Building Simulation 2003

Eighth International IBPSA Conference
Eindhoven, Netherlands
August 11-14, 2003

EXPERIMENTAL AND CFD STUDIES ON SURFACE CONDENSATION

Liu Jing[1], Yoshihiro Aizawa[2], Hiroshi Yoshino[3]
[1]School of Municipal and Environmental Engineering, Harbin Institute of Tech, Harbin, China
[2]Technical Research Institute, Tokyo Gas Co., Ltd., Tokyo, Japan
[3]Tohoku University, Sendai, Japan

ABSTRACT

Condensation and mold problems have been identified as one of the severest IAQ problems in Japan. Especially in the wintertime, moisture condenses on cold wall surfaces where it can cause deterioration of the building materials and mold growth related to allergic symptoms. This paper discusses the possibility of using the CFD method to solve condensation problems. Firstly, a CFD model for simulating condensation is developed, and then the validity of this model is examined experimentally. The results indicated that there is a good agreement between the experimental results and the model predictions. Then, the effect of ventilation on condensation risk is discussed using this model, and it is found that the condensation performance is significantly affected by the ventilation design.

INTRODUCTION

Condensation and mold problems have been identified as one of the severest IAQ problems in Japan. The reduction of ventilation levels for energy saving has increased these problems in many cases. Especially in the wintertime, moisture condenses on cold wall surfaces where it can cause deterioration of the building materials and mold growth related to allergic symptoms (Nielsen et al., 2000; Ekstrand-Tobin et al., 2000).

To date, various studies have been carried out on the non-contact methods of measuring moisture condensation (Kubo et al., 1997), but complicated experimental apparatus and fairly skilled users are needed. Therefore, a computational simulation tool with acceptable accuracy would be very useful for solving these problems. However, when studying condensation by simulation, two very important points should be noted. 1) Many researches use multizone network models to study moisture-related problems (Ikeda et al., 1995; TenWolds, 1993), and do not take into account the temperature and moisture distributions which are very important considerations to investigate where condensation forms and its performance, and 2) For judging whether and when condensation occurs, the time-dependent condensation rate is needed in most cases, so an unsteady-state simulation method is necessary rather than a steady-state one.

Based on the above, in this paper, a CFD model for simulating condensation is developed, and then is validated with experimental data. Although the effect of ventilation on condensation risk has long been recognized, excessive ventilation may increase the heating load and produce uncomfortably dry air in cold winter climates. This paper also discusses how ventilation affects condensation risk.

VALIDATION EXPERIMENT ON CONDENSATION USING A TEST CHAMBER

Figure 1 shows a schematic drawing of the test chamber with the measuring instruments for the validation experiment. The chamber was located in an environmental room, and was made of poly-vinyl chloride, which can be considered without adsorption/desorption effects. There was a lower (Vent1: FL +250 mm) and an upper vent (Vent2: FL +1250 mm) on the two opposite sides of walls. Air was heated prior to being supplied from Vent 1 into

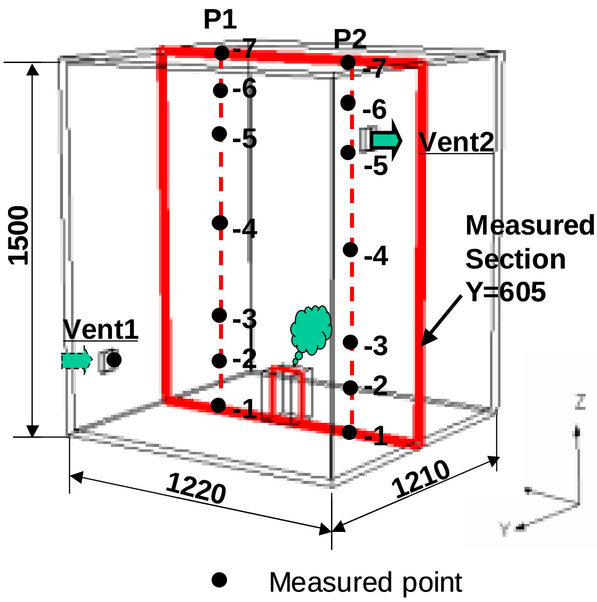

Figure 1: Schematic of the validation experiment within a test chamber (Unit: mm)

