STUDY ON INDOOR HUMIDITY DISTRIBUTIONS

Y. Aizawa[1] , J. Liu and H. Yoshino[2]

[1]*Technical Research Institute, Tokyo Gas Co., Ltd.*
16-25, Shibaura 1-Chome, Minato-ku, Tokyo 105-0023, Japan
[2]*Department of Arch., Urban Planning and Building Eng., Tohoku Univ.*
06 Aramaki Aza, Aoba-ku, Sendai, 980-8579, Japan

ABSTRACT

This study investigates the behavior of vapor in a ventilated room in which vapor is being produced. A test chamber equipped with three types of ventilation ducts and a vessel filled with heated water for evaporation was analyzed both experimentally and numerically. Experimental results showed that temperature and moisture distributions differed depending on the ventilation types. A numerical model of vapor generation from the heated water was introduced to simulate the evaporation. CFD calculations including heat and humidity were carried out and the results were in good agreement with the experiment results.

KEYWORDS

temperature distribution, humidity distribution, ventilation, chamber experiment, vapor generation model, CFD

INTRDUCTION

Condensation and high humidity have been identified as one of the IAQ problems. Mold propagates actively in a high humidity atmosphere and may cause problems to human health. Condensation on a cold surface in a building can damage the building materials. In Japan, not only in the rainy season but also the summer, humidity is very high compared to European countries. Furthermore, the Japanese living style tends to generate a large amount of water vapor from the bathroom, cooking and so on. To avoid condensation and high humidity, the factors described above are very important. With proper control of the ventilation and humidity, a healthy and comfortable indoor environment for both humans and the building can be achieved.

When designing a ventilation system for controlling indoor humidity, the behavior of vapor must be understood. However, very few studies have been conducted on indoor humidity distributions. Though the CFD approach is expected to provide useful information for humidity distributions, an appropriate model to simulate evaporation, simultaneous heat and mass transfer analysis is necessary.

In this paper, the behavior of vapor in a ventilated space in which vapor was being produced was analyzed both experimentally and numerically. A small test chamber equipped with three types of ventilation ducts and a vessel filled with heated water for evaporation was tested. Experimental results showed that different temperature and moisture distributions depend on the ventilation types. A numerical model of vapor generation from the heated water was introduced to simulate the phenomena. CFD calculations including heat and humidity were carried out. The CFD results were in good agreement with the experiment results.

EXPERIMENTAL ANALYSIS

Test Chamber and Experimental Measurements

To see the effect of ventilation type on the temperature and humidity distributions, a small test chamber equipped with three types of ventilation ducts and a vessel filled with heated water for evaporation was tested. The chamber (x:1220; y:1210; z:1500) was set in a room in which temperature and humidity were kept constant. Figure 1 shows a schematic drawing of the chamber with the measuring instruments. The chamber was made of polyvinyl chloride, which has no effect on moisture absorption. Three ducts and one fan in duct 1 were installed to test the ventilation types, which may cause different airflow patterns in the chamber. Figure 2 shows a side view of the tested ventilation types. Case 1 uses duct 1 as an exhaust and duct 2 as an inlet, which simulates the typical ventilation type. To investigate the difference, Case 2 uses two ducts in reverse, duct 1 as inlet and duct 2 as exhaust. In Case 3, duct 1 (exhaust) and duct 3 (inlet) are at the same height which pass through the center of the chamber. Hot-wire type anemometers were used to measure the ventilation rates. Temperature and relative humidity sensors were set on the two positions A and B to measure vertical distributions. Wall surface temperatures were measured by thermocouples. To humidify the chamber, a vessel filled with water was set in the center and a heater was put in the water to control the water temperature. Water temperature was kept at 35_ throughout the experiments. Evaporation rate was measured by the weight decrease of the vessel. The outside of the chamber was maintained at 13.6_ and 55% RH during all of the experiments. Table 1 summarizes the test cases.

