yonsei University, Korea

The purpose of this study is to evaluate the indoor environmental conditions for comfort and cooling performance of the capillary tube using geothermal energy. The experiments are performed to evaluate the cooling performance of the capillary tube radiant cooling and to confirm the possibility of

continuous cooling through the actual field experiments. The results of this study are as follow. In peak load, the Capillary Tube Radiant Floor Cooling System using geothermal energy can be satisfied with the indoor setting temperature without other heat sources. The set room air temperature is maintained while the floor surface temperature, vertical temperature distribution and PMV remain within the comfort range.

Technical session 10 New technologies for heat recovery system and Codes and regulations

Paper No 44

AIR QUALITY AND VENTILATION SYSTEMS ENERGY EFFICIENCY IN PUBLIC BUILDINGS

Yuri D.Gubernsky[1], Irena.V.Gurina[2], Evgeny O. Shilkrot [3]
[1] Sysin Research Institute of Human Ecology and Environmental Hygiene, Moscow, Russia
[2] ENONTEK Ltd.,Moscow, Russia
[3] The Central Research Institute for Industrial Buildings, Moscow, Russia

In the article is described the issue of energy expenses reduction in ventilation system of public buildings. Ventilation system, in the given content, is a system for maintenance the air quality in the premises, i.e. the safe (optimum or admissible) concentration of harmful pollutants in the room. Air exchange reduction, without decreasing comfort level in the room can provide adequate increase of energy efficiency of ventilation systems.

In the article attempt to establish quantity of a sufficient air exchange indoors by a Pollution Balance Method on an example of administrative building is made. Indoor air pollutants are generated by the occupants (mainly it is exhaled air) and by premises (building exhausts, furniture, carpets, etc.). As an equivalent (qualitative and quantitative) of harmful pollutants in the room is taken carbondioxide.

As a result of the analysis of Russian and foreign researches are offered the relationships, which allow to calculate necessary air exchange on CO_2 balance level indoors to provide its safe level depending on its concentration in supply air.

On the basis of CO_2 level in indoor and outdoor air, calculations of necessary air exchange rate are made for three categories of office premises. Results of calculations show that in premises of the I (highest) category as with high, and low indoor pollutants level, power input to maintain necessary ventilation rate exceeds admissible limits.

A way out from a current situation is in use of scrubbers for CO_2 removal out of recirculated or incoming air. In the article is described the possibility of CO_2 absorbers application for decreasing energy consumption of ventilation system and an estimation of energy savings on the example of application of CO_2 absorbers in an office building ventilation system.

Paper No 18

FIRST EXPERIMENTAL RESULTS OF A DESICCANT BASED EVAPORATIVE COOLING SYSTEM IN ADANA

Đrfan Uckan[1], Tuncay Yılmaz[1], Orhan Buyukalaca[2], Ertac Hurdoğan[2]
[1] Department of Mechanical Engineering, Çukurova University, Adana, Turkey
[2] Department of Energy Systems Engineering, Korkut Ata University, Osmaniye, Turkey

Evaporative air-conditioning can be coupled with desiccant technologies to expand the range for comfort cooling applications. A desiccant assisted evaporative cooling system is used to dehumidify the ventilation air first with rotary desiccant wheel, and then to apply evaporative cooling to cool the air to the desired supply temperature.

First experimental results of a desiccant based evaporative cooling system are presented in this paper. The regeneration air at 100 ^{o}C, fresh air and waste air are used at constant flow rate of 4000 m^3/h. Experiments were carried out instantly by measuring temperature, pressure, power and humidity on the system. A continuous operation of the system during a typical day of summer season in a hot and humid climate showed that the temperature of fresh air received from the external environment at 31 oC is cooled down to approximately 19 ^{o}C supply temperature and the temperature of the air conditioning room is maintained at approximately 25 ^{o}C.

Paper No 268

AIRFLOW CHARACTERISTICS AT THE BREATHING ZONE OF A SEATED PERSON: PASSIVE CONTROL OVER THE INTERACTION OF THE FREE CONVECTION FLOW AND LOCALLY APPLIED AIRFLOW FROM FRONT FOR PERSONALIZED VENTILATION APPLICATION

Zhecho Bolashikov[1], Hideaki Nagano[2], Arsen Melikov[1], Clara Velte[3], Knud Erik Meyer[3]
[1]ICIEE, Dept. of Civil Engineering, Technical University of Denmark
[2]Graduate School of Engineering, University of Tokyo
[3]ICIEE, Dept. of Mech. Engineering, Technical University of Denmark.

A workstation with a desk-mounted Personalized Ventilation (PV) unit, with circular diffuser (d = 0.185 m) supplying air from the front/above towards the face of a thermal manikin with realistic body shape and temperature distribution was set in a climate chamber (4.70 m x 1.62 m x 2.6 m). The distance between manikin's face and the diffuser was 0.4 m. Mixing overhead ventilation at 15 L/s was used to ventilate the chamber. The room air temperature was kept at 20 oC. The PV air was supplied isothermally at 4, 6 or 8 L/s. The thermal manikin was sitting 0.1 m away from the front edge of the table. Passive method for control over the airflow characteristics at the breathing zone to increase the amount of clean air in inhalation consisted of a rectangular board (0.63 m x 0.36 m) placed below the table and pressed against the abdominal. It acted as a barrier reducing the convection flow upcoming from the lower body. The resultant velocity field at the breathing zone was measured with Particle Image Velocimetry: a dual cavity laser (λ = 532 nm) and two CCD cameras with 35 and 60 mm lenses. Seeding consisting of glycerol droplets (d = 2-3 μm) was added to the total volume supply. The blocking of the convection layer by the board decreased twice the absolute mean velocity at the mouth: from 0.2 m/s to 0.1 m/s. This made it possible for the PV flow already at 4 L/s to penetrate the free convection flow, which without the board was achieved at the PV flow rate of 6 L/s.

Paper No 127

PRINCIPLES TO SELECT NEW TEST REFERENCE YEAR ASSESSING THE ANNUAL ENERGY FOR HEATING AND COOLING OF BUILDINGS IN FINLAND

Targo Kalamees [1, 3], Kirsti Jylhä [2], Hanna Tietäväinen [2], Juha Jokisalo [1], Simo Ilomets [3], Kimmo Ruosteenoja [2], Reijo Hyvönen [2], Seppo Saku [2], Asko Hutila [2]
[1] Aalto University, Department of Energy Technology.
[2] Finnish Meteorological Institute.
[3] Tallinn University of Technology, Chair of Building Physics and Architecture.

The indoor climate and energy calculation and simulation results depend heavily on the input parameters, of which one is boundary condition. The energy performance of buildings should be calculated on the basis of a standardized methodology, with the same outdoor climate. In this study, the long term climate data have been analysed and principles to select the new test reference year assessing the annual energy for heating and cooling of buildings have been worked out.

The new test reference year is based on modified SFS-EN ISO 15927-4 method. The primary selection of is made on the basis of dry-bulb temperature and global solar radiation. The water vapour pressure and wind speed is used for secondary selection. Based on heating and cooling energy consumption of example buildings weighting factors are calculated for climate parameters. To guarantee the maximum frequency and sequence of the test reference year, it contains months from a number of different years. With this modified method the new test reference year assessing the annual energy for heating and cooling has been worked out for Finnish buildings. The method is possible to adapt also for other countries in cold climate.

Paper No 152

COMPARISON OF THE USE TRICKLE VENTILATORS IN EUROPEAN RESIDENTIAL VENTILATION STANDARDS

Jelle Laverge[1], Arnold Janssens[1]
[1]Faculty of Engineering and Architecture, Ghent University, Gent, Belgium.

The performance of exhaust ventilations systems is governed by several factors. The specifications used for the trickle ventilators, is one aspect that will determine the total airflow in the building. Therefore, correct sizing of the trickle ventilators is an important aspect in the assessment of the quality of a ventilation system. However, little agreement is found in European ventilation standards on the specifications for these components.

We modelled the performance of an exhaust ventilation system in a standardized detached dwelling according to the Belgian, Dutch and French residential ventilation standards in order to assess the heat loss and indoor air quality associated with the different specifications they represent. In this paper we report the results of the simulations and assess the benefits and drawbacks of each of the approaches taken in the discussed standards.

Technical session 11 Case studies

Paper No 215

FUNGAL GROWTH PREDICTION IN BATHROOM BY COUPLED SIMULATION WITH CFD AND HYGROTHERMAL TRANSFER

Kazuhide Ito[1] and Nguyen Lu Phuong[2]
[1,2] IGSES, Kyushu University, Fukuoka, Japan.

Large numbers of studies have shown an association between the health risk of fungal contamination and residence in houses. Fungal growth is strongly related to the indoor physical, chemical and biological conditions, and therefore the development of comprehensive prediction method must be needed for healthy indoor environmental design. This research focuses on the fungal growth problems in residensial bathroom where the temperature and relative humidity becomes relatively high, and reports the procedure and the results of coupled numerical simlation of fungal growth incorporated with flow field and temerature/ humidity distribution in bathroom space. In this numerical procedure for fungal growth prediction, fungal spores transportation are analyzed by Lagrangean approach and non-uniform distribution of fungal spore deposition on wall surfaces in bathroom is predictied. For fungal growth phenomenon, reaction-diffusion model that reproduces morphological colony formation of fungi is adopted. By continuously executing these numerical analyses, fungal spore deposition and subsequent fungal growth on wall surface was demonstrated for targeting various indoor conditions in usual bathroom condition.

Paper No 148

EXPERIMENTAL STUDY ON THE INDOOR THERMAL ENVIRONMENT AND ON THE VERTICAL TEMPERATURE DISTRIBUTION WITHIN A HIGHLY GLAZED ATRIUM IN THE ITALIAN CLIMATE

Giacomo Villi[1], Michele De Carli[1]
[1]Dipartimento di Fisica Tecnica, Università degli Studi di Padova, Padova, Italy.

The term "atrium" is used to refer to a sheltered volume within a building. Glazed atria have become popular in the last years because of their attractive features. These atria are built in the form of large glazed spaces that allow occupants to experience distinguishing positive aspects such as daylight, natural appearance and vegetation. Even if atria benefits are relevant during winter season as such structures are meant to take advantage of solar radiation to provide natural heat and light, during summer atria are often reported to be responsible of thermal discomfort and overheating problems due to the solar penetration surplus.

This paper presents the results from a long term monitoring focused on the glazed atrium of the "Dell'Angelo" hospital in Mestre, Italy. Recently built, is one among the most significant examples of glazed structure in Italy. Results are intended to describe the thermal characteristics of the investigated space under typical Italian summer conditions in terms of the resulting thermal conditions people and patients are exposed to.

Paper No 312

OPPORTUNITIES FOR IMPROVING ENVIRONMENTAL PERFORMANCE OF RETROFITTED LIGHTWEIGHT OFFICES IN TEMPERATE CLIMATES.

Narguess Khatami[1, 2,a], Malcolm J Cook[2,b], Nick Hudleston[1, c]
[1] SE Controls, Lichfield, UK
[2] Department of Civil and Building Engineering, Loughborough University, Leicestershire, UK

Much of the existing building stock typically found on business parks and industrial estates in the UK are poorly designed in terms of energy efficiency, occupant comfort and indoor air quality. This paper reports work undertaken to investigate the performance of a typical retrofitted lightweight office building in the UK in order to identify opportunities for improving its environmental performance and energy efficiency. The case study building selected is a retrofitted lightweight two-storey commercial building located in Lichfield near Birmingham in the UK midlands which is considered representative of this type of building.

To assess actual performance of the case study building, quantitative and qualitative field measurements were conducted. Air temperature, CO concentrations and relative humidity have been measured in all the main zones of the building. Furthermore, to assess the building behaviour in more detail, pressure testing and energy meter readings were conducted. Subjective responses to the environmental conditions were investigated by questionnaire.

This paper provides an overview of the actual building performance and comparison of the quantitative and qualitative studies. The current performance of the case study building reveals that there are great opportunities to take advantage of natural ventilation and intelligent control strategies to improve indoor air quality and thermal comfort while reducing energy consumption.

Paper No 20

SOLAR COOLING: A CASE STUDY

Stefano Grignaffini[1], Marco Romagna[1], Antonio Cefalo[1]
[1]"Sapienza" University of Rome, Department of Astronautic, Electric and Energetic Engineering (D.I.A.E.E.) - Italy.

Throughout the years various methods for heat prevention and indoor temperatures control in the summer have been used. The alternative cooling strategies are based on various passive and low energy cooling technologies for protection of the buildings via design measures or special components to moderate the thermal gains, or to reject the excess heat to the ambient environment. All these techniques aim to reduce summer cooling loads and electricity demand for air conditioning. During the summer the demand for electricity increases because of the extensive use of heating ventilation air conditioning (HVAC) systems, which increase the peak electric load, causing major problems in the electric supply. The energy shortage is worse during 'dry' years because of the inability of the hydroelectric power stations to function and cover part of the peak load.

The use of solar energy to drive cooling cycles for space conditioning of most buildings is an attractive concept, since the cooling load coincides generally with solar energy availability and therefore cooling requirements of a building are roughly in phase with the solar incidence. Solar cooling systems have the advantage of using absolutely harmless working fluids such as water, or solutions of certain salts. They are energy efficient and environmentally safe.

The purpose of this paper is to describe a Solar Cooling System to be installed on the roof of a building in Rome, the headquarters of the State Monopoly. The medium size power plant is composed of the following components:
- Solar Collectors;
- Thermal Storage Tank;
- Absorption Chiller;

The plant design is based on a dynamic simulation in TRNSYS, a dynamic simulation tool used by engineers all over the world to make energy calculations in a transient state.

Paper No 37

DEFINITION OF BUILDING TYPOLOGIES FOR ENERGY INVESTIGATIONS ON RESIDENTIAL SECTOR BY TABULA IEE-PROJECT: APPLICATION TO ITALIAN CASE STUDIES

Ilaria Ballarini[1], Stefano Paolo Corgnati[1], Vincenzo Corrado[1], Novella Talà[1]
[1]Politecnico di Torino, Department of Energetics, TEBE Research Group

The Building Typologies are a set of model buildings with their own age of construction, geometrical, thermo-physical, equipment and energy performance properties. Their definition is a fundamental step addressed to different goals:

- identify the building types with the poorest energy performance;
- estimate the energy saving potentials of different refurbishment strategies;
- simulate and monitor the effect of specific policies and measures;
- develop analyses for energy advice, portfolio assessment or energy saving potential (at local, national or European level).

Within this scenario, the IEE-Project TABULA is aimed to create a harmonized structure for European Building Typologies, focusing on residential buildings: the topic of the research is how to collect, elaborate and analyze data characterizing national building stock in order to define "typical" buildings able to express a Building Typology.

In fact, different strategies with different level of information details can be adopted for "typical" building definition.

In this paper, the different approaches for defining the "building typologies" are presented and tested in TABULA project are firstly introduced. In particular, three methods are explained to show the developed benchmark models: the first method identifies building types based on several assumptions deduced by an expert without statistical data; the second method processes empirical data to pick out real buildings that are representative of the stock; finally, the third method provides a building that is the most probable of a group of buildings.

Then, these approaches are applied to some Italian case studies: example building characteristics, statistical analysis on residential building database, Italian building typologies structures are shown.

Moreover, critical aspects faced in the project and potentialities/limitations of the performed analyses are critically discussed.

Paper No 80

CASE STUDY OF HYBRID VENTILATION SYSTEM WITH EXHAUST SHAFTS

Stephen Ray[1], Leon R Glicksman[1], Masa Fukuda[2]
[1]Massachusetts Institute of Technology, Cambridge, Massachusetts, USA. [2]Hulic Company Ltd., Tokyo, Japan

Hybrid ventilation systems have been previously shown to reduce cooling and ventilation energy consumption, however they often require large atria and do not sufficiently attenuate ambient noise, especially in urban areas. This paper presents a case study of a new ten story 7,780 m² office building in downtown Tokyo with a hybrid ventilation system that uses only 1.7% of the building footprint for ventilation shafts.

Labrotory experiments are used to determine the acoustic performance of the facade, computational fluid dynamics simulations investigate exhaust airflow conditions, and an airflow network tool, CoolVent, is used to predict zone temperatures and flow rates.

The sound attenuation system is found to reduce ambient noise up to 37 dB at a pressure loss of 3.1 Pa at the desired flow rate of 2.63 m³/s per floor. A separator is proposed for the exhaust chamber that reduces the associated pressure loss 55% from 7.5 to 3.4 Pa. Pure natural ventilation is predicted to provide comfortable conditions within the space for 760 of a possible 1400 working hours between April and October. That time can be extended to 1040 hours if low-power fans that operate

with less than 5% of the energy required for mechanical cooling are used to leverage the low airflow resistance of the building.

This case study makes hybrid ventilation systems more accessible to urban office buildings by illustrating a suitable low footprint design.

Technical session 12 Evaluation, control or measurements of indoor air quality

Paper No 34

INDOOR AIR AS A COLLECTIVE FACTOR OF PRODUCTION

Arne Jönsson
Secondary Vocational Training, Härnösand, Sweden

The theory for a collective factor of production is applied to indoor air in an office. A collective factor is the same to all. In an office all workers breathe air of the same quality. The theory is adapted to measurements of dissatisfied and to the marginal cost for outdoor air.

The loss of production is expressed as a function of both the number of disturbed (dissatisfied) and of a loss of production for every worker. A worker has a loss if the concentration is higher than his threshold value or his sensitivity. This makes the loss sensible to new odours since they change both the number of disturbed and the loss per disturbed.

The theory is tested against measurements of production and it explains why the average loss per worker with body odour at the economic optimum concentration is only a promille and why a more irritating odour increases the loss to a percent of the production. The theory also explains why it is better to reduce the source strength of pollutants than to increase the outdoor air rate. To reduce the loss of production from 1,82 SEK/h pers to 0,17 SEK/h pers with more outdoor air an increase from 15 to 91 l/s pers is necessary. This costs an additional 2840 SEK/yr pers at the construction of the building. To reduce the source strength costs about 1000 SEK/yr pers. It is less expensive to reduce the source strength. (1 USD = 7 SEK)

Paper No 280

ODOUR DISTRIBUTION STUDIES AND INDOOR AIR QUALITY

B. Müller[1], M. Danielak[1], O. Zeidler[1], D. Müller[2]
[1]Hermann-Rietschel-Institut, Technische Universität Berlin, Germany
[2]RWTH Aachen, E.ON Energieforschungszentrum, Lehrstuhl für Gebäude- und Raumklimatechnik, Germany

A wide variety of impurities in indoor air influence the health and comfort of the occupants. Among the sources for these impurities, building products are of great significance, as they are not easy to remove and the occupants often have no say in the choice of building materials in apartments, offices and public buildings.

One important aspect for good perceived air quality is the distribution of emissions in the rooms. Up to now, little has been published on this subject. It is important to clarify the influence of indoor air flow and the configuration of the ducts on the distribution and removal of emissions – in particular odorous substances – from building materials, and whether air speed is a critical factor in the emission of odorous substances.

By means of the project described here, it should be possible to measure the distribution of odours (contaminants) of different densities from various sources (more or less volatile compounds, warm and cold sources). This information will then be used to derive a method for predicting the distribution of odours (contaminants), which would bring us one step closer to being able to predict the intensity of odours in indoor environments. The ability to predict odour emissions and their distribution would have a significant influence on the energy demand of buildings. The outside air volume flow could be adjusted according to need.

Technical session 12, Theme: Evaluation, control or measurements of indoor air quality, Room R3, Tuesday 2011-06-21, 09:00 to 10:30

Paper No 196

DO RAIN EVENTS HAVE INFLUENCE ON SECONDARY ORGANIC AEROSOLS GENERATED BY OZONE INITIATED INDOOR CHEMISTRY?

Moshood Olawale Fadeyi[1,2], Kwok Wai Tham[3]
[1] Faculty of Engineering and IT, British University in Dubai, P.O. Box 345015, Dubai, UAE
[2]Welsh School of Architecture, Cardiff University, United Kingdom
[3] Department of Building, School of Design and Environment, National University of Singapore, SDE 1, 4 Architecture Drive, Singapore 117566, Singapore

There have been several studies documenting the different factors that could influence ozone initiated chemistry processes and products but never has someone done a study on the effects of rain events on these processes and products. A condition very common in the tropical rainforest environment, especially in South East Asia where it rains often and whenever it rains, it rains 'heavily'. Thus, it is essential to understand possible effects of this common occurrence (rain events) on indoor chemistry.

Findings presented in this paper were part of the observations emanating from a large study conducted in a field environmental chamber (FEC) to understand the effects of filters, ventilation and recirculation rates on ozone initiated chemistry products in air conditioned (recirculating large fraction of its used air) buildings in the tropics. Out of the 12 experiments which were conducted, it rained on six occasions. Whenever it rains, it was observed that rain events dramatically affected pollutant levels, promoting desorption of total volatile organic compounds 'TVOC' (indication for limonene, deliberately injected into the chamber) which subsequently reacted with ozone producing additional SOA.

Paper No 221

THE INFLUENCE OF WIND ON THE INFILTRATION RATES IN A WEBBASED MONITORED OFFICE BUILDING

Manuel Gameiro da Silva[1,2], José J. Costa[2], Adélio Gaspar[2], António Paulino[2], Miguel Bento[2], Gustavo Botte[2]
[1]ADAI-LAETA, Department of Mechanical Engineering, University of Coimbra, Portugal
[2]WSBP Electronics, Lda, Coimbra, Portugal.

The present work reports the use of a flexible solution developed for building monitoring based on a wireless network of sensors with a web-accessed database to analyze the influence of wind velocity on the infiltration rate in an office building. The central unit of the network is a "router" working as the hub of all information received from the various transmission nodes.

The case study was conducted in a modern office building, located at the metropolitan area of Lisbon, where the monitoring system is installed. Indoor environmental quality parameters (thermal environment and indoor air quality), electrical energy data and outdoor weather conditions have been monitored. The post-processing of the collected data gives very meaningful information about the metabolism of the building and the operation of the installed HVAC systems. It is possible to clearly identify the occupancy routines. Time series of CO2 concentrations allow also the calculation of building air infiltration and HVAC air exchange rates. The night decays of the indoor CO2 concentration were analyzed together with the wind velocity and direction data acquired by a weather station installed on the building in order to determine the infiltration rates of different compartments. It was confirmed a relationship where infiltration rate rises with the square of wind velocity.

Technical session 12, Theme: Evaluation, control or measurements of indoor air quality, Room R3, Tuesday 2011-06-21, 09:00 to 10:30

Paper No 83

CONTRIBUTIONS FOR THE EVALUATION OF INDOOR ENVIRONMENTAL QUALITY IN PORTUGUESE SCHOOL BUILDINGS

Maria Dias[1], João Ramos[1,2] and Moisés Egido[3]
[1]INESC Coimbra - Computers and Systems Engineering Institute of Coimbra, Portugal
[2]Polytechnic Institute of Leiria), Portugal
[3]University of Salamanca, Spain

The sector of buildings is, on a global scale, one of the largest energy consumers (together with transport and industry sectors), becoming essential to ensure a higher energetic and environmental efficiency, thermal comfort and health conditions.

Portugal was one of the first to input Indoor Air Quality (IAQ) in the certification system. The present work reports the main results obtained, which are analysed in view of the present IAQ Portuguese national regulation.

This paper is essentially a study to evaluate the thermal comfort and the indoor air quality conditions in different higher school buildings, ventilated by natural and mechanical systems, with the aim also to compare indoor environmental performance of buildings located in a temperate climate region in Portugal.

Measurements were carried out in different buildings to evaluate global and local thermal comfort and indoor air quality parameters.

The IAQ and thermal conditions in higher schools has been found to be poor because of the high density of students. In particular, the indoor environmental quality is a significant issue for these buildings in order to be healthy and comfortable for learning performance of students.

Paper No 227

NUMERICAL ANALYSIS OF PARTICLE TRANSPORT IN A VERTICAL VENTILATOR BY CFD WITH LAGRANGIAN METHOD

Nguyen Lu Phuong[1] and Kazuhide Ito[2]
[1,2]IGSES, Kyushu University, Fukuoka, Japan.

In order to prevent contamination of particulate matter in indoor environments, the control of the transmission of particles from outdoors to indoors through ventilators is important. Toward this end, the overarching objective of this study is to reduce the amount of entrained particles through ventilation system, Computational Fluid Dynamic (CFD) technique and Lagrangian particle tracking approach are used to predict the properties of flow fields and particle transportation from outdoor to indoor environments in a vertical ventilation duct under various boundary conditions.

In this paper, we demonstrated the results of sensitivity analysis that changes particle diameters, particle densities, temperature gradients, and the results were confirmed to have relevance within the range of the validity of the given boundary conditions. Through the numerical predictions, the particle motions in ventilators were confirmed to depend on particle sizes, particle densities and air stream condition. The majority of the large particles with particle diameter (Dp) from 80 to 100 μm were strongly affected by gravitational settling and transported along different path of air stream and the particles size (Dp) in range from 10 μm to 50 μm tended to follow a streamline of carrier air flow and transported toward the exhaust opening located upper part of the duct. The numerical results showed that the removal efficiency or retaining fraction of vertical duct increased as particle diameter and particles density increased. The wall boundary condition play an important role in enhanced the retaining fraction in this study.