the chamber, and exhausted to the outside by a fan which was located at Vent2. Hot–wire type anemometers were used to measure the airflow rates through the vents. Water vapor was produced by an electrical immersion heater from a tray. A sensor was put into the water to control water temperature and the vapor generation rate. The weight of vapor was measured and calculated by an electronic balance. The time-dependent temperature and relative humidity were measured at 14 points within the chamber, by temperature-humidity sensors which were accurate to 0.5 degrees and 3.0%. Note that the measured section of temperature & humidity and the location of the tray are in the middle of the Y-axis (Y=605mm), differing from the section of the vents and fan (Y=970mm).

The measurement conditions are summarized in Table 1. The experiment was divided into the following two periods,

Period 1: About 240 min. The fan was run on to supply 25.2 degrees air into the chamber until the air temperature rose close to the steady state. During this period, the tray was covered to keep a steady vapor emission rate and heat flux when the next period started.
Period 2: About 30min after Period 1. The cover of the tray was removed and vapor was released to the free stream air.

Table 1: Measurement conditions

Air temperature and humidity outside the chamber	20 degrees 6.2g/kg' (50%RH)
Vapor generation rate	101.3g/h
Water temperature	70 degrees
Ventilation rate	$8.0m^3/h$ ($3.8h^{-1}$)
Supply air temperature	25.2 degrees

SIMULATION METHODS

CFD Model

The commercial Japanese CFD software STREAM4.0 was used in this study. A non-uniform mesh system was used with the finer mesh located in the near-wall region. The boundary and calculation conditions are summarized in Table 2. The simulation was divided into two parts according to the experiment,

Part 1: Steady-state simulation according to Period 1.
Part 2: Unsteady-state simulation according to Period 2, whose initial conditions within the chamber were the results obtained by Part 1.

Table 1
Boundary & calculation conditions

Turbulence model	Standard $k-\varepsilon$ Model
CFD grid points	39(X) * 47(Y) * 39(Z)=71487
Numerical schemes	QUICK
Exhaust	The measured air velocity of the fan is used
Supply	Calculated with mass balanced
Wall	Vel.: Standard log-law Heat: Fixed convective heat transfer rate
Moisture diffusion coefficient D_m	$2.6*10^{-5}m^2/s$
Initial conditions	The measured data

Condensation model

Surface condensation is assumed to form when the absolute humidity of ambient air next to the wall surface W_{sur} is higher than the saturated absolute humidity according to the temperature of the wall surface W_{a1}. The effect of adsorption and desorption is not considered in this model.
The time-dependent condensation rate per unit area on a wall surface $CON(s,\ t_1)$ and the accumulative condensation rate of the total wall surface $SUMCON\ (t_1)$ at time t_1 are defined as:

$$CON(s,t_1) = \int_{t=t1} h_D[W_{sur}(s,t) - W_{a1}(s,t)]dt \qquad (1)$$

$$SUMCON(t_1) = \int_s CON(s,t_1)ds \qquad (2)$$

In this paper, h_D is calculated using the Lewis relation.

$$h_D = \frac{h_T}{\rho c_p}\frac{1}{(L_e)^{0.67}} \qquad (3)$$

The combined simulation method is shown in Figure 2.

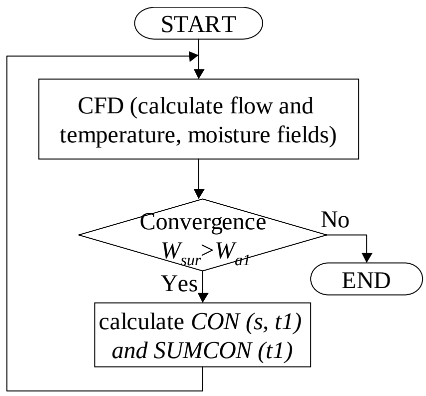

Figure 2 Flow chart of combined simulation of airflow, moisture and condensation

Case study

Firstly, CFD prediction was carried out using the measurement conditions as shown in Table 1 for validation (Case1). In addition, to assess the effect of ventilation on reducing condensation risk, two parameter studies (Cases 2, 3) were carried out. Heat transfer due to moisture transfer was set as 500W/m^2 for all the three cases according to other experiments that will be reported hereafter. Computations were time-dependent, and the time interval was set as 0.5s. The total simulation time for each case was about 5h.

