CFD CALCULATIONS AND MEASUREMENTS OF NIGHT COOLING BY NATURAL VENTILATION

F.G.H. Koene[1], R. Schuitema[1], M. Houkema[2]

[1]Energy Research Centre of the Netherlands ECN, P.O. Box 1, NL 1755 ZG Petten, The Netherlands

[2]NRG, Westerduinweg 3, P.O. Box 25, NL 1755 ZG Petten, The Netherlands

Fax +31-224-568966, email: koene@ecn.nl

Key words: Night cooling, natural ventilation, CFD-calculations, research facility

ABSTRACT

CFD calculations on night cooling by natural ventilation have been carried out for two different opening geometries: 1) a set of panels and 2) a 'parallel window'.

Ventilation rates for these geometries have also been measured in a research facility. A considerable amount of scatter is found in the measurements, due to limitations of the experimental set-up. These are discussed as well as limitations of CFD calculations

Both measurements and calculations show a general trend of increasing ventilation rate with increasing temperature difference between indoor and outdoor climate, as expected.

In spite of the relatively small opening of the parallel window (3cm all around), ventilation rates measured are only about 30% lower than those found with the panels. Apparently, this window design is quite effective in combining inlet and outlet opening into a single component, simplifying the actions needed for night cooling (manual or automated).

INTRODUCTION

Average temperatures on earth have been rising over the last decades. Other trends are improved thermal insulation and air tightness of dwellings and increasing domestic consumption of electricity (most of which is dissipated into heat inside the dwelling). The combined effect of these trends will lead to higher indoor temperatures, which in turn is expected to increase the demand for domestic cooling.

An energy efficient way to cool dwellings is night cooling by natural ventilation, where the dwelling is flushed with cool outside air by opening one or more windows. As a result, the building mass will cool down, and will be able to absorb heat during the next day. Night cooling has been studied by a number of authors (e.g. Pfafferott et al., 2003).

In this paper, the effect of night cooling is studied by comparing the results of CFD (Computational Fluid Dynamics) simulations with measurements in a research facility.

CFD simulations hold the promise of offering a much faster means to assess the performance of different window geometries than by actually building and testing them. However, the use of CFD calculations is not without risk.

Loomans (Loomans, 1998) discusses measurements and simulations of indoor air flow in his Ph.D. thesis. He concludes that in the simulations, particular care must be taken with heat transfer between air and walls. Since then, the subject has been treated in several studies (e.g. Awbi, 1998).

Choudhary (Choudhary et al., 2001) also discusses the combination of measurements and simulations of indoor air flow. The main difference between the present study and Choudhary's is that the present research facility is primarily intended for field tests, while Choudhary's experimental rig is a well-controlled laboratory set-up. Even then, Choudhary recommends a semi-empirical approach to optimise the agreement between CFD calculations and measurements.

FAÇADE OPENING GEOMETRIES

The type of ventilation investigated is single-sided ventilation. That means that only one façade in the room contains one or more openings. For sufficiently high ventilation rates it is generally assumed that there be at least two openings in the façade.

This is the case in one of the geometries studied: a façade containing two panels that open simultaneously, as shown in figure 1.

The second façade studied contains a window of a novel design. When opened, the window remains parallel to the façade instead of swinging from hinges, as shown in figures 2 and 3. Because of its size, this window combines inlet and outlet opening into a single component. It is hitherto referred to as the 'parallel window'.

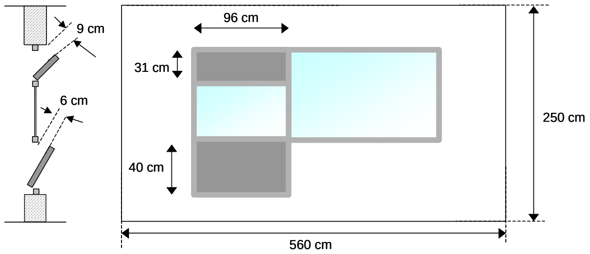

Figure 1 Cross section and front view of the façade with the two panels (shown in grey), opening to the inside.

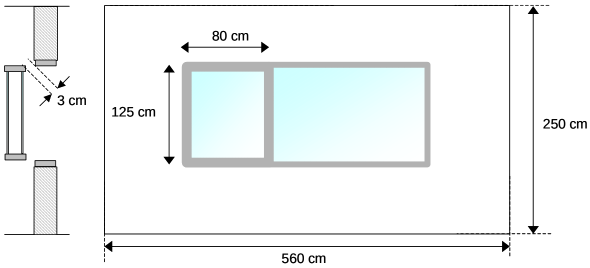

Figure 2 Cross section and front view of the façade with the parallel window, opening to the outside.