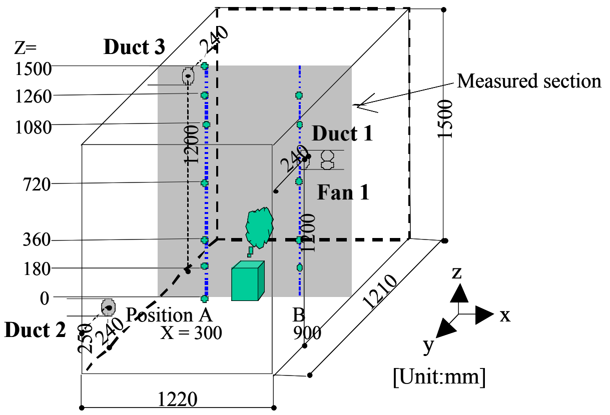

Figure 1: Schematic of the test chamber

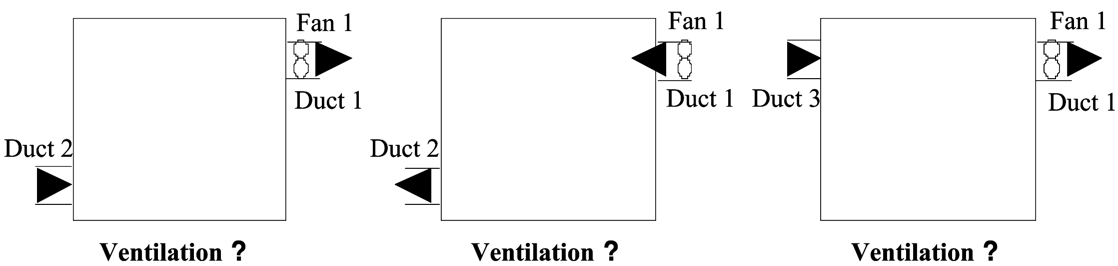

Figure 2: Schematic of ventilation types

Table 1
Measured cases

Case	Ventilation way	Ventilation rate	Vapor generation rates	Supply air
Case 1	Ventilation _	8.3 m^3/h (3.8 h^{-1})	30 g/h (water temperature: 35_)	13.6_, 55%(5.3 g/kg')
Case 2	Ventilation _			
Case 3	Ventilation _			

Effect of Ventilation on Indoor Temperature and Humidity Distributions

Figure 3 shows the vertical temperature and moisture distributions at two positions (A, B) in the steady state. The ventilation rate was kept the same for the three cases (Case 1-3). Because of the relatively higher water temperature of the vessel, the mean air temperature in the chamber was about 1 to 1.5 higher than the inlet air temperature, which was maintained at 13.6_. The temperature gradients were similar in all three cases. On the other hand, both absolute and relative humidity near the floor were different between ventilation types. In Case 1, due to the buoyancy effect of evaporated vapor and the additional exhaust flow near the ceiling, moisture was exhausted effectively, and a lower inlet position (Duct 2) gave a lower humidity value near the floor of 7.5 g/kg' (75%). In other cases (Case 2 and 3), humidity distributions were relatively flat, suggesting well mixing of supplied air and evaporated vapor.

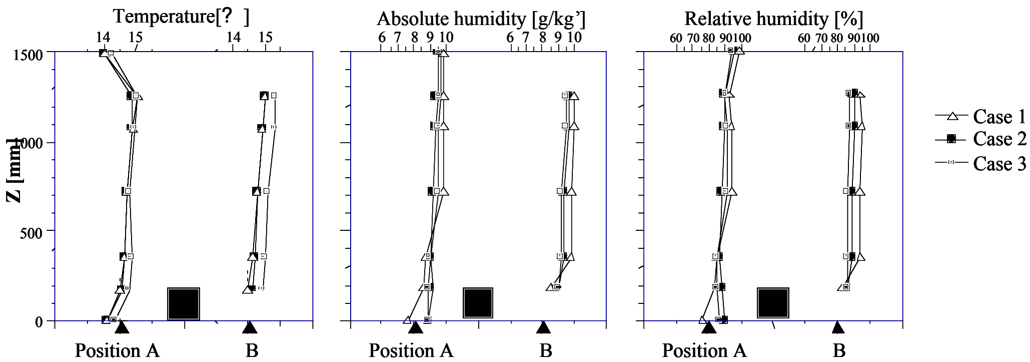

Figure 3: Temperature and humidity gradients for different ventilation types (Cases 1-3)

