

**12.**

**VENTILATION SCIENCE TECHNOLOGY — OPPORTUNITIES FOR CROSS CUTTING TECHNOLOGY (INDUSTRIAL-MINE VENTILATION).** H. Goodfellow, Techint Goodfellow Technologies Inc., Mississauga, ON, Canada; E. Tähti, Vantaa Energy Ltd., Vantaa, Finland; J. Stachulak, INCO, Sudbury, ON, Canada.

This paper will outline the recent progress on the ventilation science technology field based on more than 50 million dollars of research and development in the industrial ventilation field. Current air quality issues (i.e., diesel exhaust emissions, heat stress, etc.) in mine ventilation will be outlined. Specific examples will be given of possible areas of technology transfer and future areas of cooperation between the industrial ventilation and mine ventilation communities.

### Podium Session B2: CFD and R&D

*Papers 13–23*

**13.**

**CFD MODEL FOR A MULTINOZZLE CEILING AIR DIFFUSER.** H. Koskela, P. Saarinen, Finnish Institute of Occupational Health, Turku, Finland; J. Heikkinen, VTT Technical Research Centre of Finland, Espoo, Finland; E. Sandberg, Satakunta Polytechnic, Pori, Finland.

CFD-modeling provides a method for predicting the airflow pattern, thermal comfort, and contaminant distribution in a space. The application of the method to practical ventilation problems has constantly increased, together with the development of the modeling software and the processing power of computers. An important issue for the applicability of CFD-modeling is the availability of proper boundary conditions for the objects in the room, especially for air diffusers. For an individual modeler, it is usually not possible to determine reliable boundary conditions for each diffuser type in a modeled space. Therefore, validated boundary conditions should be made available for modelers. This study presents simplified boundary conditions for a square multinozzle ceiling air diffuser. The model is a so-called slot model, where the correct momentum of the supply airflow is adjusted by changing the dimensions of the inlet opening. The boundary conditions were determined based on jet flow measurements in the near zone of the diffuser. The measurements were carried out in a laboratory test room with an automatic traversing system and an ultrasonic velocity sensor which is able to detect all three velocity components in the jet. The slot model was created by dividing each side of the diffuser to six parts and giving different velocity boundary conditions for each part. The model was validated in the laboratory room in nonisothermal test cases with four different Archimedes numbers for the supply airflow. Heat was generated in the room with a

floor heating system. The model correctly predicted the flow pattern in all test cases. The selection of turbulence model was found to have significant effect on the spreading of the supply jet. Detailed flow measurements in the near zone were found to be necessary for determining the boundary conditions for diffusers with a complex airflow pattern.

**14.**

**USE OF COMPUTATIONAL FLUID DYNAMICS FOR PREDICTING AIRFLOW AND CONTAMINANT MASS TRANSPORT WITHIN A LABORATORY ENVIRONMENT.** A. Kolesnikov, CD-adapco, Melville, NY; J. Baker, M. Grubert, S. Ericson, The University of Tennessee, Knoxville, TN;