Figure 3 Picture of the parallel window, partly opened.

EXPERIMENT

The façades are integrated in a research facility, shown in Figure 4. The research facility contains four rooms, each of which is 3.8m deep. The façades, measuring 5.6m wide and 2.5m high, form the outer walls of the rooms. Measurements are carried out in all four rooms, but only the two facades previously described are modelled using the CFD program.

For ease of operation, the rooms are electrically heated, allowing accurate control of indoor temperatures and accurate monitoring of energy consumption.

Figure 4 Research facility containing four rooms each holding a different façade. The bottom room on the right contains the parallel window; the bottom room on the left contains the panels.

Temperatures of air, walls, floor etc., energy consumption as well as weather data (ambient temperature, solar irradiation and wind speed) are gathered every 10 minutes using a PC-based data-acquisition system. Figure 5 shows the floor plan of a room with the location of sensors. Ambient temperature and wind speed are measured on a nearby weather station at a height of 10m.

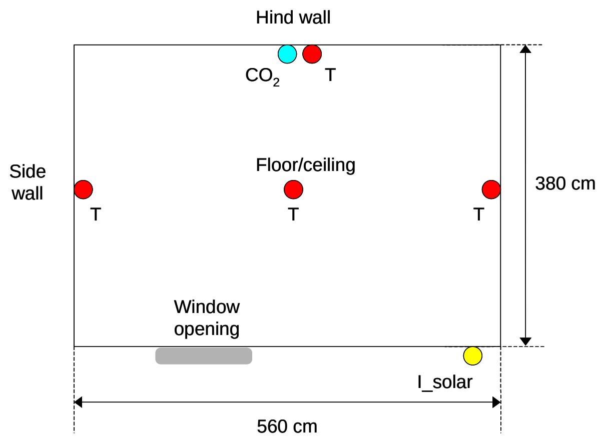

Figure 5 Floor plan of a room with the location of sensors. Sensors on the hind wall and side walls are placed at a height of 1.5m.

The computer also simulates occupation of the rooms by controlling the indoor temperature set point and injecting CO_2 as a tracer gas. The CO_2 is injected in the middle of the room at a height of approx. 2m.

During the day, the rooms are heated to a temperature of 25°C. At 11:00 p.m. the computer switches off the heaters and automatically opens the panels and windows in the façades, allowing cool outside air to enter the room. At 6:00 a.m. the following day the computer closes the openings and heats the rooms again to 25°C. This pattern is repeated for a number of days.

During the night, CO_2 is injected in each room using a flow controller and control valves. The injection of CO_2 is not continuous as the valve to each room is opened for 100 seconds every 10 minutes. The average injection rate is $0.015Nm^3/hr$. Gas velocity at the injection point is approx. 0.3m/s.

In a (quasi) equilibrium situation, the rate of CO_2 injected equals the (net) rate of CO_2 vented out of the room. The ventilation rate n, expressed in ACH (air changes per hour) can be calculated from the rate of injection, and the difference between the CO_2 concentration in the exhaust air and the ambient concentration.

$$(C_{EXH} - C_{AMB})\cdot V\cdot n = \Phi_{INJ} \qquad (1)$$

As an analogy, the CO_2 injected can be thought of as a ‘dye’ to ‘colour’ the air leaving the room. The higher the ventilation rate, the more ‘diluted’ the colour will be. So it is apparent that the CO_2 concentration of the outlet air (C_{EXH}) should be monitored rather than the CO_2 concentration in the room.

The CO_2 sensor however is originally intended to monitor the effectiveness of the ventilation system, in particular at a location away from the window. Therefore, the sensor in each room is mounted on the hind wall, as shown in figure 5. The effect of this is discussed further below.

The ambient CO_2 concentration is measured during (prolonged) periods without CO_2 injection. Values range from 380-400ppm.

The assumption of a (quasi) equilibrium situation is justified by the observation from the measurements that in general, after opening the panels/window, the CO_2 concentration reaches a new (quasi) equilibrium value within 10 minutes. To be on the safe side, data from the first hour after opening the window/panels is discarded.