Technical session 13 New technologies for heat recovery system and Modelling and visualization

Paper No 137

ENERGY AND COMFORT MODEL FOR AUTOMOBILE INTERIOR SPACES

Björn Flieger[1], Rita Streblow[1], Dirk Müller[1], Arne Jachens[2]
[1]Institute for Energy Efficient Buildings and Indoor Climate,E.ON Energy Research Center, RWTH Aachen University, Germany
[2]Center of Competence Climate Controls, Continental Automotive GmbH

With alternative drive systems such as electric and hybrid cars, air conditioning in automobile cabins requires the efficient use of available energy while at the same time satisfying comfort criteria. Using a model of a car cabin allows us to draw conclusions regarding air circulation and the resulting energy demand to evaluate the efficiency of new climate concepts and thermal comfort under the given boundary conditions.

Paper No 91

A MODEL OF DECENTRALIZED AIR HANDLING TERMINALS (DAHT) INTERACTING WITH BUILDING INFILTRATION

Gabrielle Masy[1], Jean Lebrun[2], Luc Prieels[3], Benoît Vincent[1], Nicolas Hansen[2]
[1]Haute Ecole de la Province de Liege, Liege, Belgium.
[2]JCJ Energetics, Liege, Belgium.
[3]Greencom Development SCRL, Liege, Belgium

As building insulation level is increasing, ventilation becomes a more important issue in order to improve buildings energy performances. A building ventilation model can be built on a set of resistances and generators. The ventilation model complexity can be adapted to the building under study (residential, commercial). It can handle natural air movement as well as fan powered air flows. In particular, double flow ventilation systems with heat recovery can be coupled with building envelope infiltration and exfiltration air flows. The ventilation model provides good estimations of the indoor air quality level, of the thermal comfort when free-cooling is performed, and of the building energy balance.

In order to improve ventilation control and to get rid of air duct networks, double flow ventilation can be handled through Decentralized Air Handling Terminals, integrated in walls or window ledges. Thanks to ventilation models, Decentralized air handling terminals can be compared to natural ventilation systems and hybrid ventilation systems including exhaust fans. Results regarding energy consumptions and air quality levels can be analysed.

A model of DAHT can be combined with the model of a whole building envelope, including infiltrations as well as dynamic behaviour. Various hypothesis regarding occupancy patterns can be considered. A special attention can be brought to the analysis of the risk of occurrence of an air flow from wet rooms to dry rooms.

Technical session 13, Theme: New technologies for heat recovery system and Modelling and visualization, Room R3, Tuesday 2011-06-21, 09:00 to 10:30

Paper No 208

EFFECTS OF NON-UNIFORM CHANNELS ON THE PERFORMANCE OF A RUN-AROUND MEMBRANE ENERGY EXCHANGER (RAMEE)

Howard B Hemingson, Carey J Simonson, Robert W Besant
University of Saskatchewan, Saskatoon, Canada.

The run-around membrane energy exchanger (RAMEE) is a new type of energy recovery system for building ventilation air that allows heat and moisture to be transferred between isolated supply and exhaust air streams. Liquid-to-air membrane energy exchangers (LAMEE) are placed in the supply and exhaust air ducts and transfer heat and moisture between the air and a liquid desiccant that are separated by a vapour permeable membrane. The desiccant is circulated between the two LAMEEs which results in a run-around energy exchanger system. The ability of the system to transfer heat and moisture between isolated supply and exhaust ducts makes it available for numerous HVAC applications (e.g. hospitals and building energy retrofits).

The LAMEEs can be constructed as a flat plate exchanger with the membranes separating the air and solution. However, the flexible membrane deforms into the air channel due to the pressure difference between the air and solution. These membrane deformations cause non-uniform exchanger channels which results in maldistributed fluid flows and variable convective transfer coefficients in the exchanger. The variable flow rates and convective transfer coefficients are numerically shown to bring about a decrease in the performance of the RAMEE. This reduction in performance increases as the size of channel variations increase. These numerical findings are partly supported by experimental results.

Paper No 258

DEVELOPMENT OF A NEW LOCAL VENTILATION SYSTEM WITH HEAT RECOVERY

Cleide Aparecida Silva[1], Jules Hannay[1], Jean Lebrun[1] , Luc Prieels[2]
[1]JCJ Energetics, Liège Belgium,
[2]Acte, Liège, Belgium

The paper is presenting the development of a local ventilation device with supply and exhaust fans and heat exchanger. This system is proposed as an alternative to central balanced mechanical ventilation. Focus is given here on the global performances expected from such system in different configurations and in different conditions of use. A detailed comparison is made between various options, combining so-called “passive”, “active” and “hybrid” techniques: recovery heat exchanger, adiabatic humidification and refrigeration cycles. The comparison is established by global simulation of the system supposed-to-be installed in different rooms of a dwelling.

Technical session 13, Theme: New technologies for heat recovery system and Modelling and visualization, Room R3, Tuesday 2011-06-21, 09:00 to 10:30

Paper No 23

SOME OBSERVATIONS OF INTERACTION BETWEEN THE AMBIENT AND AN AXISYMMETRIC JET IMPINGING ON A SURFACE

[1]Taghi Karimi-panah, [1]Mats Sandberg
[1]Academy of Technology and Environment, Faculty of Engineering and Sustainable Development, University of Gävle, Sweden

The properties of impinging jets are used for ventilation of rooms. Therefore to understand its properties are of vital importance for designing well-functioning ventilation. An impinging jet consists of three regions, free jet, impinging zone and finally a wall jet. Due to its complexity it is a benchmark case both for measurements and prediction with CFD. The free jet region interacts with its environment by entrainment of surrounding air. Because of the influence of the surface the jet approaches the entrainment ceases at some point above the surface.

The nozzle generating the jet was mounted with and without a ceiling. Smoke visualizations were carried out to show the entrainment into the jet. The flow force of the jet was determined by placing a digital balance under an impingement plate (surface). We assume that the reaction force is equal to the flow force.

The results show some interesting differences compared with existing jet theories. There was a clear effect of how the jet nozzle was mounted, with or without attached ceiling. Furthermore for some cases the interaction between the ambient and the jet caused the jet to rotate. This rotation was probably caused by unsymmetrical entrainment of ambient air into the jet.

Paper No 48

IMPLEMENTATION, VALIDATION & APPLICATION OF HVAC SIMULATION FUNCTIONALITY FOR OPEN SOURCE CFD PACKAGE OPENFOAM®.

Thomas Schumacher[1], Dr. Eugene de Villiers[2], Dr. Andrew P. Jackson[2]
[1]Engys UG, Am Güterbahnhof 4, 18055 Rostock, Germany.
[2]Engys Ltd., 65 Dalmeny Avenue, London, SW16 4RR, United Kingdom.

This paper gives a short overview of the open source Computational Fluid Dynamics (CFD) software OPENFOAM® and the code enhancements implemented and validated that make the code a suitable alternative to existing commercial tools for HVAC (Heating, Ventilation, Air Conditioning) applications. Among those enhancements are an improved internal radiation model, a solar radiation module, support for humidity modelling, functionality to assess human comfort by parameters like PMV, PPD, DR, etc.

Technical session 14 Ventilation strategies for large rooms in historic build. and Modeling and visualization

Paper No 266

MEASUREMENT OF THE ADVENTITIOUS LEAKAGE OF CHURCHES WITH A NOVEL PULSE TECHNIQUE

E W Cooper[1], D W Etheridge[1], M Mattsson[2], H Wigö[2]
[1]Department of Architecture and Built Environment, University of Nottingham, UK.
[2]Faculty of Engineering and Sustainable Development, University of Gävle, Sweden.

The University of Gavle is currently involved in a project on saving energy in historic buildings (churches). An important factor in the determination of the natural ventilation rate is the adventitious leakage of the envelope. Measurement of leakage is therefore a key feature of the investigations. It was decided to adopt a new technique developed at the University of Nottingham (UNott). It is a pulse technique compared to the conventional steady technique.

The conventional technique consists of generating a steady and high pressure difference (50 Pa) across the envelope by means of a fan. Such pressures are rarely encountered in ventilation and this leads to errors in the low-pressure leakage. Furthermore the use of the conventional blower door technique in churches is difficult due to their large volume and the need to replace the doors.

The underlying principle of the UNott technique is described and examples of results are given. The most important advantage of the Unott technique is that the leakage is determined at the low pressure differences that are encountered with ventilation e.g. 4 Pa. This is made possible primarily by the fact that the effects of wind and buoyancy at the time of the test are eliminated by taking account of the pressure variation before and after the pulse.

For measurements in large buildings, a number of identical piston/cylinder units have to be operated simultaneously. The University of Gävle has developed a system whereby up to seven units can be used. Such a number is required for a leaky church and this is the first time this has been done.

Paper No 218

CAPTURE HOOD EFFICIENCY ABOVE INDUSTRIAL HEAT AND POLLUTANT SOURCES

Jean-Raymond Fontaine[1], René Devienne[2], François-Xavier Keller[1], Céline Bourdel 1,2
[1]INRS, Process Engineering Department, rue du Morvan, CS 60027, Vandoeuvre-lès-Nancy, France
[2]LEMTA, CNRS UMR 7563, 2, avenue de la Forêt de Haye, BP 160 Vandoeuvre-lès-Nancy, France

Numerous industrial processes use or generate hot sources. The release of heat is generally accompanied by a dispersion of pollutants in the work area and can, in the longer term, cause occupational diseases among the employees. Characterization of the plumes of thermal sources formed by natural convection is necessary for sizing ventilation installations (fumehood or ventilation by air displacement) intended to clean work environments. An experimental method allowing a description of the plume developed above a two dimensional thermal source is presented. A passive tracer, namely helium, was injected at hot surface level, thereby simulating the existence and the transportation of a pollutant. A capture hood was positioned above the source and its efficiency was experimentally evaluated by using the tracer gas method.

The temperature, velocity and pollutant concentration distributions in a horizontal cross section of the plume, measured by a set of thermocouples, thermoanemometric probes or concentration sensors distributed in a matrix, were modeled by an elliptical Gaussian mathematical approximation. The method is illustrated for the characterization of the mass, dynamic and thermal plumes generated by an adjustable rectangular source.

In a second part of the paper the efficiency variations of a capture hood placed above a rectangular two dimensional heat source are experimentally evaluated when the following parameters are varied: source temperature, height and position of the hood with respect to the source, extraction flow rate of the hood.

Paper No 235

TRACER GAS TECHNIQUES FOR QUANTIFYING THE AIR CHANGE RATE IN CHURCHES – FIELD INVESTIGATION EXPERIENCES

Magnus Mattsson, Svante Lindström, Elisabet Linden, Mats Sandberg
University of Gävle, Academy of Technology and Environment, Sweden

Two different tracer gas techniques for quantifying the air change rate were tested in three naturally ventilated churches. The techniques were the decay method (or tracer gas dilution method) and a passive tracer gas method. It appeared that the room air in the studied churches tended to be fairly well mixed when the churches are heated, presumably due to strong natural convection air currents occurring at heat sources and cooler outer building surfaces. This seems to entail that both the decay and the passive method are fairly easy to apply during times of heating. It then doesn't seem to matter much were the tracer gas is injected or where it is sampled. During non-heating periods, however, spatial differences in tracer gas concentrations were observed, making tracer gas measurements more difficult to perform.

Paper No 274

WIND TUNNEL MEASUREMENTS OF PRESSURE DISTRIBUTION ON THE FAÇADE AND SURROUNDING GROUND OF A CHURCH.

Mats Sandberg[1], Magnus Mattsson[1],David Etheridge[2], Leif Claesson[1]
[1]University of Gävle, Sweden
[2]The University of Nottingham, UK

Elderly churches have a unique shape with their high towers and long naves. There seems to be few if any reported measurement of pressure distribution on churches. Churches are naturally ventilated buildings and therefore when the wind speed is high the wind becomes an important driving force for ventilation.

A model in scale 1: 200 was built of a 19th century Swedish church provided with a crawl space.The pressure on the façade of the model was recorded in 42 points. With the aim of studying the ventilation of the church, dedicated measuring points were located on windows, doors and in the positions corresponding to the location of the openings in the crawl space.

Some field trials were undertaken with the scope of measuring the time history of the static pressure on the façade in some positions corresponding to measuring points on the wind tunnel model. Examples of these measurements are reported in the paper.

With the aim of measuring the "region of influence" on the ground caused by the church, also the static pressure on the ground was recorded in the wind tunnel tests. The static pressure on ground was recorded with a pressure plate provided with 400 pressure taps arranged in a quadratic pattern.

Paper No 157

EXPERIMENTAL SCALED-DOWN MODELLING OF INDOOR AIRFLOWS

Max Körner[1], Christian Resagk[1], André Thess[1]
[1]Ilmenau University of Technology, Ilmenau, Germany.

We report first results that were done with a novel experimental facility with the purpose to scale down turbulent mixed convection in terms of indoor airflows. The method we use is based on the modification of the thermo-physical material properties by increasing the pressure of the working gas.

The usage of dry air or sulfur hexafluoride (SF6) as the working gas provides a range of scale factors of 1:1 to 1:19 at relatively low pressures of p = 1.0bar to p = 10.0bar. The facility consists of a pressure vessel and a working gas supply. Within the pressure vessel, we investigate the velocity field of the flow inside a rectangular model room. We visualize the steady-state large-scale flow structures at three different scale factors (1:3.3; 1:6.9; 1:10) with a laser light sheet system from the outside of the pressure vessel. The found flow structures show a completely different behaviour at the three different scale factors.

Paper No 163

EFFECT OF GEOMETRY ON FLOW FIELD AND TEMPERATURE DISTRIBUTION IN LARGE INDOOR ENVIRONMENTS

Samuele Testa[1], Michele De Carli[1], Bjarne W. Olesen[2], Reiner Heimsch[3]
[1]University of Padua, Department of Applied Science, Padua, Italy.
[2]International Center for Indoor Environment and Energy, DTU, Lingby, Denmark.
[3]IBH Ingenieurburo Rainer Heimsch VDI/AGOF, Rastede, Germany.

The present paper has the aim to perform a numerical research activity on the effects which might affect the determination of air flow distribution, thermal stratification and the thermal comfort, when trying to achieve the preservation of cultural property in buildings, such as historical churches, when retrofitting and heating the building. Many times this problem has to be faced by designers and the question is to achieve accurate results, maintaining limited time in the definition of the grid and the resolution of the problem.

One real case is here considered: St. Marien church in Wismar, a city on the Baltic Sea. Computational Fluid Dynamics (CFD) has been used in the simulations.

Different geometrical models are developed to investigate the effect of the ceiling shape on the flow patterns and temperature fields.

Technical session 15 Modeling and visualization and Case studies

Paper No 108

ZERO IMPACT SCHOOLS AND VENTILATION: INTEGRAL DESIGN APPROACH NEEDED

Wim Zeiler, Gert Boxem
Technische Universiteit Eindhoven, Netherlands

As complexity of the design increases with the efforts to reach the limit of zero impact, it really becomes necessary to involve all building design disciplines in the earliest stage of the design process. The study aims to propose a method to support architects within the conceptual phase of a building design process of a Net-Zero Impact Building. The focus of the paper will be on the ventilation aspects during the design process of a net zero energy school.

Paper No 298

THERMAL COMFORT AND ENERGY ANALYSIS FOR BUILDING BASEMENT

Ahmed C. Megri[1], Yao Yu[2], Qing Zhang[3], Jay Puckett[4] and Tony Denzer[5]
[1,2,3,4,5]Civil and Architectural Engineering Department, University of Wyoming, USA

Heat losses from slab-on-grade floor and basement are a significant part of the load for residential family homes. Several materials are used to insulate slab-on-grade floors and basements of buildings. Numerical and optimization studies were carried out using an integrated model based on "2D heat transfer" equation and steady state "zonal model" for objective to quantify the best combination of heating system and insulation configuration used for basements. A comprehensive computational analysis has been performed to predict a building's heat loss through its basement floor and walls. Thermal comfort index PMV has been predicted to investigate the thermal comfort situation at the basement location. As well, different heating systems have been investigated to determine the most appropriate system.

Paper No 152

MODELING OF EVAPORATION FROM A HEATED WATER SURFACE

Carla Cotas, António Raimundo, Adélio Gaspar and Divo Quintela
ADAI, Department of Mechanical Engineering, Faculty of Science and Technology of the University of Coimbra, Coimbra, Portugal.

This work is dedicated to the study of water evaporation in typical situations such as the case of the free surface of indoor swimming pools. The experimental results were obtained in an experimental installation consisting of a low speed wind tunnel and an evaporation tank. The numerical study is performed through a CFD numerical code, developed by the authors, that is able to simulate nonisothermal and turbulent three-dimensional flows, with mass and heat transfer. To validate the implemented formulation the numerical results were compared with the experimental measurements for the mass transfer at a free surface of a hot water tank and correlations from the literature. The best approximations obtained between the numerical results and the literature correlations occur for inlet velocities of 0.6 $m \cdot s^{-1}$.The relative errors vary between 0.37 and 36.64 %, with a mean value of 8.43 % (mean of all relative errors), which means that the implemented formulation reasonably predicts the physical phenomena involved in evaporation.

Paper No 184

COMPUTATIONAL SIMULATION OF INDOOR TEMPERATURE ON FIBRE CEMENT ROOFS: COMPARATIVE STUDY THROUGH THE APPLICATION OF DIFFERENT ROOF COLORS

Patricia Regina C Drach
Federal University of Rio de Janeiro, Rio de Janeiro, Brazil.

The objective of this work is to study ways to minimize heat exchange through the fibro-cement roofs of popular houses in the city of Rio de Janeiro, Brazil. This city is located in a hot climate region, in south-eastern Brazil, where the need to keep the heat out is fundamental to the well being of its users.

Another important fact is that, because of their low income, these people end up choosing materials less favourable to the local climate. Despite the emergence of new technologies and materials better adapted to local climate, the population continues to use fibre-cement roofs, since they use smaller amounts of wood than traditional roofs, and yet are of easier placement. In a previous work, this housing module was tested using roofs that are very common in Rio de Janeiro city: a cement roof and a fibre-cement one. The experiment with the cement painted white generated the lowest levels of internal temperature, for the same conditions of temperature and ventilation. This work was selected to cover further use of fibre-cement, while testing test the changes in the indoor temperature caused by the fibre-cement painted white.

The thermal changes were checked including solar gain factors. The analysis started by solving the air-circulation problem to determine the wind fields, using a stabilized mixed finite element method, Petrov-Galerkin like, applied to the full Navier-Stokes equations. The thermal problem is analyzed through the SUPG - Streamline Up-wind stabilized Petrov-Galerkin finite element method.

Paper No 64

INVESTIGATION ON AN AIR-TO-AIR HEAT RECOVERY EXCHANGER: MODELLING AND EXPERIMENTAL VALIDATION IN DRY CONDITIONS

Gendebien Samuel[1], Bertagnolio Stephane[1], Georges Bernard[1] and Lemort Vincent[1]
[1]Thermodynamics Laboratory, University of Liège, Belgium

This paper deals with the development and the experimental validation of an air-to-air recovery heat exchanger model dedicated to ventilation of residential buildings. The first part of the paper presents a semi-empirical model based on physical characteristics of the heat exchanger. The aim of this model is to predict the behavior of the device in dry conditions. The second part of the paper describes the experimental facility designed to determine the hydraulic and thermal performance of the device and offers a comparison between simulation and experimental results. Finally, the model is calibrated in order to predict the thermal and hydraulic performance of the heat exchanger within respectively, less than 5% and less than 2% of accuracy.

Paper No 139

CALCULATION OF RADIATIVE HEAT EXCHANGES WITHIN INDOOR SPACES WITH COMMON OBSTACLES

Sara Francisco, António M. Raimundo, Adélio R. Gaspar and Divo A. Quintela
ADAI, Department of Mechanical Engineering, University of Coimbra, Pólo II, 3030 - 788 Coimbra, Portugal

The energy exchanges that occur between the human body and the surrounding environment are particularly affected by heat and mass transfer phenomena. This paper presents a method to evaluate the radiative heat exchanges between the human body and the environment based on the calculation of view factors for realistic human body shapes and complex configurations of the enclosures. An algorithm that calculates the view factors has been developed based on the Stokes' Theorem.

Originally, the numerical evaluation of the contour integral consists of dividing the contours of the surfaces into a finite number of line elements. However, this formulation is not sufficiently detailed to be applied in the evaluation of real indoor domains, namely for those situations where the obstacles are partially blinding the radiation between surfaces. The authors propose a formulation where the original surfaces are divided into a grid of elementary areas. The Stokes' Theorem is then applied to the determination of the view factors between these elementary areas. With this approach, the account of the shading effect of obstructions is significantly improved.

The case study of a common individual office room ($2.86 \times 4.70 \times 2.96$ m^3) with a person, a desk, a chair, a computer and a cooling radiant system is presented and discussed. Different configurations of radiant panels have been compared. Since the view factor is directly related to the amount of radiative heat exchanges, when a radiant cooling system is used, higher radiative exchanges will occur in the situation where the panel is closest to the human body.

Paper No 61

TOWARDS A MORE RELIABLE MODELLING OF NIGHT-TIME VENTILATION WITH BUILDING ENERGY SIMULATION MODELS

Sarah Leenknegt[1], Rolf Wagemakers[2], Walter Bosschaerts[2], Dirk Saelens[1]
[1]Building Physics Section, K.U.Leuven, Leuven, Belgium.
[2]Department of Mechanics, Royal Military Academy, Brussels, Belgium.

Night ventilation is a passive method to cool the exposed thermal mass in a room during night. The efficiency depends heavily on surface convection. Traditionally in Building Energy Simulation (BES) models, this flux is represented by one surface and reference temperature and one convective heat transfer coefficient (CHTC). In reality, the flux will vary both locally and in time, which can only be modelled with CFD. The research presented here focuses on two elements: (1) the convection regimes during the increased ventilation period, i.e. local and transient behaviour, and (2) the capability to model this behaviour in BES. To investigate this, a simple room is analyzed through a transient CFD-simulation in 2D. The CFD model includes the thermal mass in ceiling and floor, with both vertical walls considered as adiabatic and an ACH of 7 h^{-1} at 16 °C.

It is shown that within the period of free cooling, the transient behaviour is limited and buoyancy remains dominant in this case. This causes the incoming air to fall down, rather than adhere to the ceiling, which results in a four times higher CHTC at the floor compared to the ceiling. A comparison is made between the local CHTC-values and CHTC-correlations from literature. Appropriate correlations are available, but the selection algorithm in BES poses problems: each surface should be evaluated individually, rather than be assigned based on a global room convection classification. TRNSYS is capable of modelling the surface and air temperature evolution fairly well, given that the corrected CHTC's are used, rather than the internal calculation.

Technical session 16 Evaluation, control or measurements of indoor thermal environment

Paper No 132

ROOM AIRFLOW EFFECTS APPLYING UNSTEADY BOUNDARY CONDITIONS

Claudia Kandzia, Martin Schmidt, Dirk Müller
Institute for Energy Efficient Buildings and Indoor Climate,
E.ON Energy Research Center, RWTH Aachen University, Germany.

Thermal comfort and the ventilation effectiveness are important aims in the construction of modern supply systems. Both targets depend on the distribution of local air temperatures and velocities in a room. The transient behaviour of room air structures has been investigated only marginally until today.

A new experiment ought to provide new data of the transient behaviour of large scale flow structures in ventilated rooms. The test facility is 3 m high, 4 m wide and 5 m long. All walls are made of aluminium to minimize radiation heat transfer effects.

The air supply is located at the ceiling over the whole length of the test facility. Special slot diffusers are used as inlets in order to provide well-defined wall jets. The outlet air leaves the room at the bottom zone. With the aid of volume flow controllers in front of each inlet it is possible to vary the supply volume flow between steady and transient. In the transient case each volume flow controller has a sine wave as control signal. Hence the flow rate of the supply air varies temporally.

The room airflow structures are examined experimentally for isothermal and non-isothermal boundary conditions. In the non-isothermal case the thermal loads are realized by electrically heated cuboids. The velocity field in the room is measured with the aid of omni-directional anemometers. The temperature and velocity sensors are positioned automatically in the testing room with a traverse system.

The comparison of the measurements with numerical calculations shows similar results.

