SIMULATION OF TEMPERATURE AND AIR FLOW IN A SECOND SKIN FAÇADE

F. Di Maio and A.H.C. van Paassen

Laboratory for Refrigeration Engineering and Indoor Climate Technology, Delft University of Technology, Mekelweg 2, 2628 CD Delft. The Netherlands.

ABSTRACT

The Second Skin Facades can be an useful tool for increasing the efficient use of natural ventilation in order to decrease the energy consumption for ventilation and cooling purposes and to increase the indoor thermal comfort levels.
To analyze this type of facades a simulation is set up. The simulation is carried out with the simulation code Simulink.

KEYWORDS

Airflow, thermal balance, second skin façade, simulation, Simulink, stack effect, wind pressure.

INTRODUCTION

Nowadays techniques are developed for improving the buildings envelope to get better comfort with a lower energy consumption. Amongst these, the Second Skin Facades (SSF) can help the HVAC system to maintain the best conditions from an environmental point of view. In winter the Second Skin Facade can preheat the ventilation air (figure 1). In summer the SSF with blinds between the two skins is a robust shading system and together with night cooling through natural ventilation, it can save a lot of cooling energy.
In order to analyze these aspects a dynamic simulation of a four-floor building is carried out with the simulation code "Simulink". The model used in this simulation has been built up step by step, but the main framework can be considered as an interaction of two main subsystems: the ventilation model and the thermal model. In figure 2 a block diagram shows the set up of the simulation. Each subsystem is composed by a series of other more detailed subsystems. The blocks called "workspace" store the response of an output, so that it can be shown later on graphically.
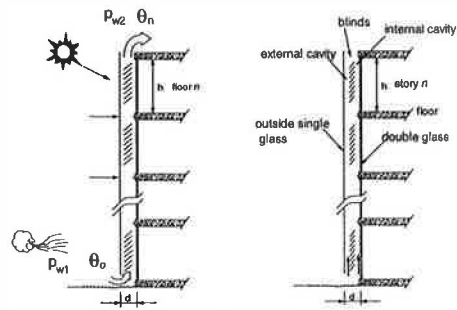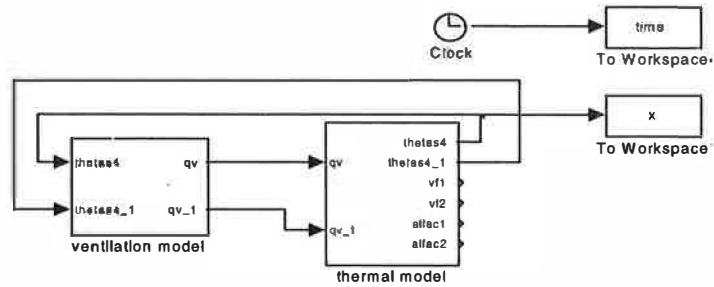
Figure 1: Second skin façade.

Figure 2: Simulink representation of the model

THE VENTILATION MODEL

The ventilation model should calculate the flows through the inlet and cavities of the second skin based on the outputs of the thermal model, the stack effect generator, the pressure generator and the wind generator.

Model hypotheses

Due to complex dynamics of the flow inside the cavity, simplification is necessary to built the model. To be able to calculate the flows through the inlet and cavities they will be represented by air resistances between two pressure points (see fig.3). The pressure differences due to stack and wind effects are indicated by the symbol normally used for pumps. The stack effect is caused by increase in air temperature in the cavities due to the absorption of solar radiation by the blinds between the two skins. Wind pressure differences are induced by the wind flow around the building.

Due to these driving forces air can overcome the resistance of inlet, cavities and grids at each floor.