An example of the CO_2 concentration measured and the ventilation rate n calculated using equation (1) is shown in figure 6 for the room with the parallel window for a week in June 2004. This particular experiment actually ran from May 24 until June 28, from which only part is shown in figure 6 to better show the evolution of the data. To average out variations, in particular the effect of discontinuous CO_2 injection, the CO_2 concentration shown is the hourly average (average of 7 data-points).

During the day, no CO_2 is injected. The CO_2 concentration slowly decreases due to infiltration of outside air. The daytime infiltration rate is estimated from the rate of decay of the CO_2 concentration. It is in the order of 0.15ACH. There is no ventilation of the rooms other than night cooling and infiltration.

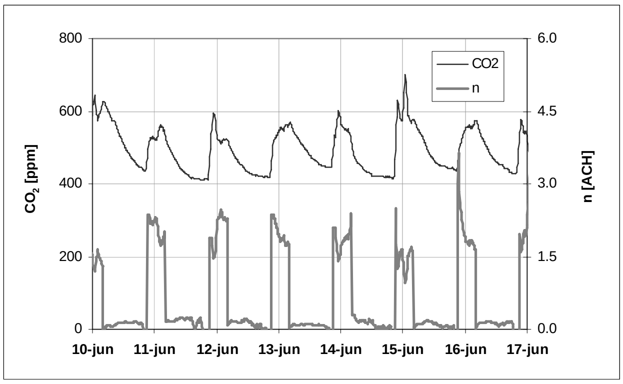

Figure 6 CO_2-concentration measured (hourly average) and ventilation rate n calculated for the case of the room with the parallel window.

CHECK ON VENTILATION RATES

To verify that the procedure for determining ventilation rates using equation (1) yields reasonable values, a dynamic building simulation model is used.

In fact, two building simulation models have been used. One is the internationally accepted and validated TRNSYS (TRNSYS, 2002) and the other is a simplified model. These models are discussed in detail in (Koene et al., 2005). Both appear to be able to adequately describe the dynamic thermal behaviour of the research facility. Because the simplified model is less cumbersome, it is used in this particular analysis.

When the heating system is switched off at night, the rooms will cool down due to conduction and infiltration losses. The rooms where night cooling is applied will show an additional temperature drop. Obviously, the higher the night cooling ventilation rate, the higher the additional temperature drop.

However, due to the dynamic nature of the process, with the thermal mass of the building and heat transfer between air and walls playing an important role, it is not straightforward to determine the night cooling ventilation rate from the additional temperature drop. For that reason, the building simulation model is used.

The model calculates the evolution of the wall temperature on the basis of experimentally determined heat gains and heat losses, such as solar heat gain, internal heat gains, conduction losses and ventilation losses. The latter are based on measured ventilation rates using equation (1). This way, we arrive at the *calculated* indoor wall temperature.

The indoor wall temperature is also experimentally determined by taking the average of the measured temperatures of side walls, hind wall, floor and ceiling. This is called the *measured* indoor wall temperature.

As an illustration, figure 7 shows the *calculated* and the *measured* indoor wall temperature for the case of the 'reference' room, where no night cooling is applied.

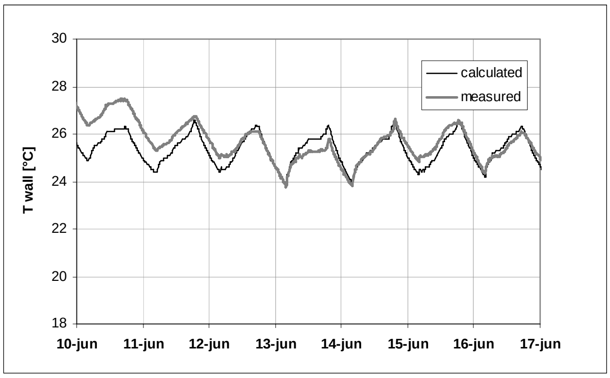

Figure 7 Calculated and measured indoor wall temperature for the case of the 'reference' room, where no night cooling is applied.

The grey line shows the measured indoor wall temperature and the black line shows the calculated indoor wall temperature. Although the agreement between measurement and simulation is far from perfect, the model correctly calculates that the indoor temperature will drop approx. 2°C during the night due to conduction and infiltration losses.

Figure 8 shows the case of the parallel window, where night cooling is applied.