J. Bennett, NIOSH, Cincinnati, OH.

Many common laboratory handling practices such as weighing, pipetting, container transfer, autoclaving, and incubating can introduce aerosols, gases, and vapors into the inhabited environment. In addition, laboratory use of high-throughput technology with hazardous solvents (e.g., DMSO, methanol), biological agents (e.g., HIV, TB, hepatitis), and novel compounds of unknown potency (e.g., drugs), is rapidly expanding. Increased safety awareness leads to steady refinement of regulations emphasizing the need to accurately perform risk assessment studies of human exposure to airborne contaminants in cases of their accidental or intentional release into an indoor occupied space. Experimental tests are costly and provide only limited data. The goal of this study is to explore, hence validate the use of computational fluid dynamics (CFD) for generating reliable predictions of laboratory airflow patterns hence contaminant mass transport distributions. Laboratory contaminant mass transport is dictated by room airflow patterns and contaminant source momentum. Mechanical ventilation (including the buoyancy effects of heating/cooling), location and characteristics of contaminant sources, occupancy, and lab equipment determine the contaminant concentration field. Quantitative inhalation exposure assessment therefore requires detailed assessment of the indoor ventilation velocity vector distribution. The presentation is inaugurated via CFD prediction for established experimental room airflow benchmarks describing supply, forced and mixed ventilation scenarios. The availability of quality experimental data for each case enables detailed validation studies to assess the impact of different turbulence closure models, numerical dissipation and domain discretization density. Following, airflow and contaminant mass transport CFD experiments are detailed for a current NIOSH experimental laboratory environment. Pollutant source, ventilation supply/exhaust and equipment locations are all shown to play important roles in the resultant contaminant distributions within the floor plan. The impact of approximations made in forming the CFD models is specifically addressed, to quantitatively assess the associated error mechanisms in prediction fidelity.

**15.**

**INVERSE PREDICTION OF CONTAMINANTS IN A ROOM.** Q. Chen, T. Zhang, Purdue University, West Lafayette, IN.

Computational fluid dynamics (CFD) has been widely used to study how contaminants are transported from one building location to another together with airflow. In practice it is also important to know where the contaminant sources are if we know the airflow pattern and contaminant concentration distributions in a building. This presentation shows an inversed CFD method for the identification of the contaminant sources. By inverting the CFD calculations, the numerical procedure becomes unstable and the convergence is a major problem. This presentation imposes a remedy method to solve the numerical instability. With the new method, this presentation will show, through a few demonstration cases, that it is possible to identify contaminant source locations if the corresponding airflow patterns and the contaminant distributions are available.

**16.**

**DEVELOPMENT OF VENTILATION STRATEGY IN DIESEL ENGINE POWER PLANT BY USING CFD MODELING.**

P. Mustakallio, R. Kosonen, Halton Oy, Kausala, Finland.

The main design target for the power plant hall ventilation is to keep the difference in the engine level temperature and the ambient air below 10 Kelvin, and to keep the temperature of hall air cooling the generator as low as possible. Nowadays, the systems are based on mixing ventilation and relatively high airflow rate. The studied system uses big fans supplying the air straight to the engine hall and mixing it well. The design targets are met but energy consumption for the ventilation is quite high, which is directly from the generated electricity. Novel ventilation strategy using displacement ventilation was studied in two-diesel engine power plant cases. New ventilation strategy was developed for power plant module. Diesel engine power plant with existing and new ventilation systems was modelled by using computational fluid dynamics (CFD) tool. Periodic boundary conditions were used on the both sides of the module to model the multiengine power plant. In the most workable configuration, air is supplied through the low-velocity units from both ends of the engine hall, and cooling air circulated through the generator is directed straight to the exhaust opening in the ceiling. This makes it possible to reduce the supply airflow rate 30% from the current setup and distributes the supply air more uniformly.

**17.**

**DESIGN AND DEVELOPMENT OF A LOW-FLOW, ENERGY EFFICIENT FUME HOOD USING ENGINEERING CONTROLS AND COMPUTATIONAL FLUID DYNAMICS (CFD).** S. Kotha, R. Ryan, Flowsciences, Leland, NC; D. Walters, KCP, Inc, Raleigh, NC.

Fume hoods, also known as laboratory chemical hoods, are one of the most important and

# Reaching New Heights

---

## *Abstract Book* **AIHce & VENT**

*May 13–18, 2006*  
*Chicago, Illinois*

### **NIOSH LIBRARY SYSTEM**

**ALICE HAMILTON LIBRARY  
4676 COLUMBIA PARKWAY  
CINCINNATI, OH 45226**



*Co-sponsored by AIHA and ACGIH®*

[www.aiha.org/aihce.htm](http://www.aiha.org/aihce.htm)