1) Case1: Validation with the experiment in the test chamber.

2) Case2: Discussion of the influence of ventilation rate on condensation performance. In this case, ventilation rate increases from 7.9 to 9.4m^3/h (3.8h^{-1} to 4.5h^{-1}) compared with Case1. Other simulation conditions are the same as those of Case1.

3) Case3: Discussion of the influence of ventilation strategy on condensation performance. In this case, air is exhausted from Vent1 and supplied from Vent2. Other simulation conditions are the same as those of Case1.

RESULTS AND DISCUSSION

Comparison between measured and predicted results (Case1)

Firstly, Figure 3 shows the measured results of three vertical points at Positions 1 and 2 for Period 1. Note that because the tray was not covered completely, some vapor was released and resulted in the rise of humidity. Then this figure shows the comparison between the predicted and measured results for Period 2. The rise of temperature and humidity is due to the simultaneous temperature and humidity transfer. Because of the upward movement of supply air due to the exhaust fan and buoyant plume from the water surface, a vertical distribution with lower values near the floor and higher values near the ceiling can be observed for not only temperature but also humidity. There are differences of about 1.0 degrees and 1.4g/kg' at the end of Period 2. After about 20 min, both measured and predicted humidity at the ceiling reached saturation point, indicating that surface condensation is likely to happen in this area.

In general, the predicted results are in reasonable agreement with the measurements for temperature and humidity for Position 1 (the average relative difference between the simulated predictions and experimental measurements for temperature and humidity at 30min range from 0.9% to 4.8%, respectively), but some discrepancies can be seen for Position 2. Especially at the floor, both the predicted temperature and humidity is lower than the measured ones, and the relative difference is up to about 7%. Although velocity vectors were not measured, it can be concluded that compared with the experimental results, supply air can not reach the region of Position 2 in the prediction. This suggests that the effect of buoyancy on streamline curvature is over-estimated in the CFD model.

Simulation of condensation by CFD (Case1)

Figure 4 shows the *CON(s, t_1)* contours at t_1=20, 30, 45 and 60min from the start of vapor production. In