Decomposition into subsystem

The system has four subsystems: pressure generator, flow generator, stack generator and wind generator. The input of the pressure generator are the flows from the flow generator. The pressures are found by solving the following differential equations:
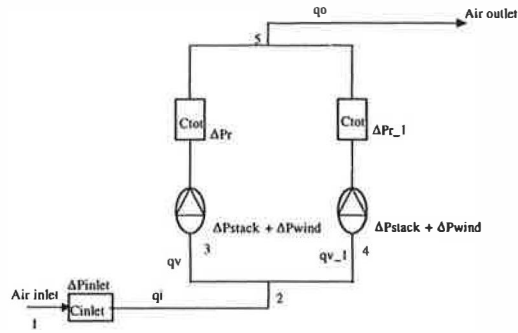

Figure 3. Flow model.

$$\frac{dp_2}{dt} = k_1 \left(q_i - q_v - q_{v\_1} \right) \tag{1}$$

$$\frac{dp_s}{dt} = k_2 \left(q_v + q_{v\_1} - q_0 \right) \tag{2}$$

With: p_2 = pressure in point 2 (see figure 3) [Pa]; p_5 = pressure in point 5 (see figure 3) [Pa]; $k_1 = k_2 = Pa/V = cost.$ [Kg m^{-4} s^{-2}]; Pa = static pressure [Pa]; V = cavities 's volume [m^3]; q_i, q_v, q_{v_1}, q_0 = flows in flow model (see figure 3).

The inputs of the flow generator are the pressure from the pressure generator, the stack effect from the stack effect generator, the resistance coefficient and the outside wind pressure from the wind generator. The outputs of the system are the airflows through the inlet, the cavities and outlet. Outputs are found by solving the following four equations:

$$q_i = C_{inlet} \sqrt{p_1 - p_2} \tag{3}$$

$$q_0 = q_v + q_{v\_1} = q_i \tag{4}$$

$$q_v = C_{tot} \sqrt{p_2 - (p_5 - \Delta p_{tot})} \tag{5}$$

$$q_{v\_1} = C_{tot} \sqrt{p_2 - (p_5 - \Delta p_{tot\_1})} \tag{6}$$

With: q_i = flow $[m^3 s^{-1}]$; C_{inlet} = inlet resistance coefficient $[-]$; C_{tot}= total resistance coefficient of internal and external cavity; Δp_{tot} = total pressure difference in the external cavity $[Pa]$; Δp_{tot_1}=total pressure difference in the internal cavity $[Pa]$.

The resistance coefficient due to the inlet and the grid in each floor, can be computed by making use of the general formula:

$$C = \sqrt{\frac{1}{\zeta}\frac{2}{\rho} A_{eff}^2} \tag{7}$$

With: ζ = flow resistance coefficient $[-]$; ρ = air density $[Kgm^{-3}]$; Aeff = effective (free) open area $[m^2]$.

For example for the inlet:

$$C_{inlet} = \sqrt{\frac{1}{2.5}\frac{2}{1.2}\left(0.7 A_{inlet}\right)^2} \tag{8}$$

The total pressure differences Δp_{tot} and Δp_{tot_1} can be caused by difference in wind pressure between the inlet and the outlet of the cavity and/or differences between the air temperature in the cavity and the external air temperature.

The pressure difference due to the wind can be computed as follows [Swami,1994]:

$$\Delta p_{wind} = P_{w1} - P_{w2} = \left(C_{p1} - C_{p2}\right)\frac{1}{2} \rho\, v_w^2 \tag{9}$$

With: P_{w1} = static wind pressure on the bottom of the building [Pa]; P_{w2} = static wind pressure on the roof of the building [Pa]; ρ = air density $[Kgm^{-3}]$; v_w = wind speed $[ms^{-2}]$; C_{p1} = pressure coefficient on windward side = 0.6, on leeward side = -0.3; C_{p2} = pressure coefficient on the roof of the building = -0.3. The pressure difference due to the chimney effect can be computed (one for each cavity) using the following formula [Liddament,1996]:

$$\Delta p_{stack} = \rho \cdot g \cdot n \cdot h \cdot \left(\frac{273 + \theta_2}{273 + \theta_1} - 1\right) \tag{10}$$

With ρ = specific density $[Kgm^{-3}]$; g = gravity acceleration (9.81) $[ms^{-2}]$; n = number of floors $[-]$; h = floor height $[m]$; θ_1 = temperature of outside air $[°C]$; θ_2 = air temperature in the cavity $[°C]$.