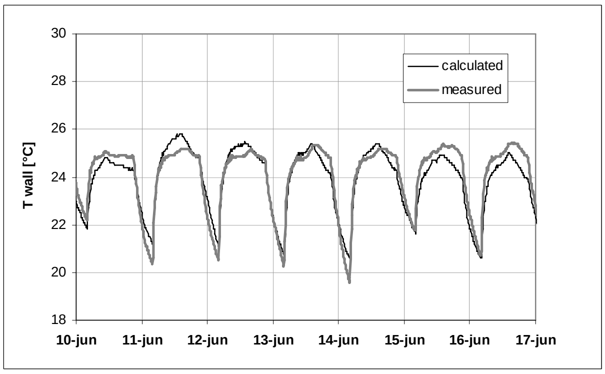

Figure 8 Calculated and measured indoor wall temperature for the case of the room with the parallel window, opened during the night.

Again, there is no perfect agreement between calculated and measured values, but compared to the reference room, the model correctly calculates an additional temperature drop of about 2°C, on the basis of ventilation rates determined using equation (1). This shows that the procedure for determining night cooling ventilation rates indeed yield reasonable results.

In the previous analysis the wall temperature rather than the air temperature is used. The reason being that measurement of the latter strongly depends on the location of the sensor. Sensors placed near the window opening will measure air temperatures near the ambient value, while sensors located at the back of the room are expected to measure higher values.

CFD CALCULATIONS

Of the various computational methods available, Computational Fluid Dynamics (CFD) has developed into the most powerful numerical technique for investigating indoor conditions on a detailed level (Choudhary et al., 2001). CFD models use a finite volume method to simultaneously solve the equations that govern conservation of mass, momentum and energy. The geometrical domain is presented by grid cells, in which variables are defined at the centre. A converged solution is obtained in an iterative way, by reducing the residuals for mass, momentum and energy.

The rooms and the façades described above are modelled using the CFD-code FLUENT6 (FLUENT, 2001). This model uses a finite volume method to simultaneously solve the equations that govern conservation of mass, momentum and energy.

Each room is modelled with a grid of approx. 500,000 tetrahedral cells. In the vicinity of the façade openings, the grid is finer (cell size approx. 2mm) than in the interior of the room (cell size approx. 30cm). Previous modelling work on mechanical ventilation of this room showed there was no discernable effect of discretisation with a grid of 40,000 cells. A finer grid is used here to accurately model the flow through the façade openings.

The outside world is modelled with a volume measuring 15x11x10 m outside the room, containing approx. 400,000 tetrahedral cells. In order to reach a steady state situation, a small wind speed of 0.1m/s is assumed at a distance of 15m from the window, perpendicular to the façade. The wind, leaving the computational domain at the top (at a height of 10m) will carry away the exhaust air from the room. The reason for selecting a small wind speed (rather than taking the average of values found in the experiments) is to eliminate as much as possible the effect of wind.

DNS (Direct Numerical Simulation) is generally considered to be the best technique available for modelling turbulent flows. With DNS, even the smallest turbulent eddies are simulated. However, this requires a very fine discretisation in space, as well as in time. As a result, the approach sets high demands on computational power.

Present computer power does not allow the use of DNS for most turbulent flow applications. For practical applications, the use of RANS (Reynolds Averaged Navier Stokes) models is the standard approach. With RANS models, the mean flow field is resolved on the numerical grid, whereas the turbulence is not. The effect of turbulence on the mean flow field is modelled with a RANS turbulence model.

In the present study, a RANS model is applied and turbulence is modelled with the standard k-ε model with wall functions. The latter prescribe the production of turbulence, as well as the temperature and velocity profiles near the walls and openings.

Previous validation on the use of these models for natural convection flows showed that wall functions are less suitable for representing boundary layers and their effect on the main flow (Kenjeric, 1998).

A more accurate way of modelling the heat transfer between air and wall with a RANS model, would be to integrate the boundary layer up to the wall, using for instance the low Reynolds number adaptation of the k-ε turbulence model (Lankhorst, 1991).

However, it has also been observed that the use of wall functions can lead to acceptable main flow solutions for cases comparable to the ones modelled here (Komen, 1998). Therefore, the use of wall functions, being cost-effective, has been selected for the current analyses.

Gravity is included in the model. The gas density is modelled using the ideal gas law. For other gas properties, constant values are used, corresponding to those of air at 20°C.