Paper No 300

THERMAL COMFORT – SIMULATION AND MEASUREMENT USING A THERMAL MANIKIN

Conrad Voelker, Oliver Kornadt
Bauhaus-University Weimar

An approach is introduced, which enables the assessment of thermal comfort considering the complex and inhomogeneous climatic conditions in buildings as well as the human physiology. Computational fluid dynamic is linked with a numerical model representing the thermophysiological behavior of the human body (UC Berkeley Comfort Model). By dint of CFD, the climatic conditions in buildings are simulated with a detailed resolution. The simulations have been validated against measurements in a climate chamber equipped with a thermal manikin. Basing on the CFD simulations, the thermophysiological model is finally able to determine the temperature distribution of the human body, the heat flux to the environment as well as thermal comfort.

Paper No 220

DYNAMICS OF HUMAN SKIN TEMPERATURES IN INTERACTION WITH DIFFERENT INDOOR CONDITIONS

Ehab Foda, Kai Sirén
Aalto University, Department of Energy Technology, PO Box 14100 Espoo-Finland

Human skin temperature represents an essential physiological variable in the evaluation of the human thermal sensation and comfort. The dynamics of local skin temperatures reflect the human thermoregulatory responses subject to environmental conditions. The variability in human responses challenges the research efforts for precise modelling of thermoregulation mechanisms; therefore, measured data always reveal useful inputs in this field. This paper reports on a laboratory-based measurement of local skin temperatures in different indoor conditions and exposure durations. The measurements aimed at investigating the human body response due to a step change and checking the variability in different duration tests with the same subjects as well as maintaining a physiological steady state at the different conditions. Eleven human subjects (males) participated in a total of 46 tests, in a controlled environment, including 33 tests in 1-hour duration, 12 tests in 2-hours duration and 1 test in 4- hours duration. The age of the subjects ranged from 27 to 40 years old and body weight from 62 to 105 kg. They wore normal office clothing (0.6clo, clo=0.155m²°C/W) and had normal sedentary office activities with own portable computers. The measurements of local skin temperatures and heart beat rate of subjects along with online measurements of the indoor conditions were carried out and recorded at short time intervals. The paper shows, in results, the variations of local skin temperatures in transients of different exposures and discusses the observations from the measurements. Main findings were related to the variability in response at body segments in different exposures and the skin temperatures at the physiological steady state conditions.

Paper No 209

NUMERICAL AND EXPERIMENTAL MODELING OF AIRFLOW AND HEAT TRANSFER OF A HUMAN BODY

Cristiana Croitoru[1] , Ilinca Nastase[1] , Alexandru Iatan[1] , Vlad Iordache[1] , Amina Meslem[2], Iolanda Colda[1]
[1] Technical University of Civil Engineering in Bucharest, Romania
[2] LEPTIAB, University of La Rochelle, France

This study is a part of larger experimental and numerical campaign intended to evaluate the influence of the turbulence intensity at the inlet of the terminal air diffusion devices on the local draft sensation and thermal discomfort of ventilation users. In this paper we present results of CFD simulations using a realistic model of human body along with an experimental validation. The recorded velocity, turbulence and temperature fields allowed us to estimate the distributions of DR, PPD and PMV indexes. While the DR around the human body seems to be less sensible to the value of the turbulence intensity imposed at the exit plane of a mixing ventilation jet, the PMV display some differences especially in the region around the ankles of the human body. In the same time the PPD index displays the existence of a particular range of initial turbulence intensities where the global comfort conditions in the room are worse than for lower or higher values.

Paper No 229

TRADE-OFF RELATION BETWEEN ENERGY CONSUMPTION AND COMFORT LEVEL ACCORDING TO THE FINNISH-2008 ADAPTIVE THERMAL COMFORT CRITERIA

M. Hamdy, A. Hasan, K. Siren
Aalto University Scholl of Science and Technology, Department of Energy Technology

In 2008, the Finnish Society of Indoor Air Quality (FiSIAQ) has introduced new criteria for the thermal comfort classes (S1, S2, and S3). The criteria stipulate adaptive model based on 24 hour-average outdoor air temperature. The maintenance of particular comfort level is linked to energy consumption and consequent energy cost. The current study uses simulation-based optimization approach to find the trade-off relation between the energy consumption and the thermal comfort level. Two fully mechanically air-conditioned single offices (north and south) are taken as representative zones in one floor of an office building. The following design parameters are addressed using 24 variables: supply ventilation air temperature profile, night ventilation control strategy, room cooling equipment size, night set-back temperature, window and shading types. The optimal ranges of the studied design variables are founded and analyzed. The analysis shows that 12 design variables have a marked influence on the results. However the others have less impact. Compared with a reference design, the simulation-based optimization approach allows 20 % energy saving and 34 % reduction in the cooling equipment power-size. It is concluded that the addressed thermal comfort criteria is restricted and could increase the energy consumption. Compromise solutions can be achieved by using a suitable simulation-based optimization approach.

Paper No 92

BUILDING ENERGY SIMULATION WITH CONSIDERING AIR TEMPERATURE DISTRIBUTION BASED ON CFD

Weirong Zhang[1], Shinsuke Kato[2], Kyosuke Hiyama[2], Yoshihiro Ishida[2]
[1]The University of Tokyo, Tokyo, Japan.
[2] Institute of Industrial Science, The University of Tokyo, Tokyo, Japan.

Network model is used in most of the building energy simulation, which uses the space-averaged air temperature. Recently the task-ambient and the personal air-conditioning system are widely introduced as they positively utilize the air current distribution and temperature distribution to meet the demand on both comfort and energy conservation. In this case, a higher cumulative error in building energy simulation may occur if the air temperature distribution is not considered. The purpose of this research is to propose an index named contribution ratio of indoor climate (CRI), obtained from the CFD results, by which the energy simulation with considering temperature distribution can be achieved.

Indoor thermal environment is influenced by various heat factors. The index of CRI is developed to estimate the individual contribution of these factors to temperature distribution. It is assumed that a temperature field in a room can be treated as a linear system, which means a temperature field can be seen as the superimposition of several sub-temperature fields, each of which was caused by a single heat source. The CRI is defined as the ratio of temperature rise (or decrease) at position x calculated by CFD and the heat released from a heat source. Then by coupling CRI with the energy simulation tool, such as TRNSYS or EnergyPlus, a more accurate energy simulation for non-uniform indoor thermal environment could be expected. In this study, CFD simulation coupled convection and radiation was performed on an atrium with full measurement results. The distribution of CRI for each individual heat source was calculated based on a fixed represent airflow field. An approximation method on heat transfer analysis and the approach of integrating network model and CRI is introduced.

Technical session 17 New technologies for heating and cooling or ventilation/AC

Paper No 244

HEMISPHERE LOBED NOZZLE JET ANALYSIS FOR MIXING AND PERSONALIZED VENTILATION

Ilinca Nastase[1], Amina Meslem[2], Mouhammad El Hassan[2], Arsen Melikov[3]
[1] Technical University of Construction in Bucharest, Roumania
[2] LEPTIAB, University of La Rochelle, France
[3] ICIEE, Technical University of Denmark

Passive control of jet flows for mixing and entrainment enhancement is of wide applicative interest. Our purpose is to develop new air diffusers for HVAC systems, by using lobed geometry nozzles, in order to ameliorate the users' thermal comfort. This paper presents some results obtained during a first part of a larger campaign dedicated to the experimental and numerical analysis of the advantages of introducing lobed hemispherical nozzles as innovative air diffusers. In this study we propose a comparison between the lobed jet generated by an orifice plate, one of the most advantageous studied lobed tubular nozzle and a jet flow from a lobed hemispherical orifice. This is the first time at our knowledge when the flow generated by such a hemispherical diffuser is presented and analyzed. It was shown that this innovative concept of lobed hemisphere jet which can be easily integrated in air diffusion devices is very efficient regarding induction of the surrounding air. A very interesting observation is also resulting: introducing a curvature on the surface where a lobed orifice is placed completely changes the behaviour of the generated jet flow which seems to obey to the same mechanisms as in the case of a three-dimensional lobed tubular nozzle.

Paper No 276

AIRFLOW CHARACTERISTICS AT THE BREATHING ZONE OF A SEATED PERSON: ACTIVE CONTROL OVER THE INTERACTION OF THE FREE CONVECTION FLOW AND LOCALLY APPLIED AIRFLOW FROM FRONT FOR PERSONALIZED VENTILATION APPLICATION

Arsen Melikov[1], Zhecho Bolashikov[1], Hideaki Nagano[2], Clara Velte[3], Knud Erik Meyer[3]
[1]ICIEE, Dept. of Civil Engineering, Technical University of Denmark
[2]Graduate School of Engineering University of Tokyo
[3]ICIEE, Dept. of Mech. Engineering, Technical University of Denmark.

A method for active control over the interaction between the free convection flow around occupant's body and locally applied airflow from front on the velocity field at the breathing zone of a seated person was studied. A workplace equipped with personalised ventilation (PV) generating flow from front/above against the face of a thermal manikin with realistic body shape and surface temperature distribution (used to resemble a seated human body) was set in a climate chamber (4.70 m x 1.62 m x 2.60 m). The air temperature in the chamber was kept at 20°C. Ceiling diffuser supplied ventilation air at 15 l/s. The PV air was supplied isothermally at 4, 6 or 8 L/s. The PV diffuser with diameter 0.18 m, was located at distance 0.4 m from the face of the manikin. The distance between the lower chest of the manikin and the front edge of the desk was 0.1 m. Box with 6 small computer fans (suction box) was installed below the table board, above the thighs of the manikin, and was used to exhaust the air of the free convection flow coming from the lower body parts of the manikin. The velocity field at the breathing zone was measured with Particle Image Velocimetry consisting of a dual cavity laser and two CCD cameras. The maximum absolute mean velocity measured in the convective layer at the mouth of the manikin was 0.20 m/s and was reduced to 0.09 m/s when the suction box was used. Thus the weakend boundary layer can be penetrated by the PV flow at the lowered velocity. The use of the suction box and the PV at 4 L/s resulted in the same velocity at the breathing zone as when

only PV was used at 6 L/s. The maximum absolute mean velocity measured at the breathing zone with the control and the PV at 8 L/s was 0.35 m/s.

Paper No 62

IMPROVING INDOOR AIR DISTRIBUTION EFFICIENCY BY MICROTURBULENT PULSED AIR FLOW

Ludmila Balandina, Viatcheslav Shkarpet
"Arktos Ltd.", Saint-Petersburg, Russia.

Objectives: Creation in indoor environment of microturbulent pulsed air flow, enabled without compromising comfort to abstract more heat than in rooms with static comfort indoor environment. This «dynamic indoor environment» allows to keep comfort at higher temperature during warm season. Eventually it results in energy economy. For example, temperature increase of 1 degree allows to reduce demand for cooling by 20%, moreover in certain terms to reduce air consumption.

Methods: The experiments were conducted in the laboratory through aerodynamic and acoustic tests. Furthermore, the structure of air flow and its main characteristics were explored through computer modeling. We used CFD code «COOLIT» based on numerical solution of Navier-Stokes equations.

Results: The conducted work enabled us to design and optimize the construction of a new diffuser, which develops auto-oscillations. Consequently, direction, velocity and temperature of air flow vary with the frequency 5-20 Hz without mechanical forces. Developed device has wide band air supply jet, high rate of damping and turbulence level of supply air flow, low level of produced noise with high sound absorption, as well as has an attractive design. Serial products were developed and introduced into particular projects.

Conclusions: For the first time models and samples were tested full-scale using aerodynamic and acoustic equipment and numerical research by CFD code «COOLIT». The work succeeded and therefore production samples of different sizes for the offices were developed and patented.

Paper No 151

RADIANT HEATING/COOLING AND VENTILATION: CONVECTIVE HEAT TRANSFER ENHANCEMENT

Francesco Causone[1], Stefano P. Corgnati[1], Bjarne W. Olesen[2]
[1]TEBE Research Group, Department of Energetics, Politecnico di Torino, Torino, Italy.
[2] ICIEE, Department of Civil Engineering, DTU, Lyngby (Copenhagen), Denmark

Radiant heating/cooling combined with different mechanical ventilation strategies shows an increasing application in new and existing buildings, because minimum ventilation rates for air renewal and suitable level of indoor air quality are mandatory for commercial and industrial buildings and necessary in new airtight residential buildings. This "hybrid" system may provide significant energy saving, nevertheless when a ventilation system operates together with a radiant system, reciprocal interactions should be investigated, mainly because the air motion has an effect on the heating/cooling capacity of the radiant surface and because the heated/cooled surface has an impact on the air velocity and distribution in the room. Further interactions may occur, influencing indoor air quality and thermal comfort.

Some algorithms for the calculation of the enhanced convective heat transfer coefficient of a heated/cooled surface are given in the literature, for the case of mixing and displacement ventilation.

On the basis of these algorithms the possible enhancement of the convective heat transfer and the heating/cooling capacity of radiant floors and ceilings are calculated, using data from experimental measurements. The highest improvements are shown for ceiling heating and floor cooling, because, under natural convection, these systems have typically very low convective heat transfer coefficients. The activation of a forced convection results therefore in a considerable enhancement of their

heating/cooling capacity, even with low air velocity or low air changes per hour. Improvements are nonetheless also reported for ceiling cooling and moderately for floor heating.

Paper No 126

A STUDY OF AN ENERGY EFFICIENT BUILDING VENTILATION SYSTEM

Rajnish Kaur Calay, Wen Chung Wang
School of Engineering & Technology, University of Hertfordshire
Hatfield, Herts. AL10 9AB, UK

Energy consumption in buildings in developed countries is up to 40% percent of the total energy consumption. Among building services, energy use for the building heating, ventilation and air conditioning (HVAC) systems is over 50% of total energy consumption in the building sector in many developed countries. Thus about half of nation's carbon emissions are associated with energy use in buildings when fossil fuels are used. Therefore, energy efficiency in buildings is critical for realising CO2 targets at local and global level. There are many heating/ cooling concepts that rely upon renewable energy sources and use the natural low temperature heat sources in the winter and heat sinks in the summer.

This paper is aimed to develop a high performance rotary heat exchanger (RHE) cooling/heating ventilation system using reverse cycle heat pump (RCHP). The coefficient of performance (COP) of the reverse cycle heat pump, the energy consumption of the heating/ cooling process and suitable climatic conditioning for reverse cycle heat pump cases, which operate better with low-energy are estimated. Waste energy from the exhaust air stream is used to precondition the outdoor before it is supplied into the building. Thus the proposed system provides free heating in the winter and free cooling in the summer compared to conventional systems. Its performance is better than the typical reheat or air-conditioning system to provide the same indoor air quality (IAQ) levels. Also it is shown that the energy saving up to 60% (heat energy) is achieved by using the proposed hybrid system in building ventilation applications.

Paper No 272

AIRFLOW CHARACTERISTICS AT THE BREATHING ZONE OF A SEATED PERSON: INTERACTION OF THE FREE CONVECTION FLOW AND AN ASSISTING LOCALLY SUPPLIED FLOW FROM BELOW FOR PERSONALIZED VENTILATION APPLICATION

Zhecho Bolashikov[1], Arsen Melikov[1], Clara Velte[2], Knud Erik Meyer[2]
[1]ICIEE, Dept. of Civil Engineering, Technical University of Denmark
[2]ICIEE, Dept. of Mech. Engineering, Technical University of Denmark.

A workstation with Personalized Ventilation (PV) unit and a thermal manikin with realistic body and temperature distribution were set in a test room (4.70 m x 1.62 m x 2.6 m). Airflow at 15 L/s was supplied from a ceiling diffuser to ventilate the room and keep the temperature at 26 oC. The PV consisted of two plenum boxes nested in each other and placed below the desk top, with discharge slots 0.06 m x 0.5 m (W x L). The PV unit was pressed against the abdomen of the thermal manikin. Each box had a separate supply fan. The airflow supplied isothermally and upwards from the inner and outer box was the same: 4, 6, 8 and 10 L/s. The mean velocity field at the breathing zone was obtained by Particle Image Velocimetry: a dual cavity laser (λ = 532 nm) and two CCD cameras with 35 and 60 mm lenses. Seeding, glycerol droplets, was added to the total volume supply. The maximum absolute mean velocity measured near the mouth was 0.1 m/s, when the boxes were installed but not working. When the two slots supplied equal amount of air, the measured absolute mean velocity increased with increasing the supplied air with a maximum of 0.35 m/s at 10 L/s.

Technical session 18 Natural, hybrid and mechanical ventilation

Paper No 195

THE ACTRESS CONCEPT. A NEW FAÇADE MODULE FOR LOW ENERGY BUILDINGS.

Fabio Favoino[1], Francesco Goia[1,2], Marco Perino[1], Valentina Serra[1]
[1]TEBE Research Group, Department of Energetics, Politecnico di Torino, Italy.
[2] The Research Centre on Zero Emission Buildings, Department of Architectural Design, History and Technology, Norwegian University of Science and Technology, Trondheim, Norway.

In the field of Zero Emission/Energy Buildings, the research focused on new building envelope concepts is of crucial relevance. The future building skin is required to be responsive and to actively transfer and store heat, light and air. The integration of ventilation strategies within the façade components plays an important role in providing dynamicity and responsiveness.

During the last few years, innovative building envelope solutions – Advanced Integrated Façades (AIFs) – have been developed and investigated. In the frame of an Italian National Research Project (PRIN 2007), a new façade concept, called ACTRESS, has been conceived. A Multifunctional Façade Module (MFM) has been designed to play different roles through its ability to change the thermophysical behaviour in order to suit different building requirements and to face different boundary conditions. The module consists of an opaque part, which has hybrid ventilation and incorporates PV modules, VIP and PCM and of a transparent part, where high performance glazing technologies are used. A mock-up of the ACTRESS module has been realized and test cell measurements are ongoing. The paper illustrates the concept, technologies and functional strategies – with a focus on the ventilation – of the module. Results obtained from a numerical analysis of the ACTRESS façade module are here presented and discussed.

Paper No 245

PRESSURE LOSS COEFFICIENT OF ROOM CONSIDERING INFLOW DIRECTION IN PREDICTION OF CROSS-VENTILATION RATE THROUGH LARGE OPENINGS

Hisashi Kotani[1], Toshio Yamanaka[1]
[1]Osaka University, Suita Osaka, Japan

The use of general discharge coefficient values of an opening in predicting the cross-ventilation rate through large openings is not suitable, because the discharge coefficient changes by preservation of the dynamic pressure and the influence of flow direction at the inlet opening. The laboratory teststo measure the pressure loss coefficient of the room were conducted and the influence of the opening size, room size and the inflow direction was investigated. The pressure loss coefficients changed by the inflow direction but the possibility to arrange them by the non-dimensional length of opening size and room depth. The wind tunnel tests were also conducted to measure the cross-ventilation rate and the calculated flow rates using the measured pressure loss coefficient were compared with the measured ones. The calculated flow rates were well agreed with them.

Paper No 160

PREVENTING OVERHEATING IN A NATURALLY VENTILATED NARROW-CAVITY DOUBLE-SKIN FACADE

T. Chenvidyakarn1[23], N. Mingotti1[2], A. W. Woods[2]
[1] Department of Architecture, University of Cambridge, UK.
[2] BP Institute for Multiphase Flow, Cambridge, UK.
[3]Faculty of Architecture and Planning, Thammasat University, Thailand.

This paper explores the prevention of overheating in the naturally ventilated, narrow-cavity, double-skin facade. A generic facade configuration is used as a basis for our study, in which the facade is connected to an occupied room and extends a storey above the room. The base of the facade is connected to the room through an opening at a high level in the room. The room is connected to the exterior environment through a low-level opening analogous to a doorway. The facade is connected to the exterior environment through an opening at its top. The room contains a distributed heat source analogous to distributed occupants and electrical equipment in an open-plan office. The facade cavity contains a vertical heat source analogous to a combination of glass and a blind/louvres heated by solar radiation. These two heat sources combined drive natural ventilation from the room to the facade. Using average climatic data, we show that, in warmer and colder climates alike, strong solar radiation on the facade or a large heat load in the room or a combination thereof can lead to overheating in the facade, which can be detrimental to the life of facade components. We then use a combination of laboratory experimentation and theoretical analysis to show how facade overheating can be prevented by adding an opening at the base of the facade which connects its cavity to the exterior environment. This additional opening will lead to multiple unidirectional flow regimes: external air will either be drawn into or vented from the base of the facade. Theoretical analysis shows that the flow regime in which exterior air is drawn into the facade leads to a lower temperature in the facade, and therefore is generally more desirable for preventing facade overheating. The flow direction through the base of the facade is controlled by the relative strength of the heat loads in the room and on the facade, the relative height of the room and the facade, and the relative size of the openings in the room and at the top of the facade. The size of the opening at the base of the facade does not control the direction of the flow. We discuss the principles of how to control these variables to prevent facade overheating in different climates, using simplified examples.

Paper No 66

WIND TUNNEL STUDY OF WIND INFLUENCE ON AIRFLOWS IN INDUSTRIAL VENTILATION SYSTEMS

Nicolas Le Roux[1,2,3], Xavier Faure[1], Christian Inard[2],
Sandrine Soares[3] and Laurent Ricciardi[3]
[1] Centre Scientifique et Technique du Bâtiment (CSTB), Nantes, France.
[2] Laboratoire d'Etude des Phénomènes de Transfert et de l'Instantanéité : Agro-industrie et Bâtiment (LEPTIAB), La Rochelle, France.
[3] Institut de Radioprotection et de Sûreté Nucléaire (IRSN), Saclay, France.

For safety assessment in nuclear industry, the French Institute for Radiological Protection and Nuclear Safety (IRSN) has developed a computational code named SYLVIA in order to simulate airflows in industrial ventilation systems. In this code, the mechanical ventilation is considered as the main driving flow, whereas other parameters like wind, internal overpressure or buoyancy may arise and thwart the mechanical ventilation effect. At present, the modelling of wind effects in safety computational codes has not been sufficiently validated. Hence, the purpose of this work is to carry out wind tunnel experiments in order to validate the integration of wind effects in the computational code SYLVIA. First, a methodology has been developed in order to study reduced-scale ventilation systems subjected to isothermal steady or transient flows. Then, a reduced-scale model has been established by applying the methodology to one real industrial configuration. Several wind tunnel

experiments have been carried out on this reduced-scale model, for different wind velocities, wind directions and with the mechanical ventilation on or off. These tests highlight the emergence of reversal airflows, particularly for external leaks. Experimental results have been then compared with numerical results of code SYLVIA in order to check the accuracy of the code to take into account these phenomena.

Paper No 291

MEASUREMENTS OF COOLING PERFORMANCE FOR AN EARTH-TO-AIR HEAT EXCHANGER WITH AN AXIAL ENTRANCE FAN

Per O. Tjelflaat[1], Jian Zhang[2] and Fariborz Haghighat[3]
[1]Norwegian University of Science and Technology, Trondheim, Norway.
[2]Pacific Northwest National Laboratory, Richland, WA, USA.
[3]Concordia University, Montreal, Canada.

The present work focuses on passive cooling of ventilation air in an underground culvert having an axial fan at the entrance. At warm summer conditions the cooling effect of the ground culvert has been assessed by analyses of measurement data during a test period. Measurements from other investigations on the same building have also been collected in order to investigate the cooling effect of the ground duct. Other experimental investigations of convective heat transfer coefficient have also been collected.

Measurements show that air temperatures in rooms are kept below 25°C when outdoor maximum is around 30°C when the underground duct is used. This result is considered satisfying. The air cooling effect is achieved as a result of convection to the culvert walls combined with thermal accumulation in concrete walls and conduction into the ground. There are clear indications that the heat transfer coefficient is substantially augmented by running of the axial fan located at the inlet of the culvert.

Paper No 172

BUOYANCY-DRIVEN FLOW BETWEEN TWO SERIES-CONNECTED CHAMBERS IN AN AMBIENT ENVIRONMENT

Lin, Yi-Jiun Peter[1], Huang, Jiun-Jay[1]
[1]Department of Mechanical Engineering, National Taiwan University of Science and Technology, Taipei, Taiwan.

The purpose of this research is to study ventilation patterns in two series-connected chambers which have a common interior divider. The space of two chambers is connected to the outside through several different arrangements of ventilation openings and the shared divider only has one connection opening. A buoyancy source is placed in the forced chamber which is connected to another chamber without a buoyancy source. The research proposes a theoretical approach to analyze two series connection chambers which are connected to the ambient environment. The study shows two main different types of displacement ventilation in the indoor space, as pull and push types of displacement ventilation, according to an additional resistance force due to the unforced chamber. Salt water and clean water are used to simulate the buoyancy force difference in experiments. A salt water source nozzle is placed at the top of the water tank. Salt plume proceeds in the same direction as the gravity, but it is opposite to the thermal plume proceeding direction. In this paper, except for in the part of theoretical analysis, we present and discuss the research results according to the experimental orientation. However, the coordinate system in both parts of theoretical analysis and experimental results is consistent. In the experiments, the pull-type takes place when the shared divider has a highlevel opening, through which ambient fluid flows into the forced chamber, and dense fluid discharges into the ambient environment through a low-level opening on the side wall of the forced chamber. The push-type takes place when the shared divider has a low-level opening and the side wall of the forced chamber has a high-level opening. The dense fluid in the forced chamber discharges into

the unforced chamber through the low-level opening on the shared divider and is pushed into the ambient environment through the opening on the side wall of the unforced chamber. This research presents theoretical analysis and laboratory experimental results of two different types of ventilation and shows reasonable agreement between them.