The outputs from the ventilation model are used in the thermal model to compute the temperatures in the double facades.

THE THERMAL MODEL

The thermal model is able to compute the temperatures of all points in the thermal network indicated in Fig. 4. This is possible by solving the differential equations describing the thermal balance in all point of the network.

The inputs required to make these calculations are the outdoor weather data, the convective heat transfer coefficient and the flows from their block generator.

Model hypotheses

Similarly to the ventilation model, simplifications are necessary to built the thermal model. As shown in Fig. 4 the thermal model is represented like a thermal network where the driving forces are temperature differences. To calculate the temperatures at the various nodes. heat balances are set up for each node. For example for the node S_2 in the second floor:

$$\frac{d\theta_{s2}}{dt} = \frac{1}{c_u \rho_a d_s A} \left[-\left(2\alpha_{c1} A + \rho_a c_a q_v\right)\theta_{s2} + \alpha_{c1} A\theta_1 + \alpha_{c2} A\theta_2 + \rho_a c_a q_v \theta_{s1} \right]$$ (11)

With: o_{s2} = air temperature in point s2 [°C]; c_a= air specific heat [J Kg^{-1} K]; ρ_a=air density [Kg m^{-3}]; d_s=cavity depth [m]; A= surface area of the window [m^2]; α_{c1}= convective heat transfer coefficient [W/m^2 K]; q_v= air flow [m^3/s]; o_1= glass temperature in point 1; o_2=blind temperature in point 2; o_{s1}= air temperature in point s1.

Three types of heat flows are present: conduction, convection and radiation. The conductive heat flow from the surfaces, will be modeled as a 2-D steady-state heat flow. About the convective heat flows from the surfaces to the air flowing in the cavities as well as through the airflows in the cavities, the following simple formulas have been used:
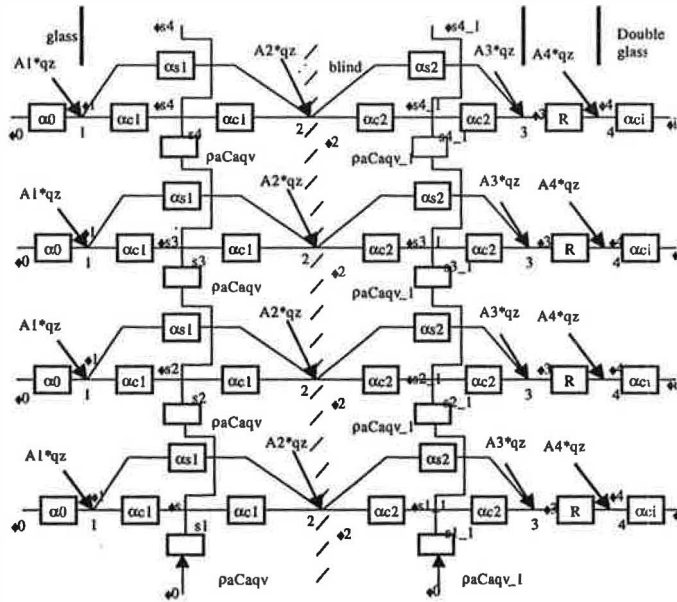

Figure 4. Thermal network

$$Q_{convection} = \alpha_{c1} A \left(\theta_i - \theta_{si}\right)$$ (12)

$$\dot{Q}_{convection} = \rho_a c_a q_v \left(\theta_{si} - \theta_{si+1}\right)$$ (13)

With: α_{ci} = convective heat transfer coefficient [W/m²K]; A = surface area of the window [m²]; θ_i = temperature of the surface at point i [K]; θ_{si} = temperature of air in points si [K]; ρ_a = air density [Kg/m³]; c_a = specific heat of the air [J/Kg K].