For reasons of simplicity, CFD-calculations are carried out for a steady state situation. Again, this approach is justified by the observation from the measurements that after opening the panels/window, the CO_2 concentration reaches a new (quasi) equilibrium value within 10 minutes, while the temperature drop of the building fabric is much slower (typically 0.5K/hr). Therefore, the inside surface of the room can be assumed to be at or near the initial temperature, while the airflow is fully developed (steady state situation).

The walls are assumed to have a constant temperature of 25°C while the outside air is at 15°C. The ambient CO_2 concentration is assumed to be 400ppm. As mentioned before, a small wind speed of 0.1m/s is assumed outside to prevent heating of the outside world by warm air leaving the room. In a sensitivity analysis, the effect of a (small) wind speed on ventilation rates turned out to be negligible.

The CO_2 production in the room is modelled as a point source at the appropriate location with a magnitude of 0.015Nm^3/hr. This value is the same as the experimental value. The momentum of the CO_2 gas injected is modelled as a momentum point source. A first order discretisation scheme in space is used. This turned out to be adequate for this case, since a simulation with a second order scheme resulted in the same solution.

RESULTS

The CFD calculations yield values for air velocity (in 3 dimensions), temperature and CO_2 concentration in each of the 900,000 cells. As an illustration, figures 9 and 10 show the distribution of temperature in the interior of the room with the parallel window and the room with the panels, respectively. The temperature distribution is shown in a plane perpendicular to the façade through the middle of the window/panels.

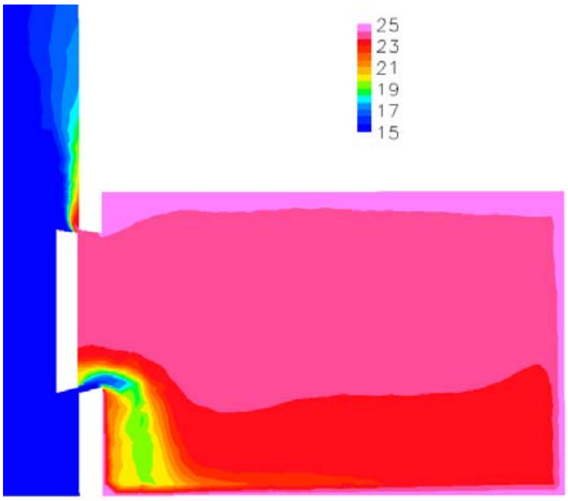

*Figure 9 CFD simulation of the temperature distribution in the interior of the room with the **parallel window** in a plane perpendicular to the façade through the middle of the window. The rectangle on the left is part of the outside world at a temperature of 15°C.*

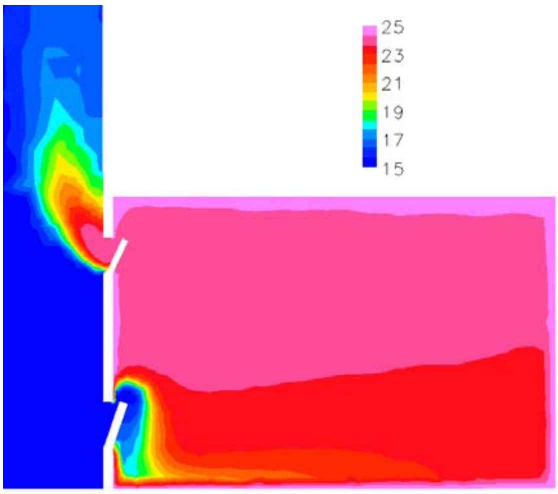

*Figure 10 CFD simulation of the temperature distribution in the interior of the room with the **panels** in a plane perpendicular to the façade through the middle of the panels.*

Figures 9 and 10 show cool outside air flowing into the room through the lower part of the facade, and air heated to approx. 25°C leaving the room through the upper part. As the temperature of the exhaust air is only slightly lower than the average temperature in the room, the (temperature) ventilation efficiency (expressed as $[T_{EXH} - T_{AMB}]/[T_{IN} - T_{AMB}]$) is in the order of 90%. Apparently there is no 'short circuit' in which the incoming air goes straight to the outlet opening without first flushing the room.

The CFD calculations are carried out for a range of outside temperatures, while the indoor temperature remains fixed at 25°C. Air velocities are integrated over the area of the opening, yielding a value for the ventilation rate. Results of the CFD calculations for both geometries are shown as solid lines in figure 11.