Technical session 19 Modelling and visualization

Paper No 143

INVESTIGATION OF AIRFLOW EFFECTS IN INDUCTION BEAMS

H. Freitag, D. Müller
Institute for Energy Efficient Buildings and Indoor Climate, E.ON Energy Research Center, RWTH Aachen University, Germany

This paper investigates the influence of various Reynolds-averaged Navier-Stokes (RANS) turbulence models on the calculated internal flow field of active chilled beams. Special attention is given to the induction behavior of the internal jets by comparing the calculated global induction rate to measurement data. Further comparisons between chosen models comprise jet deformation, attachment to walls or adjacent jets as well as separation from the wall.

The internal flow of active chilled beams is characterized by high local velocity gradients and different flow regimes resulting in large variations of the Reynolds number. This has to be carefully taken into account when determining the induction ratio by CFD methods. Since common RANS models are known to excel in certain types of flow while performing worse in others, the best compromise for the numerical investigation of flow features inside an active chilled beam, namely free stream, deflected and wall-bounded flow, has to be found.

Paper No 306

EVALUATION OF A BOX METHOD-BASED TOTAL PRESSURE BOUNDARY CONDITION FOR THE SQUARE PLAQUE DIFFUSER

Mikhail Koupriyanov, Brad Tully & Julian Rimmer
E.H. Price Ltd. 638 Raleigh Street, Winnipeg, MB, Canada R2K 3Z9

Computational Fluid Dynamics (CFD) is quickly becoming a widely used analysis tool in the design of occupied spaces. The accuracy of the simulation depends largely on the accuracy of the method used to model the diffuser. This study builds upon the work previously published by the authors. Two box method type boundary conditions with either a total pressure definition or a constant velocity are used to model a common square plaque diffuser. These boundary conditions are thoroughly evaluated by studying the behavior of the jet created by the diffuser that is placed close to a wall. Comparisons are made to experimental data as well as to CFD results that use a full geometric model of the diffuser. In general, the development of the jet is under predicted by the CFD results and the temperature profiles are not predicted as accurately as the velocity profiles. The accuracy of the box method tends to degrade as the diffuser is moved closer to the wall and as the flow rate is increased. The results also show no clear advantage in using a total pressure over a constant velocity since the predictions of both are nearly identical.

Technical session 19, Theme: Modelling and visualization, Room R1, Tuesday 2011-06-21, 13:30 to 15:00

Paper No 183

DEVELOPMENT OF INDOOR ENVIRONMENTAL SIMULATION TOOL

Akira Komatsu[1], Sumio Shiochi[1], Toshihiro Suzuki[1]
[1]Research & Development Department Environmental Technology Laboratory Sakai, Osaka, JAPAN.

Indoor terminals of air conditioning system are important because they control elements which make up indoor environment such as temperature, humidity, airflow. Indoor environment changes depending on specification of indoor terminals, the layout, and the number of terminals. CFD(Computational Fluid Dynamics) has been used to simulate indoor environment. Distribution of temperature, humidity, airflow velocity, and can be predicted by CFD. To simulate indoor environment, many processes are required such as setting of boundary condition of thermal load, setting of inlet and outlet boundary of indoor terminal, generating numerical grid. It is difficult for the people engaged in a facility design to simulate indoor environment and to perform engineering of indoor terminal with CFD. In this study, indoor environmental simulation tool has been developed. By this tool it is possible for the people who are not specialists of CFD to perform simulation. This tool can create a simulation model by the wizard system, which is incorporated technical know-how of modelling and setting of simulation. Boundary condition of indoor terminal has been designed by the measurement of airflow. The models have high calculation accuracy. In the model of heat exchanger parts, temperatures and humidity of inlet air are input conditions, and supply capacity and airflow rate are parameters. Temperature and humidity of outlet air are calculated separating sensible heat and latent heat. Time required to create a simulation model has been shortened because it is easy to set parts such as furniture, indoor terminal by the wizard system.

Paper No 273

ENERGY EFFICIENCY AND INDOOR CLIMATE: MODELING OF VENTILATION SYSTEMS USING CONTAM W

Maria Justo Alonso[1], Bjarne Malvik[1], Hans Martin Mathisen[2]
[1] SINTEF, Trondheim, Norway.
[2]Norwegian University of Science and Technology, Trondheim, Norway.

In modern houses the idea is to reduce heat losses by minimizing air leakages and increasing insulation. However, very airtight houses may lead to poor Indoor Air Quality (IAQ), if not properly ventilated.

A smartly designed ventilation system is required to avoid reaching such a situation. In order to predict the effect of alternative solutions for ventilation, it was decided to use ContamW; a simulation software program calculating concentration of pollutants and ventilation requirements based on e.g. pollutant source strengths, number of occupants, their presence, and their activities.

For this study, a model apartment has been designed. Relevant sources of contamination have been implemented and individual living schedules are included. Three different models for ventilation have been simulated. The results indicate that for the actual test apartment, DCV_IAQ will be the best solution regarding both Indoor air quality and energy consumption.

Technical session 19, Theme: Modelling and visualization, Room R1, Tuesday 2011-06-21, 13:30 to 15:00

Paper No 15

VALIDATION OF THE VELOCITY PROPAGATING ZONAL MODEL – VEPZO

Victor Norrefeldt, Gunnar Grün
Department of Indoor Environment, Climatic Impact,
Fraunhofer Institute for Building Physics, Holzkirchen, Germany

A new zonal model, VEPZO (VElocity Propagating ZOnal model) has been developed which includes the air velocity as a property of a zone and a viscous loss model. The velocity information is passed from a zone to the flow models and is therefore propagated into the room. Instead of using the power law equation to model the airflow between two zones, an equation derived from the forces acting on a flow path is used.

In contrast to previous zonal models, the VEPZO model does not need special correlations for driving flows. Therefore the VEPZO model is considered more generic, as no a priori information on the airflow pattern is needed to conduct simulations. Furthermore, numerical problems like in the power law based zonal models do not arise when the pressure difference between two adjacent zones tends to zero.

This paper compares results obtained by the VEPZO model with published results obtained by other zonal models. The first comparison focuses on a thermal buoyancy induced airflow in a twodimensional room. The second one investigates an isothermal room with a ceiling mounted inlet. Results obtained by the VEPZO model are in good accordance with the compared cases.

Paper No 115

FLOW NOISE FROM AN EXHAUST VALVE – PREDICTION BY SIMULATIONS AS COMPARED WITH MEASUREMENTS

Pekka Saarinen, Hannu Koskela
Finnish Institute of Occupational Health, Turku, Finland

Regulations on ventilation noise are today very stringent and ventilation products should be able to cope with high flow rates without generating too much noise. This sets challenges for the product development, which in HVAC industry has been made through trial and error, by using tailor-made test specimens. One alternative approach could be numerical simulation of the physics behind the sound generation. This strategy has become feasible only lately, along with increased computing power of design workstations. This is because modelling of flow noise requires heavy LES simulation of the flow, preferably followed by a separate acoustic simulation for computing the sound in the far field. In this work, a test case of two slightly different exhaust valve designs is examined. Despite a minor difference in the two geometries, the difference in flow noise is remarkable, and there is no obvious reason to this. Simulations and measurements of the flow fields and sound productions in both the valve designs have been performed, to see what brings about the difference in noise generation and whether it could have been predicted by computer simulations. Results show that simulation might be a shortcut to be reckoned with in product design.

Paper No 140

MODELING OF EVAPORATION FROM A HEATED WATER SURFACE

Carla Cotas, António Raimundo, Adélio Gaspar and Divo Quintela
ADAI, Department of Mechanical Engineering, Faculty of Science and Technology of the University of Coimbra, Coimbra, Portugal.

This work is dedicated to the study of water evaporation in typical situations such as the case of the free surface of indoor swimming pools. The experimental results were obtained in an experimental installation consisting of a low speed wind tunnel and an evaporation tank. The numerical study is performed through a CFD numerical code, developed by the authors, that is able to simulate nonisothermal and turbulent three-dimensional flows, with mass and heat transfer. To validate the implemented formulation the numerical results were compared with the experimental measurements for the mass transfer at a free surface of a hot water tank and correlations from the literature. The best approximations obtained between the numerical results and the literature correlations occur for inlet velocities of 0.6 $m \cdot s^{-1}$.The relative errors vary between 0.37 and 36.64 %, with a mean value of 8.43 % (mean of all relative errors), which means that the implemented formulation reasonably predicts the physical phenomena involved in evaporation.

Technical session 20 Evaluation, control or measurements of indoor thermal environment

Paper No 75

IMPACT OF ROOM SUPPLY REGISTER LOCATION ON THE PREDICTED THERMAL COMFORT AND IAQ

Boualem Ouazia, Iain Macdonald, Alexandra Thompson, Daniel Booth
National Research Council Canada, Ottawa, Canada.

For residential forced air heating and cooling systems conventional thinking is that air supply registers should be located under exterior windows. There were good reasons for this in the past (primarily to counteract the cold downdraft from the window) but new construction standards (well-insulated walls, better glazing and air tight wall/window interface) mean that there is now less downdraft. Positioning the supply air register away from a window could have a large impact for new construction as duct lengths could be shortened (saving materials and construction time). This could also allow greater architectural freedom as the forced air system would not need to be ducted to the building perimeter. This paper presents results from experiments conducted in a new, well-constructed building to evaluate whether the supply registers can be moved away from the traditional location without impacting indoor air quality or predicted thermal comfort. The physical experiments showed that in both traditional and non-traditional placement: (1) there are minimal changes in predicted thermal comfort (a measure of thermal discomfort - vertical air temperature difference much lower than 3K), and (2) contaminant-removal effectiveness (a measure of indoor air quality) values were in the same range.

Paper No 26

A STUDY ON THE EFFECT OF THERMAL, LUMINOUS AND ACOUSTIC ENVIRONMENT ON INDOOR ENVIRONMENTAL COMFORT IN OFFICE

Li Huang, Yingxin Zhu, Qin Ouyang, Bin Cao
Department of Building Science, Tsinghua University, Beijing, China.

In light of the growing concern of productivity, much more attention is paid to indoor environment in offices in recent years. Standards address different environmental factors such as thermal comfort, indoor air quality, aural and visual environment separately. In fact, those environmental factors have several combined effects on occupants' acceptability and office work performance. ASHRAE Guideline 10P emphasized the interactions among the indoor environmental factors and mentioned that more detailed researches should be done. In this study, a simulated field survey was carried out, aiming at investigating the acceptable range of every single environmental factor. In the survey, parameters of thermal, luminous and acoustic environments were measured and subjects' satisfaction levels of environment were investigated with questionnaires. In addition, getting the classification of indoor environmental quality in which interactions among the factors was considered was also discussed in this paper by way of chart.

Paper No 162

EVALUATION OF RADIANT SYSTEMS PERFORMANCE ENHANCEMENT BY MEANS OF PRIMARY AIR

Michele De Carli[1], Massimiliano Scarpa[1], Roberta Tomasi[1], Giacomo Villi[1]
[1]Department of Applied Physics, University of Padua, Italy.

Radiant ceiling panels are largely used for summer cooling, especially in offices, but they have to be assisted by a primary air system, aimed to dehumidify and prevent condensation on the cooled surfaces. In the present paper, the interaction between the primary air inlet and the radiant ceiling is considered. In particular, the study regards how the inlet of primary air can be used to enhance the performance of the radiant ceiling, by means of the increase of convective heat transfer.

Via CFD (Computational Fluid Dynamics) simulations, the interaction of radiant ceiling panels and primary air has been analyzed in an office case study, with typical internal sensible heat gains and various ventilation conditions. Heat gains from occupants and computers are simulated by means of electrical cooling load simulators (dummies) distributed in the room.

Primary air inlet is considered both through typical vents and through induction air terminals. Different temperatures and air flows were chosen for different vents and the convective heat flow at the ceiling surface has been calculated for each configuration, in order to quantify the increase in the heat exchange.

The study showed that the exploitation of primary air allows just a modest enhancement of total heat transfer coefficients at the ceiling surface, even if induction air terminals are used, and consequently the ceiling cooling capacity is not increased.

Paper No 206

THE EFFECT OF AIR DISTRIBUTION ON THE PERFORMANCE OF RADIANT PANELS

Panu Mustakallio[1], Risto Kosonen[1], Arsen Melikov[2], Marcin Duszyk[2]
[1]Oy Halton Group Ltd., Helsinki, Finland.
[2]International Centre for Indoor Environment and Energy, DTU Civil Engineering, Technical University of Denmark, Lyngby, Denmark.

The cooling power of radiant panels can be affected by the room air distribution system. This impact can be an important because often the cooling output is the limiting factor for the design and use of radiant panels. In this study the impact of air distribution generated in a room by linear slot diffuser, radial multi-nozzle diffuser and radial swirl induction unit on the cooling power of radiant panel was compared. The impact on the thermal environment was studied as well.

Measurements were done without and with supply air in a test chamber equipped with two ceiling radiant panels and air distribution units flushing the radiant panels. The test room was constructed according EN-15116 standard to allow for accurate measurement of the cooling capacity. The cooling power of the radiant panels was increased with the studied air distribution methods. The increase was from 8% to 17% depending on the air distribution method. The difference between air temperature and operative temperature in the occupied zone was small.

The effect of air distribution on the performance of radiant panels can be taken into account during the design of radiant cooling panel systems. The cooling power of the panels depends on the type of air distribution, supply air jet velocity flushing the panels and supply air temperature.

Paper No 69

VALIDATING THE THERMAL BEHAVIOR OF AN OPEN-PLAN OFFICE IN AN INNOVATIVE LARGE SCALE COMMERCIAL BUILDING

Florian Judex[1], Renate Teppner[1]
[1]Austrian Institute of Technology, Energy Department, Sustainable Building Technologies, Giefinggasse 2, 1210 Vienna, Austria; corresponding author: renate.teppner@ait.ac.at

The 7.500 m² office building ENERGYbase in Vienna, Austria, was planned and built satisfying the "passive house" standard. The building utilizes innovative façade design as well as concrete core activation (CCA) supplied by heat pumps, solar heating and groundwater. Furthermore, the ventilation and air-conditioning (HVAC) systems are based on solar heating and solar cooling, with the heat pumps as backup. Approximately 500 sensors were installed for permanent monitoring in the whole building. A 250 m² open-plan office, directly influenced by the south-oriented façade, was chosen to assess the thermal comfort in the building with respect to planned uniformity objectives. The aim was to examine whether the indoor air temperature in the selected office can be deduced from the 20 permanent surface temperature sensors and the 4 permanent illumination sensors.

Additional 49 air temperature sensors, one glass surface temperature sensor and 19 air velocity sensors were placed within the office. Most of the sensors were freely suspended, with a few in the overflow ventilation area and at air openings. Measurements were recorded every minute for approximately four months (July through November 2009). Linear statistical models were chosen to determine a direct relation between permanent and temporary monitoring.

The collected data proved that the planned thermal homogeneity of the room air was nearly reached, showing increased sensitivity to solar radiation close to façade during the summer-winter transitional period. Consequently, the data from permanent and temporary sensors were well correlated, allowing good air temperature approximation based upon the wall sensor readings in the room.

As part of prospective weather predictive control strategies the linear models used for approximating room air temperatures via recorded wall temperatures and illumination values will be implemented to improve the current building control strategies, since they are a viable state observer for the actual room air temperature.

Paper No 303

USER HEATING BEHAVIOUR AND THERMAL COMFORT IN WINTER IN KOREAN HIGH-RISE RESIDENTIAL BUILDINGS

Sumee Park[1] , Seok-Yoon Kang[2], Young-Tag Kim[2]
[1] Fraunhofer Institute for Building Physics, Valley, Germany.
[2] Samsung C&T Corporation, Seoul, Korea

This study is a part of an investigation on the development of comprehensive alternatives for the existing high rise-residential buildings in Korea regarding the energy efficiency, thermal comfort and moisture problems in the indoor environment. As a first step in the project, a long-term monitoring of the indoor climate in 24 dwellings for one year as well as spot measurements and questionnaires in 85 dwellings were conducted for three times. In this study the user heating behaviour and thermal comfort perception are investigated. The temperature measured in Korean dwellings during spot measurements is relatively high with an average air temperature of 23.9 °C. The thermal comfort analysis shows that Koreans feel indeed thermally neutral at 24 °C operative temperature. However, they feel more comfortable at lower temperatures between 21 °C and 23 °C and in slightly cool environment with a PMV of -1. The higher air temperature in Korean buildings might result from the typical high ratio of windows in the high-rise residential buildings and the low performance of these windows. Also the high indoor air temperature in summer might influence the comfort perception of residents in winter. It was observed that the residents in the building with high air temperature in summer heat their dwellings more in winter. However, an "indoor adaptation" hypothesised in this

study could not be confirmed by means of statistical significance tests, although this tendency could be observed in the spot measurements as well as in the monitoring.

Technical session 21 New technologies for heating and cooling or ventilation/AC

Paper No 228

SUPPLYING VENTILATION AIR AT A SUB-WET BULB TEMPERATURE BY INDIRECT EVAPORATIVE COOLING

Ala Hasan
Aalto University, School of Engineering, Department of Energy Technology, Finland

The objective of this paper is to test a proposed method to supply cold ventilation air to rooms by indirect evaporative cooling at a sub-wet bulb temperature (without using a chiller). The idea lies in manipulation of the air flow inside the cooler in order to indirectly pre-cool the working air before it participates in the cooling process. In this paper, a model is developed to find the performance of the cooler based on heat and mass transfer balance. The advantage of this model compared with other ones is that it takes into account the effect of the heat conduction in the separating wall in the xdirection.

Experimental measurements are carried out to evaluate the idea experimentally. For this purpose, a one-cell two-passage counter-flow regenerative indirect evaporative cooler was built and tested. It is found that by the proposed method, indirect evaporative cooling is capable of cooling air to temperatures lower than the ambient wet bulb temperature (i.e. wet bulb effectiveness > 1). For a range of tested inlet air temperature of 27.1 to 33.7 C and relative humidity of 13 to 55%, the produced outlet air temperature was 12.9 to 24.3 C with a wet bulb effectiveness of 1.02 to 1.14. The validation of the model results indicates that model error is between −6.5% to +3.5%. The theoretical ultimate temperature for the proposed process is the dew point temperature of the ambient air.

Paper No 114

CONTAMINANT DISTRIBUTION AND THERMAL STRATIFICATION IN PERIMETER ZONES EQUIPPED WITH UFAD SYSTEM

Arash Soleimani Karimabad[1], Fariborz Haghighat[1], Ali Bahloul[2]
[1]Concordia University, Montreal, Canada.
[2]IRSST, Montreal, Canada.

Recently UnderFloor Air Distribution (UFAD) system has been considered as a method for minimizing building energy consumption while providing thermally comfortable and healthy environment.

Based on various numerical simulations and laboratory tests, improvement in Indoor Air Quality (IAQ) and potential for energy savings are claimed to be among major benefits associated with using UFAD system. Even though there are few field reports of successful experiences, interviews with professional, building owners and occupants show that the gains in “real installations” have not been sensibly achieved in all cases. Literature survey shows that improvements in IAQ have been majorly assessed for interior zones. Yet, UFAD is particularly recommended to be an efficient solution for open-plan commercial buildings. Regardless of whether an environment is conditioned by overhead or UFAD systems, the perimeter zones require separate thermal conditioning. Open-plan buildings include large area in which perimeter air-conditioning system isolate large variable loads at building envelope from comparatively low constant heat gains at interior spaces. However, there has been no study to consider the impacts from perimeter air-conditioning system and the influence of thermal decay on air temperature within supply plenum onto interior stratified space.

Results from both numerical simulation and a field measurement indicate considerable impact on contaminant and thermal stratification close to perimeter. As a result, in real cases, when stratification is not properly achieved, cross contamination and poor IAQ increase risk of exposure to chemical and biological contamination, and compromise energy saving potentials of UFAD systems.

Paper No 178

COMPARISON OF TEMPERATURE AND VELOCITY FIELD IN ROOMS WITH CHILLED BEAMS AND RADIANT PANEL SYSTEMS COMBINED WITH MIXING VENTILATION

Marcin Duszyk[1], Arsen Melikov[1], Risto Kosonen[2], Panu Mustakallio[2]
[1]International Center for Indoor Environment and Energy, Department of mechanical Engineering, Technical University of Denmark.
[2]Oy Halton Group Ltd., Helsinki, Finland.

Thermal environment generated by chilled beams (ChB) was compared with the thermal environment achieved with chilled ceiling combined with mixing ventilation with ceiling mounted air supply diffusers (RP). Large number of experiments under different conditions was performed in full scale mock-up of office room. Comprehensive database of mean velocity, air temperature, operative temperature, mean radiant temperature, draught rating and vertical temperature difference was collected. The experiment conditions represented summer and winter conditions with different heat loads created by dummy, PC, heating foil and window. The room temperature was maintained at 26 ˚C in summer conditions and at 21˚C in winter conditions.

The thermal environment in the room with regard to the vertical temperature difference and draught discomfort can be qualified as Category A for all of the studied cases. There was no significant difference in the vertical temperature gradient between the radiant panel and chilled beam systems in the heating mode and in the cooling mode. The vertical temperature difference between 1.3 m and 0.1 m above the floor was less than 1 oC in the cooling mode and less than 2 oC in the heating mode. In the winter case (heating mode) the vertical temperature difference obtained with the chilled beams was slightly higher (0.3 oC) than with the radiant panels. In the summer case (cooling mode), the maximum velocities were below 0.25 m/s with both systems. In heating mode, velocities are generally below 0.1 m/s in the occupied zone.

Paper No 286

TESTING OF INDUCTION VAV BOX WITH ONE OR TWO DAMPER BLADES

Maria Hurnik
Department of Heating, Ventilation and Dust Removal Technologies, Faculty of Environmental Engineering and Energy, Silesian University of Technology, PL-44-101 Gliwice, Poland.

The airflow in VAV systems with traditional VAV dampers and constant geometry air diffusers can be reduced only to 50-60%. If induction boxes are used instead of traditional VAV dampers the primary airflow may vary in wider range. The induction boxes induce room air and mix it with conditioned primary air. The objective was to investigate the induced secondary flow of two box types with one and two damper blades.

The CFD modelling was used to determine the characteristics of two types of VAV induction boxes. The boxes with square inlet 110×110 mm, square outlet 150×150 mm and 1300 mm of length were modelled. They differ in the number of damper blades and in the way of air induction. Commercial CFD code was used and k-ε turbulence model was applied.

The induction boxes characteristics were identified for the inlet static pressure of 150, 300 and 500 Pa as well as for the outlet airflow resistance of 30 Pa at primary air nominal airflow. Primary airflow was changing in the range of 10-110% of nominal one, which corresponds to 10 m/s inlet air velocity.

The results show that induction box with two damper blades at inlet static pressure 150-500 Pa induces about 30% more of a room air than the box with a single damper blade. Comparison of the CFD results for modelled induction boxes and catalogue data of one produced induction box showed a high potential of the induction box construction improvement.

Technical session 21, Theme: New technologies for heating and cooling or ventilation/AC, Room R3, Tuesday 2011-06-21, 13:30 to 15:00

Paper No 117

FREEZE PROTECTION METHOD IN VENTILATION SYSTEM USING TWO HYDRONIC CIRCUITS

Natasa Djuric[1], Hans Martin Mathisen[2]
[1]SINTEF, Energy Research, Energy Processes, Trondheim, Norway.
[2]Norwegian University of Science and Technology, Trondheim, Norway.

Freeze protection in ventilation systems is important to avoid freeze damages and increase in maintenance costs. Freezing in a ventilation system can appear in construction and operation phase. The aim of this study was to test a new method for freeze protection in ventilation system. This method implies use of an additional heat exchanger. The method used two hydronic circuits: the first one with water on energy supply side, and second one with mixture of glycol and water at the secondary side. The mixture transfers heat from the energy source via an additional heat exchanger to the coil in the air handling unit (AHU). AHU with freeze protection from a manufacturer was tested in the laboratory. Since it was not possible to obtain a supply air temperature of -20 oC in the laboratory conditions, a model was developed on the MATLAB\Simulink platform. Several operation scenarios were tested on the Simulink model and they included use of three types of energy sources: boiler, heat pump, and district heating with outdoor temperature compensation. Results showed that even new method with mixture of glycol and water could tolerate low air temperature, the AHU would require known sequence control for freezing protection, where the circulation pump between energy source and heat exchanger starts first. This sequence control is necessary regardless of energy source. Finally, the article gave recommendations on how to define sequence control to avoid freezing in AHU.