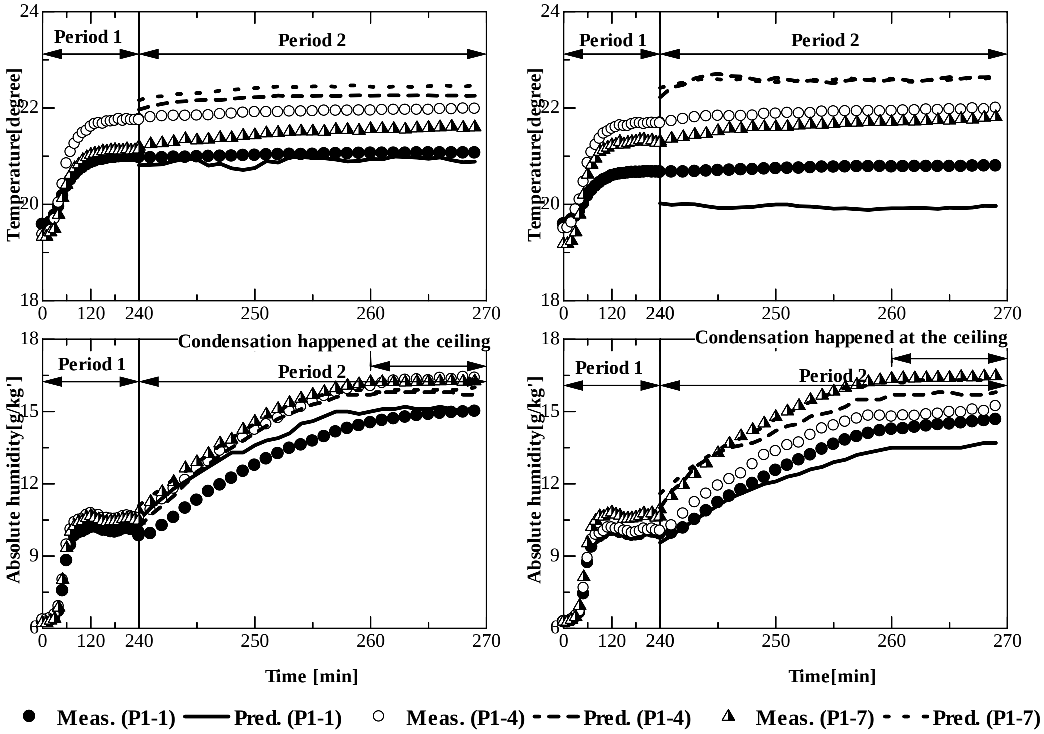

Figure 3 Comparison between measured and predicted temperature and moisture variations (Case1)