Figure 11 also shows the measured data as symbols. To minimise the effect of wind speed, only measurements are used where wind speeds were below 2.5m/s (measured at a height of 10m).

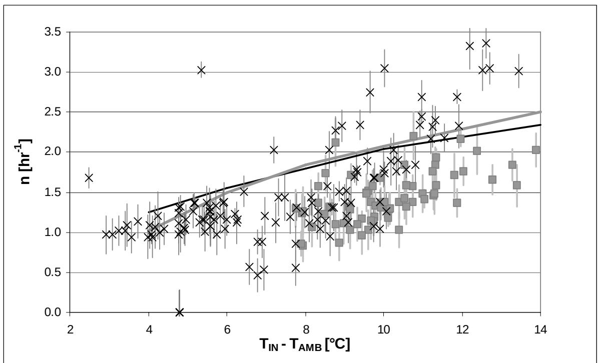

Figure 11 CFD calculations (solid lines) and measurements (symbols) of night cooling ventilation rate versus the difference between indoor and ambient temperature. Black line and black crosses: two panels, grey line and grey squares: parallel window. Error bars are due to the (estimated) effect of wind speed.

DISCUSSION OF THE ANALYSIS

The measurements show a considerable amount of scatter. There are several reasons for that. One is that the effect of wind is not completely eliminated. Analysis of another set of measurements (not shown here) with small temperature differences and varying wind speeds indicate that an increase of wind speed of 1 m/s (measured at a height of 10m) increases the ventilation rate by a value in the order of 0.2ACH. This relationship is used to estimate the magnitude of the error bars in Figure 11. Unfortunately, lowering the maximum allowable wind speed below 2.5m/s would keep us bereft of the majority of the measurements (at ECN, the climate is rather windy).

The data presented in figure 11 come from three experiments carried out between May 24 and August 23, 2004. In each experiment, a constant value for the ambient CO_2 concentration is used in equation (1). This is another possible source of error since the ambient CO_2 concentration actually varies with wind direction and wind speed.

Figure 6 shows that during a typical night, CO_2 concentrations in the room are in the order of 100 ppm over the ambient value. Thus, an error in the ambient value of 10-20ppm will cause an error of 10-20% in the calculated ventilation rate. The effect is higher, the closer the CO_2 concentration in the room is to the ambient value (which is the case for higher ventilation rates).

A solution would be to use a second CO_2 sensor to simultaneously monitor the ambient CO_2 concentration. An alternative would be to leave the window/panels open in the morning for a few hours without injecting CO_2. This would allow the CO_2 concentration in the room to approach the ambient value (assuming ideal mixing, a ventilation rate of 1.5ACH for 3 hours would decrease the value of 100ppm over the ambient value to approx. 1ppm).

Another possible source of error is the location of the CO_2 sensor. As discussed previously, it is located at the hind wall, while it is the CO_2 concentration in the exhaust air that should have been monitored. So, in retrospect, the sensor is not located at the ideal position for this kind of analysis. The CFD calculations show that the CO_2 concentration at the hind wall is somewhat lower than the concentration in the exhaust air flow. Using equation (1) with too low a CO_2 concentration will yield an erroneously high ventilation rate.

The accuracy of the experimentally determined flow rates is expected to increase by using a CO_2 sensor to continuously monitor the ambient CO_2 concentration, and by using one or more CO_2 sensors to measure the CO_2 in the outlet air flow.

On the other hand, there are also uncertainties in the CFD calculations. Already mentioned is the challenge of adequately describing turbulence in the air and the heat transfer between air and wall.

Yet another uncertainty in the CFD calculation is a simplification of the geometry of the opening of the parallel window (without edges), offering a more favourable air flow path.

A difference between CFD calculations and measurements concerns the gauze in the openings, which is intended to keep insects out. The gauze is not modelled as it is not believed to have a large effect on ventilation rates. The reasoning is that it is loosely draped and thus does not decrease the minimum opening in the flow path. However, the gauze does represent an extra resistance to the airflow, which is not taken into account in the model.

A further difference between CFD calculations and measurements is related to the wall temperatures. In each of the CFD calculations, a uniform temperature of the inner fabric of the room is assumed. In the experiments, the temperatures of walls, floor and ceiling, each measured in one particular spot (see figure 5) are generally found to differ less than 1ºC (as expected, the floor temperature is lowest). However, temperatures in other locations (e.g. in the corners) may differ more, especially after a night of cooling.