Paper No 197

MODELING OF THERMAL COMFORT BASED ON PHYSIOLOGICAL AND PSYCHOLOGICAL PARAMETERS

Takashi Shinozuka[1], Toshiharu Ikaga[1], Chosei Kaseda[2], Haruka Ueda[2]
[1]Department of Science and Technology, Keio University, 14-620, 3-14-1 Hiyoshi, Kohokuku, Yokohama, Kanagawa, Japan
[2]Yamatake Corporation,1-12-2 Kawana, Fujisawa, Kanagawa, Japan

Indoor climate is closely associated with the productivity and thermal comfort of office occupants. In recent years, air-conditioning systems in office buildings have been required to consume less energy while maintaining the same level of thermal comfort. Therefore, it is necessary to understand the thermal comfort of occupants and reflect it in the context of air-conditioning use. In this study, we identified the relationship between increasing indoor temperature and the thermal comfort of occupants by analyzing psychological and physiological parameters. In addition, we found that a gradual increase in indoor temperature provides a more comfortable transition than an abrupt increase.

Poster session 1

Paper No 294

PREDICTION OF WATER TEMPERATURE OF A MANTLE TANK FOR SOLAR HOT WATER SYSTEMS USING A NEW ZONAL MODEL

Ahmed Cherif Megri[1], Yao Yu[2]
[1, 2]Civil and Architectural Engineering Department, University of Wyoming, USA

A novel dynamic zonal model, POMA+, is developed to improve the design of mantle tanks for solar hot water systems. This new transient model takes into account the water heat capacity and has the ability to predict inside tank hot water temperature distribution for any time step. This model has been integrated into TRNSYS library. The results of temperature distribution of this model has been evaluated and validated using other models based on finite difference and finite elements methods. First, the model description is presented. Secondly, an investigation of a better stratification is performed to improve the performance of domestic solar hot water tanks.

Paper No 293

DEVELOPMENT AND APPLICATION OF A NEW DYNAMIC ZONAL MODEL (DZM)

Yao Yu[1], Ahmed Cherif Megri[2]
[1,2]Department of Civil and Architectural Engineering, University of Wyoming, USA

A new dynamic zonal model has been developed (DZM), based on the original Pressurized Zonal Model with Air-diffuser steady state (POMA). The objective of this new transient model is to predict room air temperature distribution over time using small time steps. Several case studies have been investigated and tested to evaluate this program. The results have been compared to conventional transient one-single zone model predictions. This new model has been integrated into TRNSYS library and coupled to the multizone thermal model. The coupled model DZM/TRNSYS has been used to predict room temperature distribution during a whole day. Thermal comfort index, such as, PMV and TSENS distributions using the Fanger Two-Node Model have been also predicted.

Paper No 288

IMPACT OF FACIALLY APPLIED AIR MOVEMENT ON THE DEVELOPMENT OF THE THERMAL PLUME ABOVE A SITTING OCCUPANT

Daria Zukowska[1], Arsen Melikov[1], Zbigniew Popiolek[2] and Joanna Spletsteser[1,2]
[1]International Centre for Indoor Environment and Energy, (www.ie.dtu.dk), Department of Civil Engineering, Technical University of Denmark, DK-2800 Denmark.
[2]Department of Heating, Ventilation and Dust Removal Technology, Faculty of Energy and Environmental Engineering, Silesian University of Technology, PL-44-101 Gliwice, Poland.

In the future the implementation of low power office equipment will make thermal plumes generated by occupants one of the dominant flows affecting the air distribution in spaces. Advanced air distribution methods, such as personalized ventilation, are expected to become widely used in practice. In this study the impact of locally applied airflow on the thermal plume generated by a sitting human body was investigated.

The experiment was performed in a climate chamber ventilated with an upward piston flow. A thermal manikin simulating an occupant was sitting on a computer chair behind a table. The air speed and temperature were measured across the plume 0.7 m above the manikin head when airflow of 10 l/s was supplied first against its face and then upward from the front edge of the desk.

The use of the flow, against the face or upward tangentially to the chest, disturbed significantly the free convection boundary layer enveloping the body and caused scattering in the

measured values of air speed and temperature excess in the plume. In comparison with the case without airflow, the integral characteristics including volume flux, momentum flux, buoyancy force density and enthalpy flux were greater when the flow was supplied upward tangentially to the chest and lower when it was supplied against the face.

Paper No 121

THE IMPORTANCE OF ACCOUNTING FOR RADIATIVE HEAT TRANSFER IN ROOM AIRFLOW SIMULATIONS

Maria-Alejandra Menchaca-Brandan, Leon R Glicksman
Massachusetts Institute of Technology, Cambridge, MA

Because modeling air flow in rooms can become a very complex task, in order to understand the effect of each element in a room (heat sources, ventilation jets), it is common practice to model the convective heat transfer from heat sources separately from the effects of radiation. Doing this greatly simplifies the task of understanding the physics of the flow above hot surfaces, and allows simulating the behavior of buoyant jets and plumes using liquid experiments (such as emptying filling boxes). However, it is often overlooked that radiation may have a first order effect on the resulting convective behavior of buoyant elements, as well as on the final temperature distribution in a room. The objective of this paper is to shed light on the critical importance of accounting for radiative heat transfer when modeling airflow and temperature distribution in rooms. We ran multiple Computational Fluid Dynamics simulations of flow in heated rooms, with and without the radiation. The effect of accounting for radiation on the temperature profile in a room was studied, as well as the influence of this stratification on the development of buoyant jets and plumes. The vertical change of temperature in rooms where radiation is accounted for is consistently linear. On the other hand, when radiation is not accounted for, the air in the room is divided into two layers of different temperature, result consistent with water experiment results. Because the development of buoyant plumes is highly dependent on thermal stratification, a plume growing in a two-layer thermally stratified room will develop in a very different manner than in a linearly stratified environment. We conclude that, while filling boxes experiments are extremely useful to understand the physics of pure convection, they may not provide an accurate description of the real flow behavior in real buildings.

Paper No 52

VENTILATION STRATEGY FOR SUBWAY CABINS USING NUMERICAL SIMULATIONS

Soon-Bark Kwon[1], Duck-Shin Park[1], Youngmin Cho[1], Jong Bum Kim[1], Taesung Kim[2]
[1]Korea Railroad Reserach Institute, Uiwang, Korea.
[2]Sungkyunkwan University, Suwon, Korea.

More than 7 million passengers use the Seoul Metropolitan Subway (SMS) everyday and this number has been increasing. With the increasing trends in concerns over indoor air quality (IAQ), the management of IAQ has become an important issue, especially in the case of subway operators, because most of subway lines are placed underground with poor ventilation conditions. The SMS is known to be the one of the most heavily used transportation means in the world. The subway cabins are naturally ventilated when the cabin doors are opened and through the gaps caused by the incomplete air-tightening of cabin bodies. Although, subway trains are equipped with a mechanical exhaust fan, the apparatus is rarely operated due to the problem of heating/cooling efficiency especially in the summer and winter seasons.

In this study, we analyzed the transient carbon dioxide (CO2) concentration level during the journey of a subway cabin when a heavy load of passengers of up to 200% of designed capacity using a computational fluid dynamics (CFD) analysis. Air streamlines were analyzed and CO2 output by passenger was related with flow patterns during the cooling of the cabin. With an increased journey time, the CO2 concentration increased by up to 5,000 ppm depending on passenger load. Through the operation of a mechanical exhaust fan, the high concentration of CO2 decreased down to 1,500 ppm.

The concentration of CO2 could be used as an index of cabin IAQ which indicates the possible coexistence with other gaseous pollutants such as volatile organic compounds (VOC) and formaldehyde (HCHO). The effect of the exhaust fan operating on the dilution of indoor air was estimated by comparison with a closed cabin. In addition, the energy consumption for cooling in summer time was assessed for exhaust fan operations.

Paper No 134

SINGLE-SIDED NATURAL VENTILATION STRATEGIES FOR HEALTHCARE BUILDINGS

Z. A. Adamu, A.D.F. Price, M.J. Cook
Loughborough University, United Kingdom.

Control of airborne pathogens, while achieving comfort and energy efficiency, places strains on typical mechanical air-conditioning systems of hospitals. These buildings expend over 40% of their energy for heating of air and spaces (DoH, 2006) while still being challenged by the problem of airborne infection. Natural ventilation remains a largely unexplored alternative which could alleviate this problem; however, achieving acceptable indoor air quality (IAQ), thermal comfort and energy efficiency from this technique is challenging and requires careful design and modelling. Many hospitals appear to use same openings (windows) as both inlets and outlets, making the air exchange process inefficient. The aim of this study is therefore, to demonstrate the feasibility of using dual opening single-sided, buoyancy-driven natural ventilation for achieving low-energy comfort and reduction of airborne pathogens in 1-bed and 4-bed hospital wards that are designed or refurbished according to the Department of Health's Activity Database (ADB). Conceptual design conditions were based on provisions of HTM-03 guidelines and openings were sized through empirical methods. The design conditions are tested using dynamic thermal simulation (DTS) to demonstrate the longterm airflow and comfort implications. Computational fluid dynamics simulations (CFD) are then used to provide an indepth steady-state prediction of the distribution and quality of air as well as pathogen dispersal, with respect to airflow patterns/directions for the selected strategies. These research findings provide insights into the airflow, and comfort performances of 1-bed and 4-bed wards for both existing and proposed healthcare buildings whose design or retrofit calls for singlesided natural ventilation.

Paper No 237

IMPACT OF CONTROL STRATEGIES ON ENERGY PERFORMANCE OF A DOUBLE-SKIN FACADE

Deuk-Woo Kim[1], Cheol-Soo Park[1], In-Han Kim[2]
[1]Department of Architectural Engineering, Sungkyunkwan University, South Korea
[2]Department of Architecture, College of Engineering, Kyung Hee University, South Korea

Nowadays, there is an increasing demand for environmental improvements and high performance in office buildings. A double-skin facade system has advantages in terms of energy savings, protection from external noise, reduction of wind load, daylighting control, and decreased asymmetric thermal discomfort, etc. However, there is too little information available on impact of different control strategies on performance in terms of energy, daylighting, thermal comfort, and ventilation. For that reason, this paper aims to investigate the performance of double-skin facades with three different control strategies: (1) a rule-based approach, (2) rule-based slat angle control with heuristic airflow control, and (3) optimal control of slat angle and pseudo-optimal airflow control. It was found that the differences in energy performance among the aforementioned three control strategies are significant, especially in the winter season. This is caused by dynamic control of the airflow in the cavity that reduces cooling energy efficiently. Surprisingly, it was found that the effect of dynamic control of slat angles has a little impact on energy use.

Paper No 103

HEAT CONSUMPTION ANALYSIS IN PUBLIC BUILDINGS MANAGED BY RIGA CITY MUNICIPALITY

Ēriks Krūmiņš[1], Ilze Dimdiņa[2], Arturs Lešinskis[2]
[1]Latvia University of Agriculture, Jelgava, Latvia
[2]Riga Technical University, Riga, Latvia

The aim of the present study was to analyse heat energy consumption in public buildings (managed by Riga City Municipality) in the year 2008. Our data contained information about more than 400 public buildings, including data on heat energy consumption depending on buildings use, electric energy consumption and data on the quantity and quality of windows in these buildings. Data were analysed by dividing all public buildings into 12 subgroups: schools, special status educational institutions, day-care centres, hospitals, libraries, religious buildings, recreation centres, local government buildings, museums, sports centres, music academies and shelters. The largest groups are schools (158 analysed units) and day-care centres /kindergartens/ (143 analysed units).

Our analysis focused on heat energy consumption in buildings with new double-pane windows and polyvinylchloride (PVC) frames with a heat transmittance $U \leq 1.8$ (W/m²·K), and in buildings with simple windows divided into two-panes with two separate wooden frames with a heat transmittance $U \geq 2.5$ (W/m²·K). We also compared heat energy consumption in each building group with windows covering more than 20 % of the building facades against those with windows covering less than 20 % of the building facades. We analysed data for each group separately. Our analysis showed that partial renovation – the heat insulation of buildings with windows, doors, etc. change – does not provide the heat energy consumption economy required and in of the majority of cases even increases consumption.

Paper No 267

INTERACTION BETWEEN INDOOR ENVIRONMENT AND HUMAN EXERGY LOAD

Matjaz Prek[1], Vincenc Butala[1]
[1]University of Ljubljana, Ljubljana, Slovenia.

The aim of this work was to establish a human – indoor environment interaction based on exergy analysis. Most human thermal models are based on following requirements: the body is in heat balance, and the mean skin temperature and sweat rate influencing this heat balance are within certain limits. These processes are described by applying the first law of thermodynamics and thermal comfort is related to the thermal load. Applying the second law principle to the human body, exergy is destroyed (or entropy produced) as a consequence of heat and mass transfer or conversion. These processes are dependent on the human thermoregulatory system and on the state of the environment. For the determination of heat and mass flows, a relatively simple and well-known two-node model was employed, dependent on environmental and physiological conditions. This model was expanded by applying the 2nd law of thermodynamics in order to calculate the exergy destruction. The extended analysis clearly shows the impact of specific environmental parameters on exergy destruction.

This approach enables the definition of a combination of indoor parameters that ensures minimal human exergy destruction with regard to physiological parameters. We verified of this approach by comparing the standard PMV value and the corresponding exergy destruction. The results also promise the possibility of linking human thermal comfort with human exergy load.

Paper No 90

CFD SIMULATION BASED ON THE RELATIONSHIP BETWEEN ABSOLUTE HUMIDITY AND INFLUENZA SURVIVAL RATE

Takeshi Ogasawara[1], Takashi Kurabuchi[1], Yutaro Asari[2], Kazukiyo Kumagai[3]
[1]Tokyo University of Science, Tokyo, Japan.
[2]Kajima corporation (Formerly, Tokyo University of Science), Tokyo, Japan.
[3]California Department of Public Heath, California, USA.

In this research, Harper's previous experiment data on survival ratio of the influenza virus was re-analyzed from a viewpoint of absolute humidity according to Shaman. Consequently, it was confirmed that the survival ratio of the influenza virus was more correlated with absolute humidity than relative humidity. Moreover, survival change model using the half lifetime of an influenza virus based on Harper's data was found to explain experiment results very well. The virus survival model and diffusion of a virus as a Lagrangian model was incorporated to CFD, and the case study which changed the air-conditioning system was carried out. Consequently, a possibility that indoor influenza infection would be controllable was shown by combining humidification and ventilation.

Paper No 150

CALCULATION OF INDOOR AIR TEMPERATURE FOR NIGHT VENTILATION AND ANALYSIS OF INFLUENCE FACTORS

Chang Peng, Junli Zhou, Guoqiang Zhang
College of Civil Engineering, Hunan University, Changsha, Hunan, China

Night ventilation is an effective and energy-efficient approach to improve the indoor thermal environment in summer, especially for heavyweight construction. However, due to uncertainties in the prediction of night ventilation performance, architects and engineers are still hesitant to apply this technique. This paper presents a method for calculating indoor air temperature with night ventilation on the basis of the calculation method of natural ventilation. Harmonic response method is used to estimate the indoor air temperature influenced by outdoor air and the thermal mass is considered in this method. A simple room model in Changsha is developed to analyze the effect of important parameters such as air change rates, external envelope, thermal mass and diurnal temperature range on efficiency of night ventilation. Night ventilation efficiency is mainly evaluated by the peak indoor air temperature, TDR and COP values.

Paper No 170

STUDY ON INDOOR ENVIRONMENT OF NATIONAL MUSEUM OF WESTERN ART FOR DEVELOPING RETROFIT SCHEME

Hiroshi Ochiai[1], Takashi Kurabuchi[1], Takeshi Ogasawara[1]
1Tokyo University Of Science, Tokyo, Japan.

In the National Museum of Western Art which aims at world heritage registration, the restorations of an original design and the building facilities have been an urgent subject. The purpose of this paper is to understand the indoor environment of the main building of the National Museum of Western Art through a measurement survey, and also to examine the changes in the building's internal environment through CFD in the event that it should be reverted back to its original design. For the survey, we measured the building's internal pressure, air volume around the air conditioner, air supply outlet wind speed, age of air. Based on the gathered data, we examined an indoor airflow and detailed distribution of age of air through CFD. Through the observation, Pressure of all of the rooms is negative compared to outdoor pressure. Age of air was longest in the exhibition room and shortest in the restaurant. Also, CFD examinations showed the building's internal airflow distribution, temperature distribution, age of air in the event that it should be removed the partitions.

Paper No 147

PREDICTION METHOD OF COOLING SENSATION DUE TO BREEZING AIRFLOW DRIVEN BY CROSS-VENTILATION IN DETACHED HOUSE LOCATED IN URBAN AREA

Shunsuke Suzuki[1], Takashi Kurabuchi[1], Takeshi Ogasawara[1] Masaaki Ohba[2], Tomoyuki Endo[3]
[1]Tokyo University of Science , Tokyo, Japan.
[2] Tokyo Polytechnic University, Kanagawa, Japan.
[3] Kanto Gankuin University, Kanagawa, Japan.

Utilization of cross-ventilation is an important issue for reduction of energy use in residential buildings in hot and humid seasons. Prediction of indoor airflow structures of cross-ventilated rooms is essential for evaluating thermal sensation of the occupants because enhanced heat release due to the elevated convective heat transfer is an essential part to improve thermal comfort in cross-ventilated indoor space.

This paper describes a CFD (computational fluid dynamics) procedure to estimate pressure distribution around a residential building in urban area, then cross-ventilated indoor airflow is reproduced by applying domain decomposition method of cross-ventilation. Domain decomposition method separates CFD for internal cross-ventilated flow with external flow of buildings, and enables to predict indoor airflow with reasonable computational effort and sufficient accuracy. Micro environment around human body is estimated by putting numerical thermal mannequin in the room. It was confirmed that CFD prediction results were in good agreement with corresponding experimental observations. Cooling effect due to breezing airflow was quantitatively well reproduced by the CFD technique.

Paper No 192

INDOOR SPREAD OF AIRBORNE CONTAMINANTS: A REVIEW OF APPLYING METHODS

Sandro T. Conceição[1], Arlindo Tribess[1]
[1] University of São Paulo - USP, São Paulo, Brazil

The concern about airborne contamination has worsened after the latest spread of different influenza strains and the outbreak of the virus SARS in 2003, what is somehow correlated with the increasing number of people flying around the world. In other environments, such as buildings and hospitals, several studies have shown the importance of airborne infections transmission. The researchers have applied different techniques to study the indoor transport of particles, such as tracer gas, particle atomizers and CFD simulations. Based on that, the main scope of the present work is to discuss the state of the art of methods applied to evaluate the dispersion of airborne contaminants. A variety of studies applied to aircraft cabins, buildings and hospitals are reviewed, and their main aspects related to the experimental techniques and CFD simulation are briefly discussed. Based on the presented review, it was possible to preliminary characterize the main requirements to be considered to study the airborne spread of contaminants.

Paper No 269

A STUDY ON THE ANALYTICAL MODELING OF THE RADIANT CEILING PANEL

Mi Su Shin[1], Joo Hyun Park[1], Sang Hoon Park[1], Myoung Souk Yeo[2], Kwang Woo Kim[2]
[1]Department of Architecture, Graduate School of Seoul National University, Seoul, Korea.
[2]Department of Architecture, Colleage of Eng., Seoul National University, Seoul, Korea.

Heat transfers in radiant ceiling panel can be separated into two. One is the transfer from water in pipe to each component of panel. And the other is from surface of each component to indoor space. The most precedent studies deal with first one in depth. In the latter case, transfer from surface of ceiling plate is considered but transfer from others surfaces of pipe, heat emitting plate and so on are neglected.

Because the temperature of pipe that has the water is higher in heating or lower in cooling than those of other components, heat transfer from pipe's surface to surrounding is promoted. In addition, since heat emitting plates have large area and quite different emissivity and convective transfer coefficient from other components it needs to be considered separately.

Therefore in this study, the methods to consider these were suggested and the program to calculate each temperature and heat flow of components using RC-network was developed.

Poster session 2

Paper No 232

PIV VISUALISATION STUDY IN A TWO-DIMENSIONAL ROOM MODEL WITH RAPID TIME VARYING VENTILATION FLOW RATES

Amir Sattari[1], Bengt E. G. Fallenius[2], Jens H. M. Fransson[2], Mats Sandberg[1]
[1]University of Gävle (HiG), Gävle, Sweden.
[2]Royal Institute of Technology, Stockholm, Sweden.

Optimal control of inlet jet flows is of wide applicative interest in order to enhance mixing and reduce stagnation in a ventilated room. The general approach in mechanical ventilation is to use a constant flow rate forced convection system providing the ventilation air. This type of ventilation may cause several problems such as draught, stagnation at certain occupied locations, and subsequently low ventilation efficiencies. An alternative to increase the ventilation quality that has been of interest in this study is to introduce flow variations, which is considered as a potential to reduce stagnation and increase efficiency of the ventilation. The study was conducted as a model experiment in a small-scale, two-dimensional (2-D) room model with dimensions 30200.9 cm3 with water as operating fluid. The size of the model made it possible to investigate the 2-D velocity vector field within the entire room using Particle Image Velocimetry (PIV) method and further consequent dynamical and statistical analyses have been done from the resulted PIV vector fields. The comparison between cases of constant flow rate and flow variations have been conducted for the cases of two set of base flow rates and for each one, the cases of constant flow rate and flow variations with frequencies of 0.3, 0.4 and 0.5 Hz, is considered. In this investigation we show that the calm region, with a large stagnation zone, without pulsating inflow condition becomes more active in the sense that the stagnation points are moved and that the small-scale structures are grown for increasing pulsation frequency.

Paper No 97

NUMERICAL INVESTIGATION OF AIRFLOW IN AN OPEN GEOMETRY

Boris Brangeon[1], Alain Bastide[1], Patrice Joubert[2]
[1]PIMENT, Université de La Réunion, 117 Avenue du Général Ailleret 97430 Le Tampon, France.
[2]LEPTIAB, Université de La Rochelle, Avenue M. Cr**é**peau, 17042 La Rochelle Cedex 1, France.

This paper presents a numerical investigation of airflow in an open geometry. The case under consideration is room with two opposite and decentred openings which create a strong potential for ventilation. The building characteristics dimensions are the followings: H=2.50 m height and W=6.50 m width. A temperature difference between the walls and the outside air is fixed, resulting in a characteristic Rayleigh number (Ra) ranging from 105 to 1.49 108. This room model proceeds from a benchmark exercise “ADNBATI” (http://adnbati.limsi.fr) coordinated by the by the ”Centre National de la Recherche Française -CNRS-”. This paper presents and discusses the results of this numerical study. Velocity, temperature fields, as well as heat transfer at the walls are analyzed. Values of the Nusselt number and of the mass flow rate according to the Rayleigh number are established from these first results.

Paper No 231

THE APPLICATION RESEARCH ON ONE KIND OF DYNAMIC AIR SUPPLY DEVICE IN A REAL OFFICE

Jinjing Hua, Yingxin Zhu, Qin Ouyang, Yiran Wang
Tsinghua University, Beijing, China

One kind of dynamic air supply device based on the control of brushless DC motor was designed. Experiment on effect of imitating natural wind on human sensation was conducted in a real office. Twelve subjects used the device on work in warm environment. The data of their thermal sensation was analyzed. Results suggested that the imitating natural wind was more effective in improving the thermal comfort vote (TCV) value than the constant mechanical wind in warm environment. It was proved that using imitating natural wind in warm environment could reach closely the same thermal comfort as calm air-conditioning (AC) neutral environment.

Paper No 79

WARM AIR FLOW RECIRCULATION SYSTEM FOR INDUSTRIAL HALLS

Joerg Schmid[1], Bing Gu[2]
[1]HLK Stuttgart GmbH, Germany.
[2]University of Stuttgart, Institute of Building Energetics, Germany.

This paper is supposed to be a brief feasibility study for the use of fans in industrial or similar halls (sports halls, storage halls, show rooms etc.) due to a lack of similar studies on this issue. The expected stratification of the air temperature in winter case causes higher heat losses through the ceiling and outside walls of the upper hall. The main point here is the use of fans should help to reduce these heat losses.

Paper No 233

CALIBRATION OF A NODAL NETWORK SIMULATION MODEL FOR RESIDENTIAL HYBRID VENTILATION SYSTEM

Young-Jin Kim[1], Cheol-Soo Park[1]
[1]Department of Architectural Engineering, SungKyungKwan University, Korea.

Nodal network ventilation simulation tools have been widely used for design and performance assessment of ventilation systems (Energy Recovery Ventilator (ERV), hybrid). However, the simulation results of the tools can be influenced by unknown parameters (flow exponent, flow coefficient, local terrain coefficient, leakage area of doors and windows, CO2 source strength by occupants, the stochastic nature of weather, occupants' behavior, building components, operation of diffusers/grilles, fan efficiency, etc.). To investiagte the degree of uncertainty, a series of experiments were conducted for ERV and hybrid ventilation systems in residential apartments, and compared with the CONTAMW simulation results. Based on the comparison, the CONTAMW model can predict the behavior of an ERV accurately. However, there is a significant gap between the simulation and measurements in the case of hybrid ventilation. With this regard, the authors employed a parameter estimation technique, which determines the best values for unknown parameters to minimize the difference between observation and prediction. To solve such a nonlinear and constrained minimization problem, the function LSQNONLIN in the MATLAB optimization toolbox was used. After calibration, the hybrid model was tested with another set of experiments. It is shown in the paper that the calibrated model is surprisingly accurate.