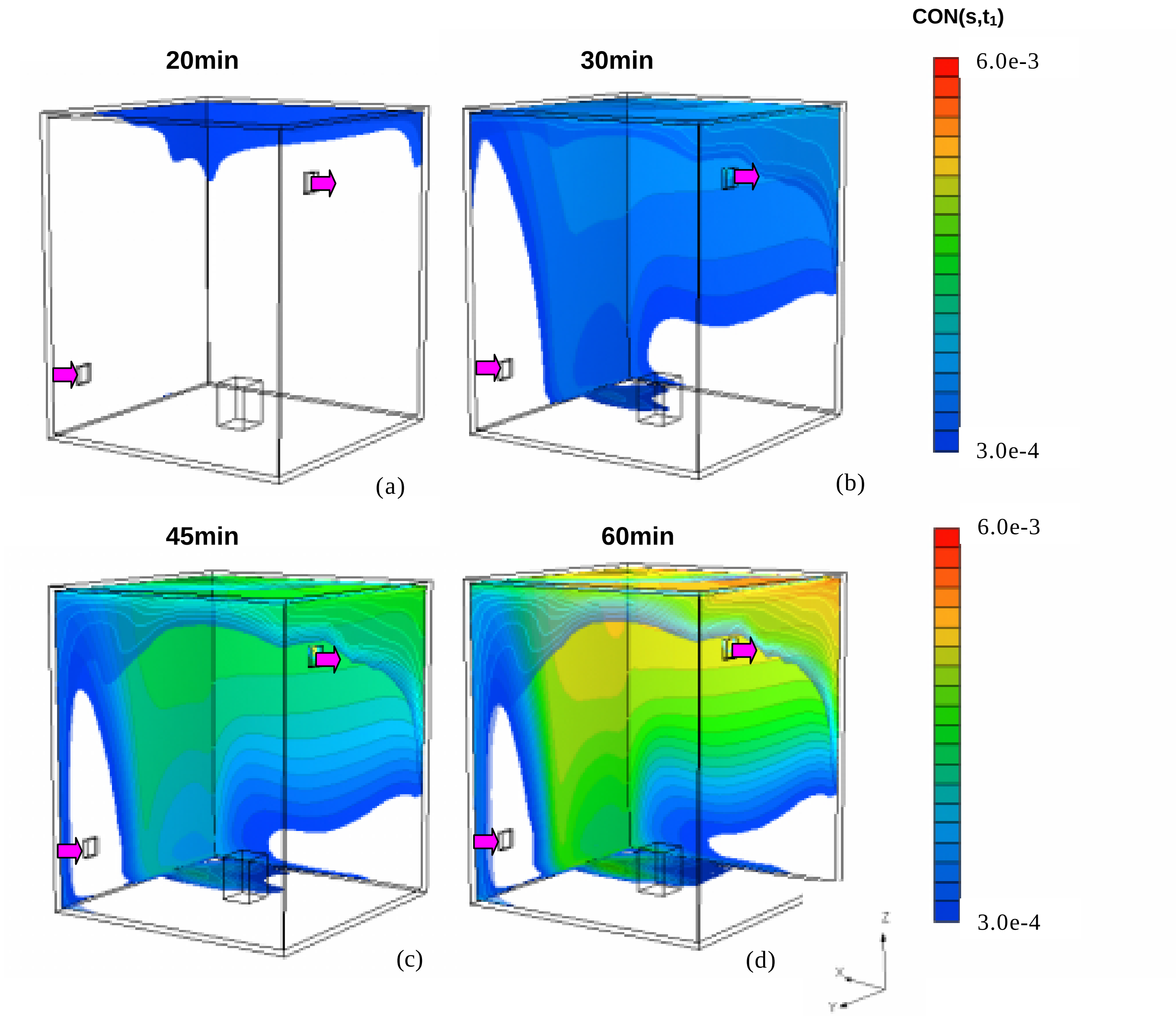

Figure 4 Predicted CON(s, t_1) contours for Case1

this paper, it is assumed that if *CON(s, t_1)* is below 1g/m^{2}h (3*10^{-4}kg/m^{2}s), it cannot cause significant damage to the wall surface. Although surface condensation was not measured directly, considering the reasonable agreement between the measured and predicted temperature and humidity distributions mentioned before, the prediction of condensation is expected to be acceptable. The results show that condensation happens in the corner near the ceiling firstly, then expands along the ceiling and the vertical walls. After about 45 min, the area of condensation does not change significantly. Because of the air movement by heated supply air, the temperatures of the wall surfaces near the fan and vents (i.e., the wall Y= max and X= min) are higher than those of other walls, and result in a lower condensation rate. Indeed, at the end of the experiment, significant water droplet condensation and frost buildup were observed at the regions corresponding to the simulation.

Discussion of ventilation on condensation performance

Figures 5 and 6 show the predicted *CON(s, t_1)* contours at 60 min after the start of vapor production for Cases 2 and 3 (compare with Figure 4 (d)). For Case2, as expected, *CON(s, 60min)* decreases as the supply flow rate increases. For Case3, *CON(s, 60min)* decreases significantly due to the change of ventilation strategy. Because the exhaust vent is moved to near the moisture source, moisture can be extracted directly to the outside instead of moving through the whole space, thus reducing the humidity levels near the ceiling. Figure 7 shows *SUMCON (60min)* at each wall and the total value of *CON (60min)* for all the cases. The total *SUMCON (60min)* value of Case2 and Case3 is only 26% and 20% compared with Case1, respectively.

CONCLUSIONS

A CFD model for simulating surface condensation has been developed and discussed. Based on the results and discussions, the following conclusions can be drawn:

1) The CFD model gives good agreement with the experimental results for simulating temperature and moisture distributions. It has been proved to be a powerful tool for studying the condensation risk and evaluating the impact of various parameters.

2) Except for the sufficient supply flow rate, the ventilation strategy has a marked effect on the moisture distribution and condensation performance. By placing the air inlet and extract positions appropriately, the condensation risk can be reduced significantly.

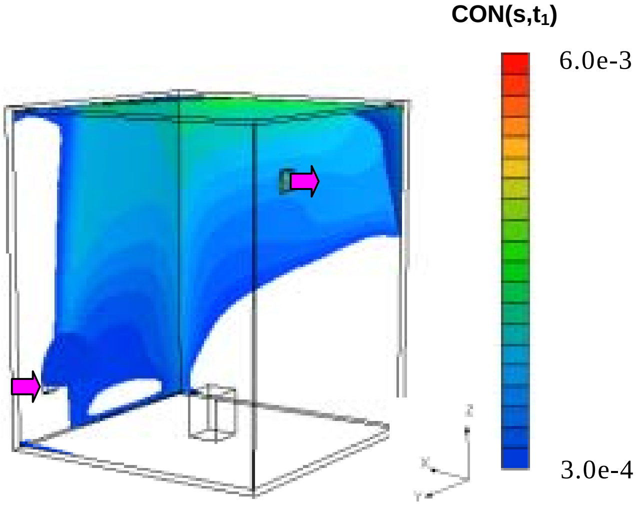

Figure 5 Predicted CON(s, t_1) contours for Case2 (t_1=60min)