The black line in figure 10, representing the results of the CFD calculations for the case of the *panels*, runs more or less through the scattered measurements (black crosses), although individual measurements may differ as much as a factor of 3 from the calculated values.

Both measurements and calculations show a general trend of increasing ventilation rate with increasing temperature difference, as expected.

In the case of the *parallel window*, the same trend of increasing ventilation rate with increasing temperature is observed. However, a line fitted through the measurements (grey symbols in figure 10) would lie approx. 30% lower than the calculated values (grey line). Here, individual measurements may differ a factor of 2 from the calculated values.

Although the CFD calculations predict comparable ventilation rates between both geometries, measurements show that the parallel window allows ventilation rates that are typically 30% lower than the panels do. This is rather good an achievement in view of the relatively small opening that the parallel window offers for the airflow (3cm all around). Obviously, larger openings are expected to yield higher ventilation rates.

As mentioned previously, there are openings in only one façade in the room (single-sided ventilation). Higher ventilation rates are possible with openings in more than one façade or with openings in different floors.

SUMMARY AND CONCLUSIONS

CFD calculations on night cooling have been carried out for two different geometries, a set of panels and a 'parallel window'.

Ventilation rates for these geometries have also been measured in a research facility. A considerable amount of scatter is found in the measurements, due to limitations of the experimental set-up.

A more controlled experiment e.g. by using more CO_2 sensors is expected to increase the accuracy of the experimentally determined flow rates. This would allow a better basis for assessing the merits and limitations of CFD-calculations.

In spite of the limitations of the analysis, both measurements and calculations show a general trend of increasing ventilation rate with increasing temperature difference between indoor and outdoor climate, as expected.

For the case of the *panels*, the results of the CFD calculation run more or less through the scattered measurements, although individual measurements may differ as much as a factor of 3 from the calculated values.

For the case of the *parallel window*, the 'centre of weight' of the measurements lies approx. 30% below the CFD values. Here individual measurements may differ from the calculated values by a factor of 2.

In spite of the relatively small opening of the parallel window (3cm all around), ventilation rates measured are only about 30% lower than those found with the panels. Apparently, this window design is quite effective in combining inlet and outlet opening into a single component, simplifying the actions needed for night cooling (manual or automated).

Finally we may conclude that the combination of experiment and CFD simulation proves to be a valuable way to obtain detailed information on night cooling by natural ventilation.

NOMENCLATURE

C Concentration of CO_2 [ppm]

n Ventilation rate [hr^{-1} or ACH]

V Volume of the room [m^3]

Φ Flow rate [m^3/hr]

Subscripts

IN indoor

AMB ambient

INJ injected

EXH exhaust

REFERENCES

Awbi, Hazim B., Calculation of convective Heat Transfer Coefficients of Room Surfaces for Natural Convection, Energy and Buildings 28 (1998), pp 219-227.

Choudhary, R. and Malkawi, A., A methodology for micro-level building thermal analysis: combining CFD and experimental set-ups. Seventh International IBPSA Conference, 2001, pp 1275-1282.

FLUENT 6.0 User’s Guide, Fluent Inc., 10 Cavendish court, Lebanon, NH 03766, USA, December 3, 2001.

Kenjeric, S., Numerical Modelling of Complex Buoyancy-Driven Flows, PhD thesis, Delft University of Technology, Delft, The Netherlands, 1998.

Koene, F.G.H., Bakker, E.J., Validated building simulation tool for 'active façades'. Paper for IBPSA 2005 conference (in preparation).

Komen, E.M.J., Houkema, M., CFD Modelling of Natural Convection - Validation report, ECN-R—003 71375/NUC/EK/mh/012006 Petten, The Netherlands, 1998.

Lankhorst, A., Laminar and Turbulent Natural Convection in Cavities - Numerical Modelling and Experimental Validation, PhD thesis, Delft University of Technology, Delft, The Netherlands, 1991.

Loomans, M., The measurement and Simulation of Indoor Air Flow, Ph.D. thesis, Eindhoven University of Technology, 1998.

Pfafferott, J. et al., Design of passive cooling by night ventilation: evaluation of a parametric model and building simulation with measurements, Energy and Buildings 35 (2003) 1129-1143.

TRNSYS 15, The Transient Energy System Simulation Program, Solar Energy Laboratory (SEL), University of Wisconsin Madison, USA, 2002.