Paper No 109

ACTIVE CONTROL OF INDOOR CARBON DIOXIDE CONCENTRATION USING ADSORPTIVE CHARACTERISTICS OF ZEOLITES

Youngmin Cho, Duck-Shin Park, Soon-Bark Kwon, Woo-Sung Jung
Korea Railroad Research Institute, Uiwang, Korea

Ventilation is one of the easiest ways to control the carbon dioxide (CO2) concentration of indoor spaces. However, ventilation is not easy when the outdoor air is seriously polluted, e.g., subway, roadside. In this study, more active way to control indoor CO2 using adsorptive characteristics of zeolite was suggested. Conventional zeolites as 13X and 5A were used as the adsorbent of CO2. A 100 mL of small reactor was prepared, and the nitrogen gas containing 5,000 ppm of CO2 was passed through this reactor at a flow rate of 3 L per minute. CO2 concentration before and after the reactor were monitored by using an infrared CO2 detector. It was observed that 5A adsorbed larger amount of CO2 than 13X. 5A could be reused more than three times by baking at 180 for 24 hours, meanwhile 13X was almost deactivated after just one use. Effects of temperature and pressure on CO2 adsorption and desorption characteristics were also investigated. It was observed that the adsorbed CO2 was desorbed better at higher temperature and lower pressure. It was found that conventional zeolite can be used as CO2 adsorbent of indoor spaces.

Paper No 295

NEW PASSIVE SOLAR WALL DESIGN: SOLAR AND DAYLIGHTING ANALYSIS

Ryan Meyer[1], Ahmed Cherif Megri[2]
[1,2]Department of Civil and Architectural Engineering, University of Wyoming, USA

Passive solar heating by means of reradiating and convecting heat into a space using a vertical thermal mass, or Trombe wall, is a proven strategy in certain areas around the world. Fundamentally, the energy storage being directly related to the thermal capacity and surface area of the wall, results in not only limited design options, but ideal heat capture designs eliminate natural day lighting of a space. Using the concept of Biomimicry and the cell structure of a leaf, a new design of such a passive wall is shown to both capture more solar radiation than a standard vertical construction, while still providing sufficient daylight into the occupied space behind. A solar analysis study, using Autodesk Ecotect and a pre-assigned location with the associated Department of Energy Weather file, determined that mimicking the cell structure of a leaf captured up to 49% more solar radiation during the months when heating load is greatest. A Radiance analysis and rendering show the day lighting capabilities of the design.

Paper No 87

ANALYSIS OF PM10 SOURCE IN THE SUBWAY CABINS

Duckshin Park[1], Kyunghee Choi[2], Myunghee Kwon[2], Jungyun Lim[2],Younghoon Yoon[3] , Choonkeun Bong[4]
[1]korea Railroad Research Institute, Korea.
[2]National Institute of Environmental Research, Korea.
[3]Korea Environmental Industry and Technology Institute, Korea
[4]Greenbee Co. Ltd., Korea.

The aim of this study is to specify PM10 source in the subway cabins using receptor modelling. It was found there were three contamination sources. Based on the analyzed and the studied results of PM10 with the existing underground stations and platforms, we determined the contamination sources of this study. Source 1 contained higher Fe, Ba and Si, leading us to determine Source 1 as being ferrous related. In the case of source 3, Al, Si and Mg accounted for much of the portions, compared to those of other contamination sources. Accordingly, we analyzed this as being

soil related. Compared to other contamination sources, Source 2 showed a higher concentration in SO4 2- and NO3-.

Accordingly, we determined this source as being outdoor vehicle related.

Paper No 89

STUDY OF THE DEVELOPMENT AND CALIBRATION OF A PARTICULATE MATTERS MEASUREMENT DEVICE WITH WIRELESS SENSOR NETWORK FUNCTION

Duckshin Park[1], Soonbark Kwon[1], Youngmin Cho[1], Wootae Jeong[1], Eunyoung Park[1], Changhun Shin[2], Sungjoon Bae[2]
[1]Korea Railroad Researh Institute, Korea.
[2]Seoul Metropolitan Subway Co., Korea.

A Zigbee-based Ubiquitous Sensor Network (USN) is broadly used in many industrial applications to provide flexible measuring environments. In particular, the USN system substitutes existing measuring devices under harsh environments such as measuring the levels of air pollutants in subway stations. In order to monitor the particulate matter (PM) levels in subway tunnels, this research utilizes the USN technique. A new wireless sensor module, PMX, has been designed and manufactured to sense PM10 and PM2.5 simultaneously. Measurements has been conducted at subway tunnel in Seoul. The effects of the Zigbee-based communication protocols have been verified. The

effects and PM level have been measured, analyzed and compared with those from existing commercial PM measurement device, the Dust spectrometer, in sensing the PM10, PM2.5 levels of the selected subway tunnel. The test result indicates that the two systems showed a similar time series trend while the measured values were somewhat different. A simple correlation analysis of the two groups of data showed the coefficient of determination of 0.7 for PM10 and 0.9 for PM2.5. In the case of PMX data, most data were concentrated around the trend curve. Therefore, the calibration of PMX data was needed to apply it in the field. For the calibration, statistical processes of simple linear regression and nonlinear regression were selected. The result showed that the correlation coefficient of simple linear regression was 0.7 for PM10 and 0.9 for PM2.5 while that of nonlinear regression was 0.8 for PM10 and 0.9 for PM2.5, indicating the nonlinear regression showed higher correlation coefficient. Therefore, it was determined that nonlinear regression was a better method to calibrate the system developed for this study.

Paper No 190

A STUDY OF INTEGRATED PERFORMACE EVALUATION OF SUSTAINABLE RESIDENTIAL BUILDING IN SUMMER

Lee, Ga-Ram[1], Kim, Ji-Young[2], Kim, Taeyeon[3], Leigh, Seung-Bok[4]
[1]Graduate Student, Department of Architecture, Yeonsie University, Korea
[2]Ph.D. candidtate, Department of Architecture, Yeonsie University, Korea
[3]Professor, Department of Architecture, Yeonsie University, Korea
[4]Professor, Department of Architecture, Yeonsie University, Korea

Nowadays, the world is trying to solve the environment and energy problems. One of efforts to address these problems is Sustainable Housing such as Passive House and Zero Energy house. Korean government plans zero carbon in all houses from 2025 and stepped energy reduction goals. In this situation Greenhomeplus was constructed on the purpose of developing energy reduction models in May 2010. It has 5 models of a base model and 4 energy reduction models with 40%, 60%, 80% and 100%. To accomplish the energy reduction goal, insulation and air-tightness are improved and various high efficient service systems are installed and lastly renewable energy systems such as geothermal, solar thermal and photovoltaic are applied.

The purpose of this study is to verify the effect of the integrated technology of each element on the building. To achieve this purpose, these models experiments and network simulations are used. In the simulation annual cooling load is estimated after simulation verification with the experimental

data without internal gains. Annual cooling load decreased by 27% in Gh+40 model, 23% in Gh+60 model, 33% in Gh+80 model and 27% in Gh+100 model compared to base model.

Paper No 136

SEASONAL PERFORMANCE OF HEAT PUMPS IN COLD CLIMATE

Lari Eskola, Juha Jokisalo, Kai Sirén
Aalto University, School of Engineering, Department of Energy Technology, Espoo, Finland.

In cold climate heat pumps are used for heating. Standard EN 15316-4-2:2008 provides a method to calculate heat pump systems energy requirements and efficiencies. Though the standard gives detailed step by step guideline for calculating performance factors the method is too sophisticated to be used in everyday estimations. For that reason there has been a need to generate more simplified methods to calculate energy performance of heat pump in the Finnish climate. Performance of heat pump depends on climate, temperature levels in distribution system and heat requirement. The guidelines of two different methods, simplified and detailed method, were developed for users with different requirements of information. The simplified method is based on ready-made tables of seasonal performance factors for different heat pumps in different climate zones. From tables you can find SPF values e.g. for air to air heat pump (2.0-2.8) or ground source heat pump (2.5-3.5). With detailed method more accurate seasonal performance factors can be calculated. It is intended to be used when more information is available and needed. According to the results of these methods heat distribution network temperature has more effect to seasonal performance factor than climate.

Both of the methods will be presented as a Finnish guideline for heat pump calculation.

Paper No 156

3D VENTILATION AIRFLOW IN A TEST FACILITY: VALIDATION OF REYNOLDS-AVERAGED NAVIER-STOKES APPROACH APPLICATION

Nikolay G. Ivanov[1], Segrey V. Egorov[2]
[1] New Technologies and Services, St. Petersburg, Russia.
[2] St. Petersburg State Polytechnic University, Russia

The objective of this contribution is to validate 3D modeling of ventilation airflow based on the RANS approach using velocity data available for a test facility. The test facility used in the experiments by Mocikat et al. (Experiments in Fluids; 34: 442-448) is a scaled room 0.6×0.4×0.4 m with a partition wall blocking half of the cross-section. The inlet circular section is located at the end of the straight pipe with the length of 45 diameters. The Reynolds number values based on the mean inlet velocity and inflow pipe diameter are in the range of 2-5×104. The flow in the room is induced by suction through the circular outlet pipe.

The computational data were obtained with a commercial code of second order accuracy ANSYS FLUENT. The incompressible fluid model with a constant molecular viscosity was assumed. The standard and RNG versions of the k-ε model, both with the enhanced wall treatment option, were used for computations. The computations have been carried out using grids of hexahedral mesh elements. Even though the RANS approach is used, the computed air flow is unsteady, with lowfrequency velocity fluctuations of high amplitude. Time-averaged velocity fields obtained with the unsteady RANS approach correlate well with the experimental data on the longitudinal and vertical velocity components. To obtain a steady-state solution it was suggested to increase in the default value of the closure model constant $C\mu$. The method proposed could be useful if it is necessary to perform fast CFD-based evaluation of a ventilation configuration.

Paper No 25

STUDY ON THE DISTRIBUTION OF CARBON DIOXIDE FROM BREATHES IN DENSELY POPULATED INDOOR SPACES

Youngmin Cho, Soon-Bark Kwon, Duck-Shin Park, Woo-Sung Jung, Ji-Yun Lee
Korea Railroad Research Institute, Uiwang, Korea

The carbon dioxide (CO2) concentration of densely populated indoor spaces often exceeds the recommended level of 1,000 ppm. Because high concentration of CO2 causes dizziness, headache, and the lack of concentration, it is important to keep low CO2 of classroom and library is important. The averaged CO2 concentration in these indoor spaces have been frequently reported, but the distribution of CO2 in indoor space has been hardly reported. In this study, CO2 concentrations of 10 different points in one indoor space were monitored to see how CO2 exhaled from breathes of people moves in the space. 5 CO2 sensors were placed at the bottom and the others were placed at the ceiling of the room to investigate the effect of height. The results showed that CO2 concentration was higher at the ceiling than at the bottom due to the movement of warm breathing air to the ceiling. CO2 near the ceiling moves toward wall, and the air was cooled down to move to the bottom due to the heavier molecular weight of CO2 than the air. This study will be used for the modeling of CO2 movement in indoor spaces to elucidate the diffusion route of CO2.

Paper No 284

THE EFFECT OF DISTINCT VENTILATION STRATEGIES ON THERMAL PERFORMANCE OF BUILDINGS WITH HIGH THERMAL RESISTANCE IN BRAZIL

Karin M. S. Chvatal, Michele Marta Rossi
Institute of Architecture and Urbanism of São Carlos, University of São Paulo (USP), Brazil

The benefits of decreasing the U-value of the external envelope are evident in a typical winter situation, where heating needs are directly reduced. But the consequences during the summer are not so evident. Recent studies indicate cases where added insulation does always not contribute to more comfort and less energy consumption, especially in countries where summers are long with high outdoor temperatures and high solar radiation exposure. The aim of this paper is to investigate the coupled influence of the U-value of the external envelope and distinct ventilation strategies on the energy consumption and thermal comfort of buildings in Brazil. Commercial buildings were simulated using the program EnergyPlus with various combinations of parameters: climate, internal gains and ventilation strategies, against distinct insulation levels of the external envelope. The studied ventilation strategies were mechanical cooling with distinct temperature set-points, day ventilation and 24-hours ventilation. The results showed that heating energy consumption always decreased when envelope insulation level was increased. In São Paulo and Curitiba, cooling consumption always increased when the U-value of the envelope was reduced. For Manaus, a distinct pattern of result was found: a lower envelope U-value can be beneficial, reducing the cooling energy consumption. Increasing the set-point temperature to 25.5 ºC or to the upper limit of the comfort zone were important measures to save energy. Natural ventilation, especially when it was extended to the nonoccupied period, proved to be an important strategy for reducing the negative impact due to envelope insulation increase.

Technical session 22 Modelling and visualization

Paper No 243

TWIN CROSS-SHAPED JETS NUMERICAL SIMULATION FOR INNOVATIVE AIR DIFFUSER OPTIMISATION

Meslem Amina[1],*, Aliou Dia[1], Claudine Beghein[1], Mouhammad El Hassan[1], Ilinca Nastase[2], Pierre-Jean Vialle[3]
[1] LEPTIAB, University of La Rochelle, France
[2] Technical University of Construction in Bucharest, Roumania.
[3] CIAT Research and Development Center, Culoz, France.

We are interested in HVAC applications using the passive control of the air injection geometries. The studied diffuser is a perforated panel using lobed cross-shaped orifices (Meslem, Nastase et al. 2010). Aiming the optimisation of the perforated panel relatively to induction and mixing, the isolated twin jets flow is analysed numerically as a function of orifice to orifice spacing and relative orifices orientation.

In this study, we propose a comparison between three popular turbulence models and experimental results, in order to establish which is more appropriate, to predict the behaviour and entrainment performance of a non-circular orifice twin jets at moderate Reynolds number. Numerical simulations using the standard k-e model, the Shear Stress Transport (SST) k-ω model and the Reynolds Stress model (RSM) are compared with PIV measurements. It is shown that the k-e and the RSM are more appropriate to predict jet potential core length, jet centreline streamwise velocity evolution, and flow expansion in the symmetry plane of the twin jets flow. However, these models overestimate the global flow expansion and the jet volumetric flow rate. For the prediction of such dynamic integral quantities, SST k-ω seems more appropriate. It is shown that k-e and RSM generate an important level of turbulent kinetic energy in the jet near field. This leads to earlier merging of the parallel jets in comparison to measurements and SST k-ω model results. Important production of turbulent kinetic energy by these models in the jet near field is probably at the origin of the overprediction of the induction. Based on the obtained results, the SST k-ω model becomes an interesting compromise for quickly optimizing non-circular orifice design, orifice to orifice spacing and relative orifices orientation on the perforate panel diffuser.

Paper No 122

COMPARING *k-ε* MODELS ON PREDICTIONS OF AN IMPINGING JET FOR VENTILATION OF AN OFFICE ROOM

Huijuan Chen[1.2], Bahram Moshfegh[1.2]
[1]Department of Building, Energy and Environmental Engineering, Faculty of Engineering and Sustainable Development, University of Gävle, Gävle, Sweden
[2]Division of Energy Systems, Department of Management and Engineering, Linköping University, Linköping, Sweden

The objective of this study is to compare the performance of different *k-ε* models, i.e. the Standard *k-ε*, the Renormalization Group (RNG) *k-ε*, and the Realizable *k-ε*, with a two-layer model for the prediction of the mean velocity field and the temperature pattern from a newly designed impinging jet supply device for ventilation of an office room. The numerical predictions are validated against the detailed experimental measurements.

The experimental investigation was performed in a test room with the dimensions 4.2×3.6×2.5 m, as a mock-up of a single-person office. Detailed velocity and temperature field measurements including the comfort zone and the jet developing region along the floor were carried out. The in-house made single-sensor hot-wire probe and the thermocouple are measuring instruments used to investigate the mean velocity, turbulence intensity and temperature. The boundary conditions for Computational Fluid Dynamics (CFD) study are obtained from the same set-up measurement.

The results mainly consist of the flow field presentation, i.e., the velocity and temperature profiles in the comfort zone and the jet developing region along the floor. The comparisons between the results from the three versions of the *k-ε* models and measurements show generally satisfactory agreement, and better consistency is observed at the free jet region and the wall jet region that farther from the impingement zone. Among the three tested turbulence models, RNG shows the best overall performance.

Paper No 128

INTEGRATION OF THREE-DIMENSIONAL CFD RESULTS INTO ENERGY SIMULATIONS UTILIZING AN ADVECTION-DIFFUSION RESPONSE FACTOR

Kyosuke Hiyama[1], Yoshihiro Ishida[1], Shinsuke Kato[1]
[1]The University of Tokyo, Tokyo, Japan

Indoor climates have a three-dimensional spatial distribution caused by threedimensional airflow. To understand building performance, we must integrate these spatial distributions into building simulations. However, conventional energy simulations are based on the assumption that perfect mixing of air streams with different temperatures occurs. Therefore, it has been difficult to evaluate the effectiveness of energy conservation methods that utilize a thermal distribution mechanism within a room. Taking into account the above conditions, we have developed a calculation method that can achieve more accurate timeseries analysis. This is accomplished by combining the newly developed method with the conventional energy simulation method. In the new method, we calculate, in advance, the heat response in a static flow field using computational fluid dynamics (CFD) analysis. Then we calculate advection-diffusion response factors and integrate them into the energy simulation as a factor in the three-dimensional thermal distribution within a room.

In this paper, we show a calculation example using the model for high ceilings with high-temperature exhaust. As a result, we conclude that our new calculation method, in combination with a dynamic heat load calculation, will offer possibilities for a long-term, non-steady-state energy simulation, even on personal computers, based on the room temperature distribution data obtained using steady-state calculations with CFD analysis.

Paper No 41

A VALIDATION PROCESS FOR CFD USE IN BUILDING PHYSICS – STUDY OF THE DIFFERENT SCALES

Mathieu Barbason[1], Sigrid Reiter[1]
[1]University of Liège, Liège, Belgium.

Due to growing environmental concerns, Computational Fluid Dynamics (CFD) is more and more used in building physics. Until today, the research community has validated separately several cases but there is no global validation process for this method. The aim of this paper is to provide a way for new users to develop and improve their CFD skills. This paper deals with the different geometry scales involved in building physics. Experimental results are available and will assess the ability of CFD to predict accurately the thermal behaviour of buildings. Thanks to this validation process, building engineers and architects can improve their simulations results and their understanding of the physical phenomena in building physics.

Paper No 51

A NEW DES MODEL FOR INDOOR AIRFLOW MODELING

Miao Wang[1], Qingyan Chen[1]
[1] School of Mechanical Engineering, Purdue University, West Lafayette IN 47907, USA.

The airflow in enclosed environments is a wall bounded flow, consisting of circulation, flow separation, and thermal plumes in transitional to fully developed turbulence. This type of flow is difficult to simulate using Reynolds Averaged Navier-Stokes (RANS) models due to the complex flow features. The more advanced Large Eddy Simulation (LES) may solve such flow features, but requires very fine grids near solid surfaces, which makes it very computationally demanding. The hybrid RANS/LES simulation, or Detached Eddy Simulation (DES), which uses a RANS model for the near-wall boundary layers to avoid excessively fine grids, as well as a LES for the far-wall flow region, looks promising. However, the available DES models did not perform well for indoor airflows due to the RANS model they used. This study has developed a new DES model for indoor airflow using a semi-v2f model, and this model correctly predicted near-wall flows by taking into account the wall normal stress. This study applied the new DES model to a mixed-ventilation and a strong buoyancy-driven flow in rooms. The new model can accurately predict the flows, and it is robust.

Paper No 175

A COMPARISON OF LES AND RANS IN A SIMULATION OF COLLIDING PLANE JETS

Tomas Brockmann[1], Hannu Koskela[2], Panu Mustakallio[3], Timo Siikonen[1]
[1]Aalto University, School of Engineering, Espoo, Finland.
[2]Finnish Institute of Occupational Health, Turku, Finland.
[3]Halton Ltd., Kausala, Finland.

The collision of plane jets between ceiling-mounted active chilled beams is studied in the present paper. Draught problems are often caused by the deection of an attached ceiling jet downwards into the occupied zone. As computational uid dynamics is utilized, a correct simulation of these jets is important for the estimation of thermal comfort. RANS (Reynolds-averaged Navier-Stokes) simulations tend to overestimate the maximum ow velocities and the thermal discomfort in the occupied zone. In order to improve the prediction, LES (large-eddy simulation) models are studied in comparison with RANS using di_erent turbulence models. Subgrid-scale turbulence modeling is based on dynamic Smagorinsky or turbulence kinetic energy equation models. The desirable symmetric ow state was found to be negatively stable with LES when applying non-turbulent inlet boundary conditions. As a remedy, turbulent time-dependent ow distributions are applied. Simulations are made using the OpenFOAM 1.5 -code. The beams were installed in a laboratory test room and the ow pattern between the units was measured for validation. The ow pattern after the collision of the jets was found to be uctuating.

Technical session 23 Evaluation, control or measurements of indoor thermal environment

Paper No 35

AVERAGE AND VARIATION OF THE INDOOR TEMPERATURE IN SWEDISH OFFICES

Arne Jönsson
Secondary Vocational Training, Harnosand, Sweden

To investigate if the indoor temperature distribution calculated with the theory for indoor temperature as a collective factor of production coincide with the distribution of measured indoor temperatures. A collective factor of production is the same to everybody and the cost for producing the factor do not change if more uses the factor. An occupant does not have a loss of production when the temperature is above a threshold temperature for that occupant. The threshold temperatures for the occupants in an office are the same as for the households in Swedish single unit dwellings. It is the members from the households who work in the offices.

The theory uses economic data such as the price of a work hour 250 SEK/h, the price of heat 0,5 SEK/kWh and the specific heat demand 73 W/°C pers. (7 SEK = 1 USD)
The measured temperatures follows a distribution between the theoretical distributions for zone size 5 and zise 10 workplaces. In zones with 5 or 10 workers there are 20 or 10 % disturbed. If only one worker in every zone is disturbed. This coincides with earlier estimates of the percentage dissatisfied. The theory also predicts that smaller temperature control zones will get a lower average temperature and a higher variation between the zones.

Paper No 86

SEATS LOCAL THERMAL COMFORT EVALUATION TOOL

Clenilson Jordão Gonçalves[1], Arlindo Tribess[2]
[1]Embraer S.A. São Paulo, Brazil.
[2]University of São Paulo, Department of Mechanical Engineering, São Paulo, Brazil.

Seats acclimatization systems have been lately developed and delivered as microclimate solutions in response to the personal comfort increasing demand, mainly on the transportation industry. The influence of the use of such systems on the improvement of passengers' satisfaction with the ambient thermal comfort is well recognized, either heating or cooling the seat.

In the present work the heat transfer characteristics at the interface between the person and the seat as well as its relation with the pressure distribution were studied. A thermal manikin was developed and calibrated thermal and ergonomically to simulate the sensible heat transfer of a seated person.

The heat transfer tests showed that: the local heat flow directly depends on the contact pressure at the passenger seat interface; the changes on the heat flow caused by passenger metabolism (MET) modifications are inversely proportional to the seat global thermal resistance, i.e., the higher the MET the lower the thermal resistance; and that modifications on the seat pressure distribution due to the weight increase or decrease are directly proportional to the seat global thermal resistance, i.e., the lower the weight of the passenger the lower the seat global thermal resistance.

Based on the results it may be concluded that the testing methodology and the developed thermal manikin adequately explore the opportunities for optimising the seat heat transfer characteristics and its systems toward the local thermal comfort.

Paper No 168

INDOOR ENVIRONMENT AND CAPACITY OF AN AIR CONDITIONER IN A RESIDENTIAL HOUSE USING NATURAL VENTILATION

Hiromi Habara[1], Shigeki Nishizawa[2], Akinori Hosoi[3], Hisashi Miura[2], Takao Sawachi[4]
[1]Osaka University, Osaka, Japan.
[2]National Inst. of Land and Infrastructure Management, MLIT, Ibaraki, Japan.
[3] Prefectural University of Kumamoto, Kumamoto, Japan.
[4] Building Research Institute, Ibaraki, Japan.

In Japan, natural ventilation through large openings has been traditionally used for cooling in summer and medium seasons. Nowadays, as global warming has become a serious problem, conscious utilization of natural ventilation is encouraged for energy conservation. A new Japanese standard enforced in 2009 first adopts a quantitative evolution method for the energy conservation effectiveness of natural ventilation. Such a social trend indicates that natural ventilation is expected to be an important technique for energy conservation.