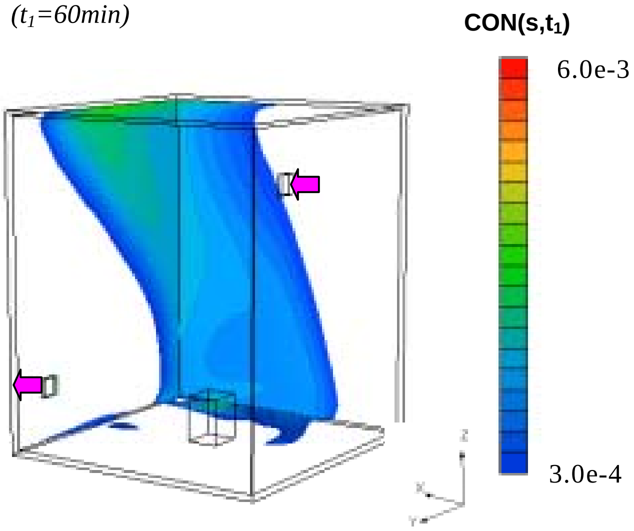

Figure 6 Predicted CON(s, t_1) contours for Case3 (t_1=60min)

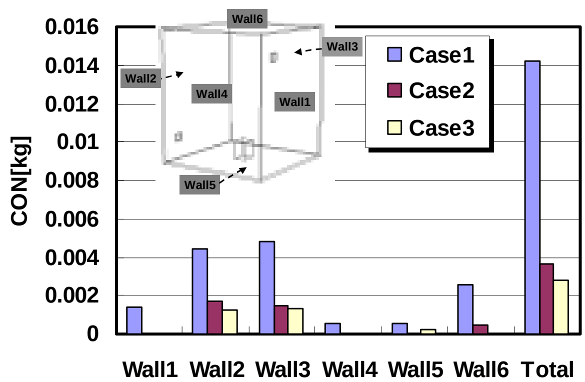

Figure 7 SUMCON (t_1) and the total value for the three cases (t_1=60min)

REFERENCES

Aizawa, Y., Liu, J., Yoshino, H. 2002. Study on Indoor Humidity Distributions, The 23rd Conference of the AIVC and the 3rd European Conference on Energy Performance & Indoor Climate in Buildings, pp.201-206

ASHRAE. 1997. ASHRAE handbook - fundamentals, pp.5.11-5.12, Atlanta

Ekstrand-Tobin, A., Lindgren, S. 2000. Indoor Environment and Allergic Parameters before and after Renovation of Moisture Damaged Constructions, Proceedings of Healthy Buildings, Vol.3

Ikeda, T., Imanaka, M. 1995. Properties of Condensation and Effects of Hygroscopic Materials in a Dwelling House under Door Open and/or Close conditions, Nol.473, pp15-24 (in Japanese)

Kuba, T., Miyamura, H., Tarumi, H., Okazaki, T. 1997. Measurement of Surface Condensation Using Multi-spectral Scanner, The 6th International Symposium on Building and Urban Environment Engineering and Management, pp.53-59

Liu, J., Aizawa, Y., Yoshino, H. 2002. Experimental and Numerical Study on Simultaneous Temperature and Humidity Distributions, 8th International Conference on Air Distribution in Rooms, Roomvent'02, pp.169-172

Nielsen, K. F., Nielsen, P. A., Holm, G. 2000. Growth of Moulds on Building Materials under Different Humidities, Proceedings of Healthy Buildings 2000, Vol.3

TenWolde, A. 1993. Ventilation, Humidity, and Condensation in Manufactured Houses During Winter, ASHRAE Transactions 99(1), pp.103-115

NOMENCLATURE

c_p = specific heat at constant pressure, kJ/(kg*K)

$CON\ (s,\ t_1)$ = condensation rate per unit area on a wall surface at time t_1, kg/m^{2}s

$SUMCON\ (t_1)$ = accumulative condensation rate on a wall surface at time t_1, kg/s

D_m = moisture diffusion coefficient, m^2/s

h_D = moisture transfer coefficient, m/s

h_T = heat transfer coefficient, W/(m^{2}K)

Le = Lewis number, dimensionless

W_{a1} = absolute humidity of the air close to the wall surface, kg/kg'

W_{suf} = saturated absolute humidity according to the temperature of the wall surface, kg/kg'

γ = latent heat of moisture vapor, kJ/kg

ρ = air density, kg/m^3