NUMERICAL ANALYSIS

New Numerical Model of Simultaneous Heat and Mass Transfer from Water Surface

In general, an experimental approach provides measured data at the measured points. To investigate three-dimensional humidity distributions and flow patterns, a numerical approach is one of the possibilities, and proper modeling of phenomena is essential for appropriate numerical results. Heated water evaporates from a vessel and flows as vapor phase. To model those phenomena, not only heat transfer but also moisture transfer should be taken into consideration. Several methods [1],[2] have been developed for different fields to model. In this study, the process was considered as the following two paths (see Figure 4):

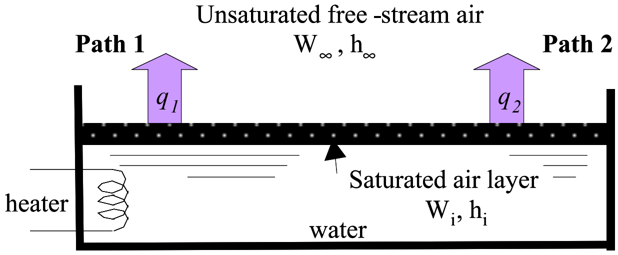

Figure 4: Two paths of heat and mass model of vapor

Path 1) If the ambient air temperature is much lower than the water surface temperature, the saturated vapor across the surface to the air will be changed into the liquid phase quickly and

latent heat will be transferred to the air.
Path 2) If the temperature difference between the ambient air and the water surface is not large enough, the vapor will remain as gas phase and the simple heat and mass transfer should be considered.

The real phenomenon is considered to be somewhere between Path 1 and Path 2. Assuming the steady state to simplify the model, a constant value α is introduced to express average heat flux during a certain period. The applied equation including the two paths is as follows:

$$q'' = q_1 + q_2 = \alpha * \dot{m} * \gamma + \frac{(1-\alpha) * \dot{m}}{(W_i - W_\infty)} * (h_i - h_\infty) \qquad (0 < \alpha < 1) \quad (1)$$

where $\dot{m}$ = moisture flux, kg/(s_m^2); W_i, W_∞ = absolute humidity of the air-water interface and main fluid stream, kg/kg'; h_i, h_∞ = enthalpy of the air-water interface and main fluid stream, kJ/kg; W_i and h_∞ are unknown values so that they are approximated here using the simple multi-zone model; γ = latent heat, kJ/kg; q'' = heat flux per unit area, W/m^2; α = coefficient having a value between 0 and 1, it express the ratio of heat flux generated by latent heat to the total heat flux. It can be considered to be a function of the temperature difference between the water surface and ambient air.

Numerical Model of CFD

The standard _-_ model for CFD was used in this study. The boundary and calculation conditions are summarized in Table 2. The flux of heat and mass of vapor from water is set up to the ambient air. As the simulation tool of CFD, STREAM for windows was used.

Table 2
Boundary and calculations for CFD

Turbulence model	Standard $\kappa-\varepsilon$ model
CFD grid points	39(X)_39(Y)_47(Z)=71487
Numerical schemes	QUICK
Supply, exhaust	Duct 1: measured value in corresponding experiment is used as the airflow velocity of opening with fan; Duct 2,3: calculated with mass balance
Wall	No-slip
Moisture calculation	Using diffusion equation; coefficient of vapor diffusion :2.6_10^{-5} m^2/s
Heat and mass model	Cal1: no heat transfer; Cal2: only path 2_ α=0.0_; Cal3: only path 1_ α=1.0_; Cal4: Path 1,2_proper α_