Accordingly, we have conducted experiments in a reinforced construction (RC) multi-family residential building to verify the effectiveness of natural ventilation in energy conservation by simulating the various occupants' behavior automatically. In this paper, we performed experiments with 2 patterns of thermal control behaviour which differ in natural ventilation usage. Based on the experimental results, we analyzed cooling hours and indoor environment. In addition, we estimated heat discharge rate by natural ventilation and heat capacity of an air conditioner to discuss heat removal structure.

The results suggest that although heat discharge by natural ventilation has a positive effect on the reduction of cooling hours, thermal energy storage during occupants' absence in the daytime could cause additional air conditioner usage in the evening and night. And natural ventilation is estimated to reduce heat capacity of an air conditioner by 35% in an average year.
In future studies, we will research on the energy conservation techniques which utilize heat discharge effects of natural ventilation to prevent thermal energy storage in the room during the occupants' absence.

Paper No 189

COMPARISON OF THE THERMAL ENVIRONMENT IN ROOMS WITH CHILLED BEAM AND RADIANT PANEL SYSTEMS

Risto Kosonen[1], Panu Mustakallio[1] Arsen Melikov[2], Marcin Duszyk[2]
[1]Oy Halton Group Ltd., Helsinki, Finland.
[2]International Centre for Indoor Environment and Energy, DTU Civil Engineering,
Technical University of Denmark, Lyngby, Denmark.

Compared with convective systems radiant systems explore the advantage of influencing the mean radiant temperature for maintaining similar thermal conditions at lower room air temperature in heating mode and higher room air temperature in cooling mode. Some manufactures claim that the difference of the operative temperature between radiant and convective systems is claimed to be as high as 2-3 oC. The difference between radiant system with ceiling installed pannels combined with mixing air distribution and convective system of chilled beams was studied with regard to the mean radiant temperature and radiant temperature asymmetry in a mock-up office at different internal load in both heating and cooling modes.

The operative temperature and the radiant temperature asymmetry were measured at four locations and three heights in the occupied zone. The radiant temperature asymmetry was measured in three directions: vertical direction between floor and ceiling, and two horizontal directions respectively parallel to walls.

The difference in operative temperature at 1.1 m height measured with the two systems under the same conditions and at same locations was insignificant (about 0.15 oC). The operative temperature was more or less uniform over the occupied zone and there was not any significant

difference in the operative temperature measured with the two systems. The radiant asymmetry was lower than 5.5 oC in all measured cases. The difference in the radiant temperature asymmetry measured with the chilled beam system and the radiant panel system was also small.

Paper No 146

REDUCED SCALE APPROACH OF TENT NATURAL BUOYANCY AND WIND-DRIVEN VENTILATION

Sophie Herpin[1], Xavier Faure[2] and Hervé Brébion[2]
[1]Oxylane, Thermal Comfort Laboratory, Villeneuve d'Ascq, France.
[2]CSTB, CAPE department, Nantes, France.

The present contribution consists in a reduced-scale approach of buoyancy and wind-driven ventilation inside a classical family tent of moderate dimensions (5.4m long, 1.85m high and 2.3m large). The similarity hypothesis chosen are based on the internal Archimedes number and volumetric flow rates through the tent openings. Both the full scale and a reduce-scale model (1/2) are investigated in a climatic wind-tunnel, and the similarity hypothesis can hence be assessed experimentally. The measurements performed inside the tents are focused on global quantities: air change rate obtained with the tracer gas technique, and mean thermal gradient in height. In order to maintain the same Archimedes number (and hence the global air column weight), the thermal gradient in the half-scale model needs to be twice bigger than for full scale. This is achieved by increasing the tunnel radiation intensity on the reduced-scale model. The similarity hypotheses are studied for different configurations, by varying the wind orientation and speed, and the surface area of the tent openings. The application of the reduce-scale methodology is finally identified as relevant, and some limitations are pointed out, mostly for forced convection regime.

Paper No 256

CONTROL OF AIR VELOCITY AND TEMPERATURE DISTRIBUTION IN CLASSROOM USING CEILING FAN

Yoshihisa Momoi[1], Toshio Yamanaka[1], Kazunobu Sagara[1], Hisashi Kotani[1], Natsuka Wakamatsu[1]
[1]Osaka University, Osaka, Japan.

This study aims to establish a design and operation method for an air-conditioning system with ceiling fans, which can realize comfortable air-conditioning and energy conservation all year round. A target space of this study is a classroom. Recently, the number of schools which are equipped with airconditioners has been rapidly increasing in Japan. This paper reports the results of a measurement and questionnaire survey conducted on a classroom environment where ceiling fans (under downward airflow condition) and air-conditioners (under cooling condition) are operated simultaneously, in order to consider energy conservation and comfortable air-conditioning with ceiling fans for cooling during the summer. As a result, it is confirmed that the ceiling fan is beneficial to reduce the vertical temperature difference. There is a correlation between thermal sensation and SET* values and the SET* value range for the neutral thermal sensation is 25.6°C–28.9°C. There is a significant correlation between pleasantness of air movement and thermal comfort. It can be understood that pleasantness of air movement and thermal comfort tend to depend on individual preferences, and desirable airflow and temperatures differ greatly with each individual.

Technical session 24 New technologies for heating and cooling or ventilation/AC

Paper No 301

A COMPARISON OF COOLING COIL ENERGY PERFORMANCE OF TWO DIFFERENT STRATEGIES IN AN OPERATING ROOM IN A HOT AND HUMID CLIMATE

Chandra Sekhar
National University of Singapore, Singapore

In hot and humid climates, considerable energy is used in the conditioning of outdoor air, which involves cooling and dehumidification. The high latent load poses a challenge in extracting the moisture out of the outdoor air without resorting to overcooling that eventually leads to reheating and increased energy consumption. This is particularly true in ventilation applications that require 100% outdoor air, such as operating rooms in a hospital. A standard cooling coil selection program is used in this study to compare the energy performance of two strategies : a) Low Face Velocity – High Coolant Velocity (LFV-HCV) coil and b) Heat pipe integrated cooling coil. The design of LFV-HCV coil involves operating parameters that are distinctly different from conventional practice but are aimed at achieving the maximum dehumidifying performance per unit of sensible cooling. The incorporation of a heat pipe with a conventional coil provides “free” precooling and “reheating” that allows overcooling to be employed as a strategy to achieve better dehumidification, and thus, resulting in attractive energy benefits. The findings reported in this paper are based on the comparison of cooling coil performance for an operating room in Singapore.

Paper No 283

BUILDING THERMAL ENERGY STORAGE – CONCEPTS AND APPLICATIONS

Georgi Pavlov[1], Bjarne W. Olesen[1]
[1]ICIEE, Department of Civil Engineering, Technical University of Denmark, 2800-Lyngby, Denmark.

The use of Thermal Energy Storage (TES) in buildings in combination with space heating, domestic hot water and space cooling has recently received much attention. A variety of TES techniques have developed over the past decades, including building thermal mass utilization, Phase Change Materials (PCM), Underground Thermal Energy Storage, and energy storage tanks. In this paper, a review of the different concepts for building or on-site integrated TES is carried out. The aim is to provide the basis for development of new intelligent TES possibilities in buildings.

TES systems for cooling or heating capacity are utilized in applications where there is a time mismatch between the demand and the economically most favourable supply of energy. TES can provide short term storage for peak shaving as well as long term storage for the introduction of renewable and natural energy sources.

Sustainable buildings need to take advantage of renewable and waste energy to approach ultralow energy buildings. Utilization of low-exergy heating and cooling sources requires that energy storage is integrated into sustainable building design. A coordinated set of actions for improved TES designs are needed if the potential benefits are to be fully realized. Well designed systems can improve building's energy efficiency and comfort level, yielding significant cost savings and promising payback period.

Technical session 24, Theme: New technologies for heating and cooling or ventilation/AC, Room R3, Wednesday 2011-06-22, 09:00 to 10:30

Paper No 39

YEAR-ROUND ENERGY SAVING POTENTIAL OF A STRATUM VENTILATED OFFICE WITH TEMPERATURE AND HUMIDITY CONTROL

C.K. Lee, Zhang Lin, K.F. Fong
Building Energy & Environmental Technology Research Unit, School of Energy and Environment & Division of Building Science and Technology, City University of Hong Kong, Kowloon, Hong Kong, China

Stratum ventilation has been proposed to cope for elevated indoor temperature recommended by governments in East Asia. TRNSYS is used for computation of the space cooling loads, sensible and latent, and system energy consumption. For the typical office, the year-round energy saving is found to be substantial at 20% and 40% when compared with displacement ventilation and mixing ventilation respectively.

Paper No 292

FAVOURABLE SCENARIOS TOWARDS ZERO ENERGY BUILDINGS

Paris A. Fokaides
RD Hydraulis Ltd
P.O.Box 22101, 1517, Nicosia Cyprus

EU member states forged in 2009 a long-awaited compromise on the recast buildings directive, agreeing that all new buildings would have to comply with high energy-performance standards after the end of 2020. The recast Energy Performance of Buildings Directive will require the public sector to take the lead by owning buildings with "nearly zero" energy standards by the end of 2018, two years in advance of the private sector. The objective of this paper is to determine the range of potential consequences to the European cities resulting from widespread implementation of ZEBs and relevant environmental technologies in accordance with the national goals set by the EU Member States. As EU member states are moving ahead with their targets and strategies for ZEBs, this work presents the most possible scenarios for the implementation of the EU recast buildings directive regarding ZEBs by 2020, and adapts them from national to community- level scenarios. A detailed review regarding the existing EU member states definitions and policies on ZEBs and financial incentives, as well as the current status of renewable energy technologies for ZEBs is presented. The context of the most favourable scenarios determine the optimum combination of environmental technologies and thus building insulation, renewable energy technologies promotion and energy saving, in order to reach a cost effective energy neutral level. The examined case studies are based on existing best practice examples for ZEBs and pilot projects in EU.

Paper No 82

ENERGY EFFICIENT NANOENCAPSULATED VARIABLE AIR VOLUME THERMAL STORAGE AIR CONDITIONING SYSTEM FOR MECHANICALLY VENTILATED BUILDINGS

R. Parameshwaran[1], S. Kalaiselvam[2], S. Harikrishnan[1]
[1] Department of Mechanical Engineering, Anna University Chennai, Chennai, India.
[2] Centre for Nanoscience and Technology, Anna University Chennai, Chennai, India.

This work aims at achieving enhanced energy-savings potential through the implementation of a new variable air volume (VAV) air conditioning (A/C) system integrated with the nanoparticleembedded latent thermal energy storage (NLTES) system for commercial buildings subjected to hot and humid climatic conditions. The new VAV-latent thermal energy storage A/C system configured with nanoencapsulated phase change material (PCM) was experimentally investigated using the energy efficient demand controlled ventilation (DCV) and combined DCV-

economizer cycle ventilation (ECV) techniques. The nanoparticle-embedded PCM synthesized and characterized in this study was considered to be a promising candidate for acquiring enhanced energy storage capabilities. Appropriate mole concentration of the nanoparticles introduced into the base PCM had exhibited high latent heat of fusion, high rate of heat release, good charging and discharging characteristics as well and improved the energy performance of the integrated VAV-NLTES A/C system.

The test results infer that the per day average energy efficiency of the VAV-NLTES A/C system was increased substantially to 32% and 53% in the DCV and combined DCV-ECV modes of ventilation respectively, while compared to the conventional chilled water and fan coil A/C systems operated for the same design conditions. In total, the on-peak energy conservation potential for cooling and ventilation in DCV and combined DCV-ECV ventilation schemes as achieved by the present integrated A/C system was expected to be 41% and 46% respectively. Furthermore, the effective ventilation modeling and control strategies executed on the air streams had enabled the VAV-NLTES A/C system to maintain the purity of the air and achieve thermal comfort as well without compromising the energy efficiency that is being attributed for the development of a sustainable built environment.

Paper No 85

IMPROVED PERFORMANCE OF LATENT THERMAL STORAGE COOLING SYSTEM USING DISPERSED NANOPARTICLES FOR LOW ENERGY BUILDINGS

S. Harikrishnan[1], S. Kalaiselvam[2], R. Parameshwaran[1]
[1]Department of Mechanical Engineering, Anna University, Chennai – 600 025, India.
[2]Centre for Nanoscience and Technology, Anna University, Chennai – 600025, India.

In this work, the energy performance of the latent thermal energy storage (LTES) air conditioning (A/C) system dedicated for building cooling was experimentally analyzed using the plain and finned encapsulated dispersed nanoparticle techniques, that yielded improved heat storage capacities during the freezing and melting cycles of the phase change material (PCM). The cooling potential of the present A/C system was characterized using nanoencapsulated PCM in order to satisfy the peak cooling and ventilation load demands as well as to achieve good thermal comfort inside the conditioned space.

The experimental results suggest that for the estimated cooling and ventilation load demands of the building considered, freezing and melting time consumed by the nanoparticle enriched PCM in plain encapsulation configuration was reduced by 26.6% and 30.2% respectively while compared to the referenced conventional PCMs. Similarly, the freezing and melting time for the nanoparticle based PCM with finned encapsulation was reduced by 31.7% and 35.3% respectively. Moreover, the dispersed nanoparticles facilitated for an improved thermal conductivity and heat storage and release rate of the PCM thereby; the power consumed by the cooling and ventilation systems was minimized considerably augmenting the significance of the present nano-based LTES A/C system for the modern low energy building cooling applications.

Technical session 25 Ventilation strategies preventing airborne infections and Modelling and visualization

Paper No 161

IMPACT OF BREATHING ON THE THERMAL PLUME ABOVE A HUMAN BODY

Daria Zukowska[1], Arsen Melikov[1], Zbigniew Popiolek[2] and Joanna Spletsteser[1,2]
[1]International Centre for Indoor Environment and Energy, (www.ie.dtu.dk),
Department of Civil Engineering, Technical University of Denmark, DK-2800 Denmark.
[2]Department of Heating, Ventilation and Dust Removal Technology, Faculty of Energy and Environmental Engineering, Silesian University of Technology, PL-44-101 Gliwice, Poland.

The characteristics of the thermal plume above a human body should be well-defined in order to properly design the indoor environment and allow correct simulation of the indoor conditions by CFD or experimentally. The objective of the present study is to investigate the influence of breathing on the characteristics of the thermal plume generated by a sitting person.

The experiment was performed in a climate chamber ventilated with an upward piston flow. Air temperature was 23°C, vertical temperature gradient was approx. 0.07 K/m and velocity was lower than 0.05 m/s. Radiant temperature asymmetry was close to 0°C. A thermal manikin with female body shape equipped with an artificial lung was used to simulate the dry heat loss and the breathing process of a sitting occupant. Three cases were examined: non-breathing, exhalation through nose, and exhalation through mouth. Measurements of the air temperature and speed in the plume cross-section 0.7 m above the manikin head were performed.

Exhalation through the mouth affects the characteristics of the thermal plume 0.7 m above the manikin head, while exhalation through the nose has only small impact. Air velocity and temperature excess distributions in the plume generated by the manikin exhaling through the nose are comparable to the distributions above the non-breathing manikin. Exhalation through the mouth causes wider plume cross-section and increases the volume flux, momentum flux, buoyancy force density and enthalpy flux compared to the non-breathing case.

Paper No 307

CONTROL OF JETS FOR VENTILATION BY USING DIFFERENT COMBINATIONS OF BLOWING AND SUCTION

Elisabet Linden[1], Mats Sandberg[2]
[1]University of Gävle, Department of Built Environment, Sweden.
[2]University of Gävle, KTH Research School, Sweden.

Interaction between jets occurs in ventilated rooms. Jets issued from two supply air terminals may collide and the position of the collision point depends on the relative strength of the jets. At normal ventilation flow rate the effect of sucking is a short range phenomenon. To explore the suction effect test with high extraction flow rates have been done.

The effect of the relative strength of blowing or extracting was explored experimentally. The test room was provided with a rig with air terminals located opposite to each other. The direction of the flow was in the vertical direction. The cases studied were 1) An impinging jet obtained by discharging the flow from the upper air terminal towards a horizontal plate. 2) A combination of blowing and suction was explored by directing the jet above towards the horizontal plate provided with a nozzle through which air was sucked. 3) Colliding jets were created by blowing the upper jet downwards towards a jet blowing upwards.

The air movements were visualized by smoke. Both a digital single-lens reflex camera and a video camera were used to capture the visualization. The pictures were analyzed with image processing. The changing shape of the jet in the case of blowing and suction was retrieved and for the colliding jets both the mean position and the variation in height of the collision point were determined.

Paper No 63

AIR CONDITIONING IN HIGH CARE FACILITIES FOR IMMUNOSUPPRESSED AND IMMUNO-COMPROMISED PATIENTS

Walter Bosschaerts[1], Rolf Wagemakers[1], Bart Janssens[1], Karim Limam[2]
[1]Department of Mechanics, Royal Military Academy, Brussels, Belgium.
[2]Leptiab, Université de la Rochelle, France

The objectives of an air conditioning system in a hospital reach much further than just promoting a comfortable environment. It is often of the utmost importance to obtain the right ambient conditions for proper treatment of the patient in order to obtain rapid physical recovery or to protect either the patient or the environment from contamination. For immuno-suppressed and immuno-compromised patients the air conditioning involves a control of temperature, of swift temperature changes, of pressure relative to the outside environment, of humidity, of ventilation, of air purity and particle deposit. The clean air, delivered to the room via a laminar flow type rectangular modular perforated face, falls on the patient and the sterile equipment. A diffuser ensures a unidirectional discharge perpendicular to the mounting surface. The airflow over the patient and through the room is thoroughly studied via CFD: the commercial FLUENT solves the RANS equations with a realizable k-ε turbulence model. The (thermal) boundary conditions applied to the CFD model are based upon a dynamic thermal calculation using TRNSYS.

The energy needs for these applications are enormous and therefore the equipment must be well designed, simulations are performed with TRNSYS. The maximum power is calculated both for summer and winter conditions. The considerate energy consumption pushes the design engineer in proposing advanced optimized solutions.

After determining the flow field, the assessment of particle deposition is executed. It is shown that the approach is meaningful with an acceptable computational effort, since the RANS solution allows for a safe prediction of the particle deposition.

Different injection clouds are simulated, each with a different diameter and with several repetitions to obtain a reliable average. An important part of the cloud is evacuated through neighbouring outlets, however some particles are dispersed and eventually escape through outlets on the other side of the room.

Paper No 153

NUMERICAL STUDY OF PHYSICAL BLOCKING' PERFRMANCE ON OCCUPANTS' EXPOSURE TO RESPIRATORY DROPLETS UNDER MIXING VENTILATION SYSTEM

Xiaoping Li, Jianlei Niu
The Hong Kong Polytechnic University, Kowloon, Hong Kong, China

The outbreaks of severe acute respiratory syndrome (SARS) in 2003 and H1N1 pandemic during 2009 and 2010 arouse researchers' attention to investigate the transmission of infectious disease in built environments. Research show that the spread of respiratory viruses indoor has certain relationship with ventilation system, and adopting some non-pharmaceutical approaches can interrupt or reduce the virus transmission. Therefore, in this paper, drift-flux model is used to numerically investigate the dispersion of expelled pathogen-laden droplets from the infector and the co-occupant's exposure when the infector has his/her mouth covered or one desk partition is employed under mixing ventilation (MV) system. The size of investigated droplet residuals is in the range from 1um to 10um since these droplets can reach the alveolar region of respiratory system and cause severe damage. The results show that both mouth covering and desk partition can mitigate the co-occupant's exposure by interrupting the horizontal travelling of the exhaled air, while the later airborne dispersion of exhaled droplets driven by air distribution system can still pose high risk to the occupants. Although these two interventions achieve almost the same inhalation for fine droplets, the inhalation of the co-occupant is lower when using mouth covering for larger droplets.

Paper No 94

EXPERIMENTAL STUDY ON VENTILATION EFFECTIVENESS IN A SMOKING ROOM WITH MIXING VENTILATION AND DISPLACEMENT VENTILATION

Yukihiro Hashimoto[1], Hidenori Takahashi[1]
[1]Polytechnic University, Sagamihara, Kanagawa Pref., Japan.

This paper compares influences of environmental tobacco smoke (ETS) in a smoking room with mixing ventilation and displacement ventilation. Exposure to ETS has been associated with adverse health effects. However, health for smokers in a smoking room is not clearly discussed to avoid ETS due to the other smokers. Ironically speaking, smokers are exposed to ETS derived from the other ones in a smoking room. Displacement ventilation is expected for a smoking room to decrease the horizontal influence of tobacco contaminants. Experiments are carried out to compare ventilation effectiveness with mixing ventilation and displacement ventilation. The experimental set-up room is dimensioned by 5.3m by 5.44m by 2.5m CH. Air change per hour is set to 6, 8 and 10 times per hour for both ventilations. This ventilation system does not recirculate air to the test room. Smokers are located at the center of the room and requested to smoke 2 cigarettes for each person during the test periods of 10 minutes. The concentrations of carbon dioxide and suspended particulates are measured during the tests at 1 minute intervals. ETS is usually evaluated by the concentrations of carbon monoxide and suspended particulates. However, it is much easier to evaluate the indoor air quality in a smoking room if ETS can be investigated by the carbon dioxide concentrations. ETS is derived from the combustion of cigarettes and includes carbon dioxide. The carbon dioxide sensors used in this study are non-dispersive infrared sensors (NDIR). As a result, the time-averaged carbon dioxide concentrations in the test room are correlated to the suspended particulate concentrations due to ETS, although carbon dioxide is emitted by the respiration of smokers.

Paper No 257

COMPUTATIONAL ANALYSIS OF THE PERFORMANCE OF PERSONALIZED VENTILATION SYSTEM IN CONJUNCTION WITH A PERSONALIZED EXHAUST

Junjing Yang, Chandra Sekhar
Nantional University of Singapore, Singapore.

A personalized exhaust (PE) device was introduced to work together with personalized ventilation system in this study. The performance of combining the two systems was compared with the performance of Personalized Ventilation alone. The primary aim of a PV system is to supply fresh air to the breathing zone to enhance thermal comfort and IAQ. At the same time, it can also be seen as a solution to prevent the spread of contaminated air, especially in the context of airborne infection control. Whilst a conventional PV system would fulfil most of these requirements, it may not be able to adequately prevent the spread of contaminated air as the PV air would go past an infected person and mix with the room air. The hypothesis is that by adding a personalized exhaust device at the upper part of a chair, just behind the human head, not only is the inhaled air quality improved further but the exhaled contaminated air is extracted locally and its spread into the room air is minimized. Computational fluid dynamics (CFD) was used in this study to simulate the indoor airflow in a typical office room with two seated human manikins at two workstations. Two different kinds of PV air terminal devices (ATD) were simulated with mixing ventilation or under floor air distribution (UFAD) system. The results indicate that there is a potential for improving inhaled air quality by adding in the personalized exhaust when background is ventilated by under-floor air distribution (UFAD) system. The Personalised Exposure Effectiveness (PEE), defined as the percentage of personalized air in inhaled air, can be increased more than 24% in this combination case. It also shows that this kind of personalized exhaust can prevent the spread of contaminated air by exhausting the exhaled air directly before it mixes with the room air. The Inhaled fraction will be one order of magnitude lower with 70l/s flow rate of PE than without PE.

Technical session 26, Theme: Natural, hybrid and mechanical ventilation and Ventilation for low energy, passive houses/zero emissions buildings, Room R1, Wednesday 2011-06-22, 10:45 to 12:15

Technical session 26 Natural, hybrid and mechanical ventilation and Ventilation for low energy, passive houses/zero emissions buildings

Paper No 54

ON THE POTENTIAL OF WIND-DRIVEN VENTILATION FOR DWELLINGS AND LOW RISE BUILDINGS

Xavier Faure[1] and Nicolas Le Roux[1]
[1]Centre Scientifique et Technique du Bâtiment (CSTB), Département Climatologie, Aérodynamique, Pollution, Epuration Nantes, France.

The potential of wind-driven ventilation (cross-ventilation) has been studied using wind tunnel facility and a calibrated holes approach. Circular holes have been chosen in order to represent power laws between volumetric flow rates and pressure differences, similar as for full scale ($Q=K.\Delta Pn$) for normal wind incidence (n=0.5). Discharge coefficients have been defined depending on the wind incidence (from 0° to 180°) using a calibration bench. These laws have then been used to identify volumetric flow rates through a classical dwelling of 84 m² modelled at a reduced length scale of 1/10. The number of holes was changed in order to represent openings of different size for outside openings as well as for internal doors. For each opening size, 18 wind incidences have been studied. Configurations were tested considering internal doors as closed and opened at different levels and also the presence or not of a chimney in the kitchen. The local pressure on more than 70 positions was recorded for each configuration. Using pressure coefficient values, the methodology can identify the potential of wind-driven ventilation on real site, using the meteorological temporal values of wind speed and direction. Results can also be used to identify what should be the minimum wind velocity to ensure sufficient air change rates within the building for each incidence. These two approaches have been done in this study for one geographic site in particular.