To describe the heat flow between surfaces, the following formula has been used:

$$Q_{radiation} = \alpha_s \; A \; (\theta_i - \theta_{i+1})$$

With: α_s = radiation heat transfer coefficient [W/m²K].

To take into account the solar radiation load absorbed by the glass and the blind, it is necessary to multiply the solar radiation falling on the facade q_z by solar radiation factors A_1, A_2, A_3, A_4.

Decomposition into subsystems

For the four-story building there are five subsystems: one for each floor and one for computing the convective heat transfer coefficients. Those coefficients are calculated in the alfa generator subsystem. The inputs for each subsystem are the flows from the ventilation model, the temperatures of the lower floor subsystem and the convective heat transfer coefficients from the alfa generator subsystem. The outputs are air temperatures in each point of the thermal network.

The inputs for the alfa generator subsystem are the flows in each cavity calculated by the ventilation model. The outputs are the convective heat transfer coefficients and the air speeds in the cavities. This is possible by solving the linear approximate equation:

$$\alpha_{ci} = (-7{,}52 \, d_{si} + 8{,}68) v_{si} - 0{,}448 \, d_{si} + 0{,}542$$

With: $v_{si} = q \; / A_{eff}$ [m/s]; d_{si} = width of cavity [m]; v_{si} = speed in the cavity [m/s];

The simulation have been carried out with a second skin facade with a cavity height of four floors of 13.2 m and width 3.6 m but different cavity's depths: $d_{f1} = 0.1$ m, $d_{f2} = 0.4$ m. The solar radiation is $q_z = 250$ Wm⁻², the indoor temperature $\theta_i = 20$ °C and the wind speed $v_w = 1.5$ ms⁻¹.

Figure 5 shows the simulation responses of the cavity temperatures on a step change of solar radiation (q_z=250 W/m^2) falling on the façade. Two responses are shown for the cavity depth of 0.1 m and 0.4 m. The temperatures reach different values as well as the responses of the systems are different.

The continuous line describes the system with smaller depth while the dotted line represent the system with the bigger one. The dynamic behaviour shows to be different.
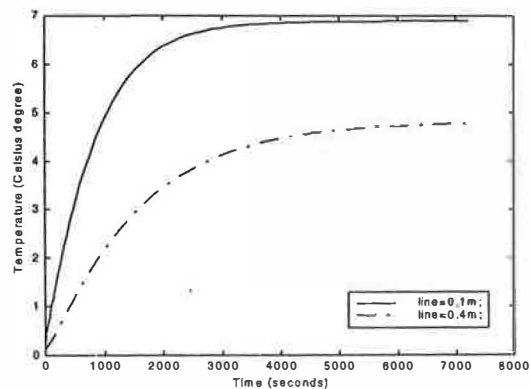

Figure 5. Air temperature of internal cavity at the highest point

CONCLUSIONS

- A simple simulation of the double skin facade can be derived from the heat balances and airflow models.
- As an example the effect of the depths of the cavity has been shown.
- It can be used to analyze the impact of the dimensions of second skin façade on heating, cooling and ventilation.

References

Paassen, van A.H.C. and Voorden, van der M. (2000) *Development of simplified tools for evaluation energy performances of double facades.* International Building Physics Conference Tools for design and engineering of buildings.

Groningen B.H., Liem S.H., Paassen A.H.C. (1998). *Control of night cooling with natural ventilation,* 19th annual AIVC conference, Oslo, Norway.

Chandra M.V. and Swami M.V. (1994). *Correlations for pressure distribution of buildings and calculation of natural ventilation airflows.* ASHRAE Transactions 94(1).

Liddament M.V. (1996). *A guide to Energy Efficient Ventilation.* Air infiltration and Ventilation Centre, University of Warwick.