Validation of New Numerical Model

To decide the value of α, four CFD runs (Cal1-4) were performed for both Case 1 and Case 2 and compared with the experimental results. Figure 5 and 6 show comparisons between the experimental results and CFD results in the steady state. Several values of α were tested including α=0 (Cal2) and α=1 (Cal3). The same value of _ was applied for both Case 1 and Case 2 because the temperature difference between the heated water and the ambient air was the same. When _ was set to 0.3, calculated humidity distributions were in reasonably good agreement with the experiments for both cases. This means that about 30% of latent heat was diffused to the ambient air. This might be explained by the fact that the temperature difference between the water surface and the ambient air was not very large

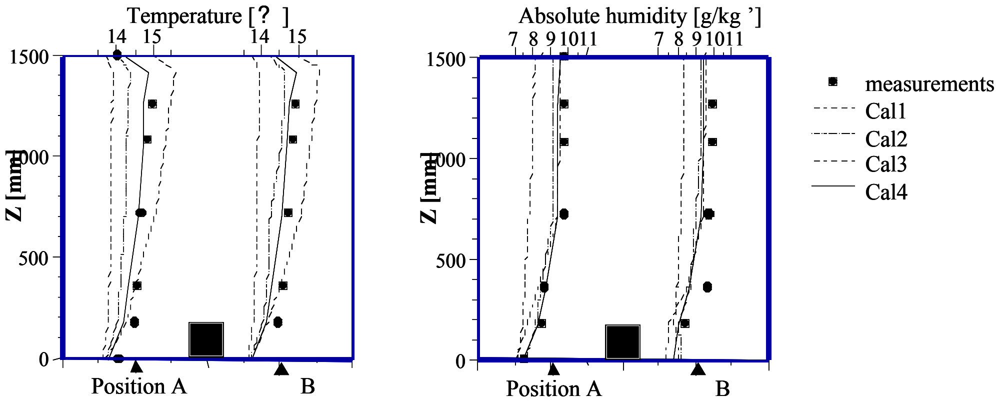

Figure 5: Comparison between calculated and measured values in Case 1

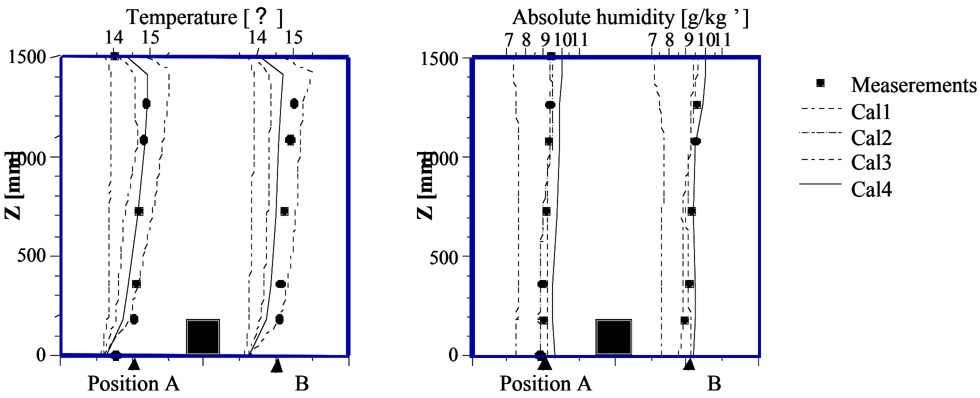

Figure 6: Comparison between calculated and measured values in Case 2

(about 20 degrees difference). The result also suggested that the value of _ might be closely related to the temperature difference between the water surface and the ambient air, and independent of the ventilation type.

Figure 7 and 8 show the temperature and moisture distributions at the section shown in Figure 1 from CFD results for Case 1 and Case 2 respectively. The moisture vertical gradients near the floor, which had a significant difference between Case 1 and Case 2 in the measured results, were observed in the CFD result as well. Figure 8 shows less humidity distributions in Case 2, with good mixing of vapor and air. In general, taking the boundary condition into account and using an appropriate value of _, the numerical approach is effective to investigate three-dimensional humidity distributions in a ventilated and evaporated chamber.