Paper No 22

EXPERIMENTAL ANALYSIS OF THE IMPACT OF ROOM/SYSTEM DESIGN ON NIGHT VENTILATION PERFORMANCE

Kim Goethals, Marc Delghust, Arnold Janssens
Ghent University, Ghent, Belgium.

Night ventilation attracts growing interest. However, predicting the nightly convective heat transfer proves to be the night ventilation Achilles' heel. Obviously, this heat transfer mechanism depends on the driving force, fluid motion and heat transfer surface and, thus, on the room and system design. Unfortunately, studies addressing this are scarce. In response, the underlying experimental effort intends to instigate global parametric analysis of night ventilation at room level. To this end, in a first stage envisaging steady-state boundary conditions, this study, held in an adapted PASLINK cell, comprises two typical mixed convection cooling regimes, for which two supply/exhaust locations are tested. The analysis is based on both airflow data, such as temperature and velocity, and derived variables related to energy, in particular the convective heat flux. The results clearly indicate the need for an integrated room/system design. After all, the positioning of the supply location relative to thermally massive elements predominates the night ventilation efficiency.

Technical session 26, Theme: Natural, hybrid and mechanical ventilation and Ventilation for low energy, passive houses/zero emissions buildings, Room R1, Wednesday 2011-06-22, 10:45 to 12:15

Paper No 124

CONTROL STRATEGIES FOR EFFECTIVE USE OF WIND LOADING THROUGH A DECENTRALIZED VENTILATION SYSTEM

Luca Baldini[1], Philippe Goffin[1], Hansjürg Leibundgut[1]
[1]Building Systems Group, Department of Architecture, ETH Zurich, Switzerland.

This paper introduces a dynamical model to assess control strategies for a ventilation system comprising a large number of decentralized supply units feeding an interlaced ducting network. Results for an exemplary control strategy based on local pressure difference are presented, applying the derived model.

Due to multiple air intakes in the façade, decentralized ventilation systems are strongly influenced by façade wind loading. As shown in earlier studies, properly designed ducting networks can balance pressure differences due to wind and lead to a regular distribution of supply air in the building. Additionally, using fan over-capacity along with flexible distribution of the fan power among supply units, the total fan power can be reduced. To use this saving potential an appropriate supply unit control strategy is required, which needs to be identified with the help of a dynamical model. A simplified two-plenum model to describe the ventilation system with its ducting network was chosen. It provided an appropriate base for testing local control strategies. It was found that a pure local pressure control strategy did lead to the desired fan power distribution but did not manage to maintain comfortable room conditions. With an additional error correction term the local strategy could be improved.

Paper No 263

EVALUATION OF APPLICABILITY OF GLAZING SYSTEM USED DYNAMIC INSULATION FOR DWELLING HOUSES

Sihwan Lee[1*], Miho Tanaka[1], Shinsuke Kato[2]
[1] Department of Architecture, The University of Tokyo, Tokyo, Japan
[2] Institute of Industrial Science, The University of Tokyo, Tokyo, Japan

In this paper, a new dynamic insulation system applied to the glass and frame of the windows is proposed. This system is composed of three parts: a double pane airflow window system with window frame made of a porous material, a mechanical ventilation system, and a heat-recovery heat pump system. The aim of this paper is to evaluate the thermal insulation efficiency and probability of moisture condensation in the proposed system in order to confirm its feasibility and applicability. First, a double pane airflow window system with porous material was designed to ventilate through the window frame and vent layer of double pane window. Then, to verify its thermal insulation efficiency, the temperature distribution of the window system was evaluated using computer fluid dynamics with different coupled conditions, such as the indoor/outdoor pressure difference and outdoor temperature. In addition, to verify the probability of moisture condensation, the relative humidity in the window system was calculated based on the various conditions. The calculated results show the thermal load was proportional to the outdoor temperature and inversely proportional to the indoor/outdoor pressure difference. Moisture condensation depends on the outdoor temperature and humidity ratio and it does not occurred when outdoor temperature is more than 6.0 ºC in the proposed system. Therefore, the proposed system is technically feasible to reduce the home energy consumption by installed dwelling houses.

Paper No 203

VENTILATION AND HEATING STRATEGIES FOR A PASSIVE HOUSE IN MEDITERRANEAN CLIMATES

Gonçalo Brites, Renato Hauer, José Costa, Adélio Gaspar
ADAI–LAETA, Dept. Mechanical Engineering, University of Coimbra, Portugal

This paper presents a preliminary study about the thermal behaviour of a low energy consumption house to be built in the centre of Portugal under the Passivhaus concept. The strategies adopted consist of a strongly insulated building envelope, double glass panes with wood frames, window shadowing to avoid overheating in summer, and a very tight envelope with low infiltration airflow. The thermal insulation is made of natural cork, and is placed in the outer surface of the walls, in order to increase the thermal mass of the house. A mechanical ventilation system equipped with heat recovery exchanger guarantees good indoor air quality, and in the summer provides free cooling when outdoor air temperature is lower than indoors.

In winter, the heat loss caused by the ventilation presents the most significant factor of heating demands, which may be reduced by the mechanically-controlled ventilation system with high efficiency heat recovery. In the same sense, the project of a passive house should also maximize the solar gains through glazing, in the winter. In summer the main goal is to obtain the required internal comfort, without using any active cooling system, which may be achieved by combining a strong thermal inertia with free-cooling whenever possible. The passive house should also be equipped with carefully designed shading systems that permit a significant reduction of the solar gains in the summer. Using thermal simulation software, the main goal of the present study is to demonstrate that the strategies above permit to obtain internal comfort both in winter and summer with very low energy consumption. For the winter simulations we want to maximize solar gains, and a biomass burner will be considered for air heating, with heat recovery in the ventilation system. The results will show that the mechanically controlled ventilation with heat recovery allows a significant reduction of the heating demands, preserving good indoor comfort and air renewal.

Technical session 27, Theme: Modelling and visualization, Case studies and New technologies for heating and cooling or ventilation/AC, Room R2, Wednesday 2011-06-22, 10:45 to 12:15

Technical session 27 Modelling and visualization, Case studies and New technologies for heating and cooling or ventilation/AC

Paper No 317

CFD ANALYSIS AND AIRFLOW MEASUREMENTS TO APPROACH LARGE INDUSTRIAL HALLS ENERGY EFFICIENCY: A CASE STUDY OF A CARDBOARD MILL HALL

Nikola Tanasić[1], Goran Jankes[1], Håkon Skistad[2]
[1]Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia.
[2]SINTEF, Trondheim, Norway.

This paper deals with numerical methods for predicting air flow patterns in large industrial halls. Some major findings of the investigation of the airflow patterns in paper machine hall of Umka Cardboard Mill are presented in the paper. The main reason for the interest in this problem is to find optimal locations for extract air intake connections of the ventilation system connected to the exhausted air waste heat utilization. Previous studies have shown that there was a big difference in air temperatures from the floor to the ceiling of the hall (30-350C). Moreover, an amount of heat released from the cardboard machine to the surrounding air in the hall and extracted by the series of ceiling mount axial fans was almost 30% of the total waste heat from the paper machine's drying section. These results have indicated the need for the waste heat utilization, but also for the optimization of the ventilation system. CFD simulation for predicting of air flow patterns was applied. The accuracy of the simulation was evaluated by comparing its results with the results of field measurements. The simulation results have shown considerable discrepancy compared to the measured values. However, results served well for qualitative analysis, gave better insight in general air movements inside the hall and indicated the extract air intake locations. By utilizing the waste heat from proposed optimal locations, fuel savings of 5% and reduction of 1,140 t/year in CO2 emissions can be achieved.

Paper No 193

JUSTIFICATION OF AIR CHANGE REDUCTION USING NUMERICAL SIMULATION METHODS

Daria Denisikhina[1,2], Maria Lukanina[1,2], Mikhail Samoletov[1], Alexandra Zuzo[1], Sergey Denisikhin[3]
[1]JSC "TECHNIQUE BUREAU. Design", St.Petersburg, Russia.
[2]Saint-Petersburg State University of Architecture and Civil Engineering (SPSUACE)
[3]Baltic State Technical University

The present work discusses numerical simulations of ventilation air flow in the stadium bowl. Existing conventional techniques cannot provide reliable information regarding variations of CO2 concentration in a domain, which opens a niche for using computational fluid dynamics (CFD). It allows for accurate prediction of velocity fields,temperature and CO2 concentration distributions for the domains with complex geometry.

This work shows that CFD methods allows significantly reduce the designed outside air flowrate for the new big stadium (62000 spectators) with roof in St.Petersburg, Russia. The computations have been done in ANSYS FLUENT package. The 3D Unsteady Reynolds Averaged Navier-Stokes equations in conjunction with non-linear quadratic k-ε turbulence model were used for numerical modeling of the air flow. We have considered two modes of the stadium bowl operation: designed flow rate of 1 240 000 m3/h and 600 000 m3/h.

Paper No 270

Technical session 27, Theme: Modelling and visualization, Case studies and New technologies for heating and cooling or ventilation/AC, Room R2, Wednesday 2011-06-22, 10:45 to 12:15

REDUCTION EFFECT OF STRIP TYPE SUNSHADE SHEETS OVER ROOF ON AIR-CONDITIONING LOAD

Eunsu Lim[1], Toshio Yamanaka[2], Hisashi Kotani[2], Yoshihisa Momoi[2] and Masako Soma[2]
[1]Kyushu University, Osaka, Japan.
[2]Osaka University, Osaka, Japan.

Direct solar radiation is the greatest influencing factor of cooling load in hot and humid area in summer. The surface of metallic roof rises up to a high temperature caused by direct solar radiation. The heat flux through roof and ceiling become air-conditioning load. Therefore, it is important to decrease the surface temperature of roof for reducing the cooling load in the building with the metallic roof. The shielding method of direct solar radiation by sunshade sheets is widely used recently. It is thought that a sunshade sheet is effective to decrease the surface temperature of roof, but the reduction effect of sunshade sheets on air-conditioning load is not clarified yet.

The purpose of this study is to clarify the reduction effect on cooling load by strip type sunshade sheets settled over folded-plate roof. The measurement and calculation were conducted on a small house with metallic folded-plate roof in summer. Two conditions were tested; condition without sunshade sheets over roof and condition with sunshade sheets over roof.

We proposed a heat transfer model that predicts the heat flux through roof with sunshade sheets and ceiling. The solar transmittance, reflectance and absorptivity were used as properties of sunshade sheet. To verify the validity of this model, the calculated values were compared with measured values. In general, calculated values are in good agreement with measured values.

It is clarified that using a sunshade sheet can decrease air-conditioning load at daytime in summer from results of measurement and calculation. However, condition with sunshade sheet has slightly a worse hand at nighttime in summer.

Paper No 202

CFD MODELLING OF AERODYNAMIC SEALING BY VERTICAL AIR CURTAINS

João Gonçalves[1, 2], José Costa[2], A. Rui Figueiredo[2], António Lopes[2], Manuel G. Silva[2]
[1]CI&DETS, ESAV-Polytechnic Institute of Viseu, Viseu, Portugal.
[2]ADAI-LAETA, Dept. Mech. Engineering, University of Coimbra, Coimbra, Portugal.

This work presents a comparative study of the sealing effect of doorways of air-conditioned or refrigerated spaces, obtained by vertical air curtain devices (ACD). An analysis is made of the effect of installing the device(s) inside, outside or both sides of the door, considering both winter and summer operation conditions.

For this purpose, a numerical model was developed aiming the simulation of the turbulent nonisothermal

3D airflow generated by the air curtain after the door opening. The flow field induced by the opening of the door with the ACD turned off was also simulated and taken as a reference to assess the efficiency of the aerodynamic sealing process.

The numerical simulations have shown plausible results which comply with the physical interpretation of the convective phenomena involved. In line with previous investigations on air curtains, an optimum velocity of the air jet was found, that corresponds to the highest sealing effect. According to the simulations results, downward blowing air curtains can present high sealing efficiencies (over 70%). Although the direct air recirculation provides a better sealing performance (over 80%), the complexity, acquisition and maintenance costs inhibit its common use in ACD installations.

Technical session 27, Theme: Modelling and visualization, Case studies and New technologies for heating and cooling or ventilation/AC, Room R2, Wednesday 2011-06-22, 10:45 to 12:15

Paper No 93

A NEW SYSTEM TYPE AND AIR FLOW ORGANIZATION FOR LARGE SPACE BUILDING

Zhang Lun[1] , Liu Xiaohua, Jiang Yi
[1]Tsinghua University, Beijing, China.

Large space buildings are attracting more attention of researchers in recent years. The characters of large space heat sources are studied in this paper by measurement of railway stations. And infiltration air is an important influencing factor in large space buildings according to the result of measurement. Then a new system type is proposed in this paper corresponding to the special thermal environment of large space building. The system applies the temperature and humidity independent control system, which means the latent load is removed by dehumidification devices supplying enough dried outdoor air, and the cooling load is controlled by high temperature cooling devices. Radiant floor is applied in the system to absorb radiant heat and convective heat with the supply water temperature of 18ºC. The whole large space is divided into three regions: low humidity zone with conditioned temperature, middle humidity zone with conditioned temperature and natural ventilation zone with high temperature. A simulation model of large space building is established and verified based on a real building in order to understand the effectiveness of the new system. The effectiveness of the new system is studied by the comparison with jet ventilation. The speed and the air flow rate of the new system are lower than jet ventilation, and radiant floor is applied, therefore the transportation energy consumption is much lower in the proposed system. Heat and humid brought in by infiltration air is exhausted through upper ventilation. The humidity above the floor is lower than the dew point humidity ratio.

Paper No 226

POTENTIAL ENERGY SAVINGS WITH PERSONALIZED VENTILATION COUPLED WITH PASSIVE CHILLED BEAMS

Velina S Lyubenova[1], Jan W Holsøe[2], Arsen K Melikov[1]
[1]Technical University of Denmark,Kgs. Lyngby, Denmark.
[2]HaCaFrø A/S, Herlev, Denmark.

Personalized ventilation (PV) is an individually controlled air distribution system aimed at improving inhaled air quality and thermal comfort of each occupant. Numerous studies have shown that PV may improve occupants' health, comfort and performance in comparison with traditional total volume air distribution used today. The potential of PV for energy saving has been studied little.

In this study, the energy saving potential of desk mounted PV in conjunction with either mixing ventilation or a passive chilled beam system is compared to mixing ventilation alone by means of computer simulations. An open plan office in a building, located in a cold and dry climate was simulated. Numerous simulations were performed to study the importance of number of room occupants, occupancy profile in time and room air temperature control. The requirements for indoor environment in office buildings as defined in the present standards were considered.

The most effective energy saving strategy with PV in use was to expand the upper room temperature limit as defined in the present standards. When PV was coupled with background mixing ventilation, the possible reduction of the air supplied to the room was approximately 20% (and up to 40% when extending the temperature in the room by 2 °C above the upper limit recommended in the standards) compared to mixing ventilation only. When PV was combined with passive chilled beams, the reduction of the supplied air was up to 80%. This ventiltion strategy may lead to energy saving especially in spaces where occupants spend most of the time at their workplace.

Paper No 239

EFFECTIVENESS OF ROOM VENTILATION AND HEATING IN NEW ZEALAND DOMESTIC HOUSING

Sarah Wakes[1]
[1]University of Otago, Dunedin, New Zealand.

New Zealand has a mostly old domestic housing stock that is substandard in terms of ventilation, heating and insulation. This has led to health issues, inefficient heating and high heating costs. A common strategy is to have a heating source in the main living area and minimal heating within the rest of the house, usually to save money. Temperatures within houses are therefore often under the recommended WHO minimums. Consequently the lack of air movement from adequate ventilation and heating can lead to condensation and damp issues which exasperates health issues such as asthma, especially in children. There is still a public perception in New Zealand that insulation, double glazing and more evenly distributed temperatures within a house are of limited benefit to well being and saving money. The objectives of this work are:

- To investigate the effectiveness of some common heating, ventilation and insulation strategies in New Zealand's South Island.
- To quantify the effectiveness of these strategies on energy efficiency and performance.

Computational Fluid Dynamics (CFD) is used to model and compare different heating and insulation strategies within a standard old New Zealand house as part of an ongoing project. It was found that there are definite limitations in current practices and better strategies can be developed using CFD modelling techniques. Relatively simple strategies are trialled to improve the house's performance in terms of energy efficiency.

The current strategies for heating, insulation and ventilation in the majority of New Zealand homes are not the most efficient in terms of energy or cost. There is more work needed in understanding the benefits of ventilation, insulation and more distributed heating within such old houses. CFD therefore allowed a range of scenarios to be modelled and compared in a relatively cost effective way in order to develop understanding and better strategies.

Workshop 1 Ventilation of heavy industry

Chairperson: Håkon Skistad

Industrial ventilation requires thorough knowledge of fluid mechanics, insight into the production processes in the ventilated rooms and knowledge of building construction. CFD is a valuable tool in the analyses of industrial ventilation problems, and analytical methods must also be used to understand the phenomena that occur.

However, real life turns out to be more complex and non-stationary than can be modelled in CFD or predicted by analytical methods. This workshop focuses on what happens in real life. Some topics that will be covered are:

- Air flows in industrial premises with large heat sources and thermally stratified flow.
- Air balance – what comes in and what goes out?
- Measuring air flow rates and contaminant emissions in practice?
- Some experience in contaminant control in furnace rooms.

Presenters:

- Helge Midtdal, SINTEF, Norway – “Measurement of plumes above a metal ladle”
- Arild Haugene, Multiconsult AS, Norway – “Contaminant control in ferro alloy smelter rooms”
- Nikola Tanasic, University of Belgrade, Serbia – “Air movement in cardboard factory”
- Christer Johansson, Molab, Mo i Rana, Norway – “Measurement of air flow and contamination in a large area stack”

Workshop 2 Ventilation in Zero Emission Buildings

Chairperson: Terje Jacobsen, SINTEF, Norway

The aim of this workshop is to raise questions about possible problems, to synthesize the state-of-the-art of the research or the practice and to point out research needs for efficient ventilation in super-insulated envelopes, as for passive houses, Zero Emission Buildings or Zero Energy Buildings. More specifically, the objective is to discuss the present or future challenges in terms energy efficiency, thermal comfort as well as indoor air quality. Some topics that will be covered during the workshop are:

- Concerns in regards to space-heating by ventilation air.
- Efficiency of the ventilation components in high-performance buildings. For example, is the consumption compatible with ZEB buildings and the electricity production on-site? What about natural ventilation to reduce the electricity consumption and mechanical cooling demand?
- Will the ventilation strategy or challenges for ZEB buildings be different than for passive houses?
- The role of active façade elements (related to ventilation) in energy reductions, as double-skin façades.
- Concerns about ventilation components reliability in airtight envelopes. For example, do we need to develop safer, more reliable components to ensure sufficient indoor air quality?
- Health concerns in airtight envelopes: quality of the air, development of bacteria, and diffusion of pollutants or humidity aspects within the building envelope.

Presenters:

- Leon Glickman, MIT, USA –
- Jan Vilhelm Bakke, Norwegian Labour Inspection Authority, Norway – “Health and indoor air quality issues”
- Rasmus Høseggen, Evotek AS/NTNU, Norway

Workshop 3 Advanced methods for air distribution in spaces

Chairperson: Arsen Melikov, Technical University of Denmark, Denmark
Scientific secretary: H. Ezzat Khalifa, Syracuse University, USA

It has been shown that poor air quality affects occupants' health, comfort and performance. Present total volume air distribution methods are inefficient because clean and cool air is supplied to spaces far from occupants. Thus the clean air is more of less mixed with the room air and thus gets polluted and warm by the time it reaches the occupants. Thus occupants do not inhale clean air. The cooling power of the supplied air is inefficiently used because the entire space, including unoccupied zones is conditioned. Furthermore the total volume air distribution principles used today aim to provide a uniform room environment. However, large individual differences exist between occupants with regard to physiological and psychological response, clothing insulation, activity, air temperature and air movement preference, etc.

There is need for development of advanced methods for efficient supply air distribution, which will ensure 100% clean air in inhalation. Furthermore providing occupants with control of the supplied air in order to achieve preferred microenvironment is essential. Personalized ventilation (PV) aims to supply clean air directly to the breathing zone of room occupants and thus improves inhaled air quality resulting in decrease complaints with Sick Building Syndrome (SBS) symptoms. The provided control over flow rate, i.e. target velocity, temperature and direction of the supplied personalized air may significantly improve occupants' thermal comfort and perceived air quality. Thus occupants' performance may increase.

The objective of the workshop is to discuss recent developments on the advance methods for ventilation of spaces, including office buildings, hospitals, vehicle compartments, etc. The performance of the methods with regard to human response, energy use and implementation in practice will be in the focus of the discussion. The workshop will help researchers in further development of advanced methods for efficient air distribution which will ensure high quality of inhaled air and preferred thermal environment. The workshop will also help designers and consultants to develop proper strategies for application of personalized ventilation in practice.

Presenters:

- Chandra Sekhar, National University of Singapore, Singapore – "Integrated Personalised Ventilation and Personalised Exhaust System"
- Peter Nielsen, Aalborg University, Denmark – "Air Distribution and Personal Environmental Control in Aircraft Cabins"
- Xiaoliang Shao, Tsinghua University, China – "Multi-mode Ventilation for Demand Controlled Air Distribution"
- Shinsuke Kato, University of Tokyo, Japan – "Space Distribution of Ventilation Efficiency Indices with CFD and Tracer Distribution Calculation"
- Ez Khalifa, Syracuse University, USA – "PV effectiveness in the presence of indoor chemical reactions"
- Arsen Melikov, Technical University of Denmark, Denmark – "Advanced air distribution in hospitals"

Workshop 4 Occupant's behavior related to ventilation energy use – IEA/ECBCS Annex 53

Chairperson: Hiroshi Yoshino

This workshop will deal with occupant's behavior related to ventilation and discuss on energy consumption.

One of the most significant barriers for achieving the goal of substantially improving energy efficiency of buildings is the lack of knowledge about the factors determining the real energy use. There is often a significant discrepancy between the designed and the real total energy use in buildings, in which a complex array of factors play a significant role, including the user/occupant behavior. The reasons for this discrepancy are generally poorly understood, and often have more to do with the role of human behavior than the building design.

For that, the IEA/ECBCS/Annex entitled as "Total Energy use in Buildings -Analysis and evaluation methods-" was initiated in the last November and started on first of January, 2010 as working phase. The ultimate outcome of this annex is to strengthen the robust prediction of energy usage in buildings, thus enabling the proper assessment of short- and long-term energy measures, policies, and technologies.

Presenters:

- Hiroshi Yoshino - "Overview of Annex 53 research works"
- Rune V. Andersen, Stefano P. Corgnati, Valentina Fabi, Marco Filippi, Bjarne W. Olesen - "Occupant behaviour, ventilation through window opening and energy consumption"
- Motoya Hayashi, Yoshinori Honma, Masanori Sugawara and Hiroshi Yoshino - "A study on prediction of indoor air quality and energy consumption in houses, Simulation method using result of measurement on dweller's opening behavior"
- Mohamed El Mankibi - "Ventilation control strategies and occupancy pattern: An Experimental investigation"
- Takao Sawachi and Hisashi Miura - "Actual energy efficiency of room air conditioners for residential buildings -Bridging the gap between the existing standardized index and the actual energy efficiency of the equipment"
- Guangyu Cao, Jorma Heikkinen, Helena Järnström - "Protection of office workers from exposure to respiratory diseases by a novel ventilation system"

Workshop 5 Air motions and air exchange in Historical Buildings

Chairperson: Mats Sandberg KTH Research School, Sweeden

Air motions and air exchange in historic buildings have an important influence on the indoor climate, both in terms of comfort and preservation. In order to facilitate a sustainable use and preservation of these buildings, in times of rising energy costs and climate change, we need a better understanding of ventilation and control of air motions in historic buildings.

Historic buildings are often complex buildings with unique architectural features. They may be very large as e.g. cathedrals or very complex buildings provided with, towers, cupolas and many rooms. They are naturally ventilated and in cases when heating is required they are often intermittently heated. Often the historic buildings contain collections of unique and valuable collections of artifacts. The conservation of these objects puts a constraint on the indoor climate which can be in conflict with people's requirements on thermal comfort.

In old buildings with humidity problems ventilation is often considered as the natural remedy. However the effect of ventilation depends on the outdoor climate, temperature and absolute humidity, in relation the indoor climate. Furthermore a high air exchange will decrease the stability of the indoor climate, an important factor for preservation.

Air motions inside these buildings are due to complex driving forces, not well understood. Air motions will increase the risk of soiling of surfaces and artifacts by particles generated by candles.

The purpose of this workshop is to first give an overview of the field followed by short presentation of results from studies and presentation of new measuring techniques. This will be the starting points for a discussion forum where all the participants can give their input and identify research needs.

Presenters:

- Professor Mats Sandberg, KTH Research School, Sweeden
- Professor Tor Bröström, University of Gotland, Sweeden