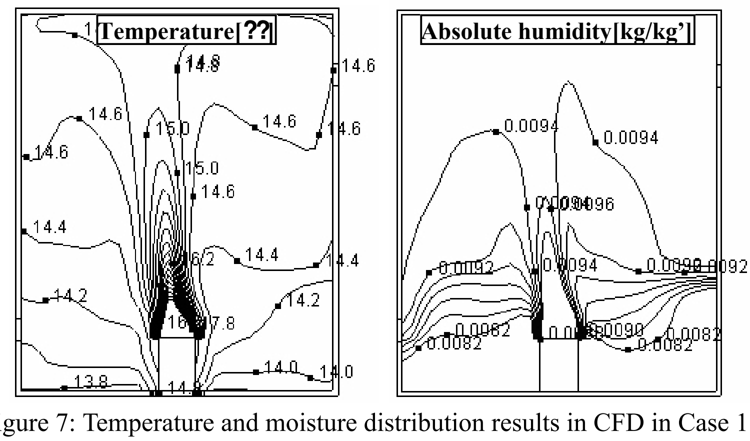

Figure 7: Temperature and moisture distribution results in CFD in Case 1

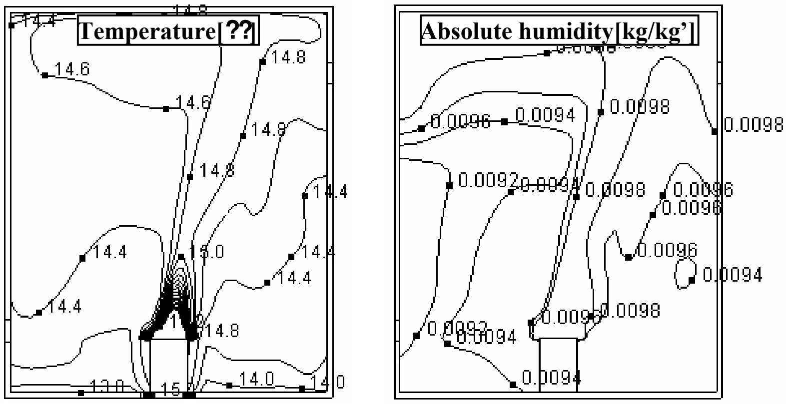

Figure 8: Temperature and moisture distribution results in CFD in Case 2

CONCLUSIONS

The behavior of vapor in a ventilated space in which vapor was being produced was analyzed both experimentally and numerically. A small test chamber equipped with three types of ventilation ducts and a vessel filled with heated water for evaporation was tested. A numerical model of vapor generation from the heated water was introduced to simulate the phenomena. CFD calculations including heat and humidity were carried out and the results were in good agreement with the experiment results.

The conclusions are as followed.

(1) Different temperature and moisture distributions were observed depending on the ventilation types.

(2) Heat flux from the water surface plays an important role in the vapor generation. With the numerical model introduced in this paper, a good agreement between CFD and measurement results was achieved.

Further the investigation is necessary for determining the appropriate value of _ depending on various conditions such as ventilation rates, temperature differences between water surface and ambient air, room dimensions and so on.

APPENDIX

The value of _ is decided if Eqn.2 is satisfied, so the decided _ is not a single value.

$$\delta = \frac{\left|V_{ex} - V_{cal}\right|}{V_{ex}} \leq 0.05 \qquad (2)$$

where V_{ex} = measured values of temperature and humidity; V_{cal} = calculated values of temperature and humidity

REFERENCES

[1] N. Arai, M. Kameda, M. Ogawa, 1999. Heat load from large bath, *SHASE*, Vol.**73-1**, pp.53-64

[2]ASHRAE, *1997. ASHRAE handbook fundamentals*, pp.5.11-5.12, Atlanta