Design and analysis of a window-integrated passive system (WIPS) with the use of solar heat gains

Fatemeh Eftekhar1, Nari Yoon2, and Yeonsook Heo1*

1School of Civil, Environmental and Architectural Engineering, Korea University, Seoul, Republic of Korea
2School of Architecture, University of Ulsan, Ulsan, Republic of Korea

Abstract. As one of the passive building design strategies to decrease ventilation energy consumption, several window-integrated passive systems (WIPS) have been developed and implemented into buildings. This study proposed a new WIPS inspired by a double skin façade (DSF) design that provides pre-heating and ventilation by utilizing solar heat gains collected in the air cavities. The performance of the proposed WIPS was analyzed and developed based on different design parameters such as material, width, height, shape, depth, and opening area, for the wintertime in Seoul, Korea. Furthermore, the computational fluid dynamics model, Fluent 2021 R2 with the RNG k-epsilon turbulence model, was used for the simulation. To focus on the buoyancy effect occurring in the air cavities, the influence of wind was excluded from the CFD modelling and analysis. Therefore, 0 Pa pressure differences between the inlet and outlet of the room were applied. The results showed that the buoyancy effect increased in the WIPS with a wider and higher inflow cavity than a case with narrower and shorter cavity, also utilizing glass material for the cavity surfaces exposed to the outside resulted in absorbing more solar radiation and more buoyancy effect. Moreover, the converged shape cavity increased the volume flow rate of the cavity due to the increased air velocity from the Venturi effect. Overall, all design parameters can impact the performance of WIPS either by hindering or assisting the airflow of the WIPS.

1 Introduction

Net zero energy and sustainable buildings have been considered a crucial strategy to reduce the energy consumption and pollution emissions from the building sector. One of key design strategies for sustainable buildings is the utilization of natural ventilation. Natural ventilation has been frequently applied in buildings to decrease energy consumption and release harmful environmental emissions [1]. Ventilation supply and indoor air quality maintenance can be accomplished by utilizing window-integrated passive systems (WIPS), also called supply-air or ventilated windows. WIPS offers an additional set of controllable components that allow fresh air supply in addition to existing window components [2]. One example of WIPS products is AEROMAT window ventilators manufactured by SIEGENIA company [3]. This product supplies the outdoor air by an additional fan connected to a heat exchanger to recover the heat. It also brings fresh air into a room in a controlled manner. Because the inlet and outlet openings of the product are located at the same height, ventilation relies on the pressure differences occurred by wind. In the absence of wind, system performance may be compromised. Therefore, they utilize a fan to bring fresh air into the system. AEROVENT window systems [4] is another commercial WISP product developed by ALTHERM company. This system differs from AEROMAT as it operates without an additional fan. However, the openings of this system is also located at the same height that the system relies on wind-driven effects. Another product of ALTHERM company is VENTIENT window system [5], which is different from AEROVENT in that it operates based on the air temperature for opening and closing the system. However, regarding ventilation, VENTIENT has also the advantage of the pressure differences because of wind-driven effect.

The existing products mentioned above mostly use a part of the window frame and locate the inlet and outlet openings at the same part of the frame that directly bring fresh air into a room, often with the aid of an additional fan. Moreover, they do not have the capability to utilize solar energy for passive heating and buoyancy-driven ventilation. One of the strategies for natural ventilation is the double-skin façade (DSF), which has attracted the attention of researchers in the building industry based on its environmental benefits. A DSF harnesses the airflow of the cavity due to the difference in the temperature between the indoors, air cavity, and outdoors. The temperature difference creates a buoyancy force, which is used in the DSF to exhaust indoor air or supply fresh air [6]. However, installing DSF, which has been developed to decrease the energy demand of buildings, requires a high initial cost [7]. This study investigates the proof-of-concept study of a new WIPS design on the basis of key design principles.
of DSF, which can potentially decrease the initial cost of implementing passive systems instead of using the double-skin façade. Instead of another glazing layer in front of the whole building façade, small-scale air cavities are integrated into a window system as inlet and outlet cavities, separately. The proposed WIPS allows for providing a certain level of pre-heating and ventilation using solar heat gains on the basis of the buoyancy effect without mechanical devices by adopting the key design principles of new DSF proposed in Yoon, Min, and Heo [8].

2 Concept of WIPS and in winter

A recent study on a dynamically compartmentalized DSF [8] proposed a new double-skin façade system that dynamically changes the partitioning of the air cavity by season to fully exploit pre-heating and natural ventilation effects depending on the seasonal needs. During winter, the outdoor air is preheated and directed into the room by an inflow cavity (lower cavity in Fig. 1), while the used air is exhausted through an outflow cavity (upper cavity in Fig. 1).

Fig. 1. Diagram of the double-skin façade for wintertime: lower and upper cavities for inflow and outflow cavities, respectively [8].

On the basis of the winter model described in Fig. 1, we designed a WIPS that harnesses the pre-heating and natural ventilation while the size of the system is more compact than the DSF system, as shown in Fig. 2.

The proposed WIPS consists of two small-scaled air cavities, each of which serves as an inflow and outflow cavity. The inflow and outflow cavities are installed on the right and left sides of a window, respectively, as shown in Fig. 2. Each cavity has two openings: one at the exterior side and the other at the interior side. In the inflow cavity, the exterior side opening is placed at the bottom of the cavity (Fig. 3 (a)), and the interior side opening at the top of the cavity (Fig. 3 (b)). In the outflow cavity, the position of the exterior and interior openings are placed in opposite ways, that is, the exterior side opening at the top of the cavity (Fig. 3 (d)), and the interior side opening at the bottom of the cavity (Fig. 3 (c)). When the outdoor air enters the bottom inflow opening (Fig. 3 (a)), it is pre-heated while passing through the cavity and reaches the top opening of the inflow cavity (Fig. 3 (b)). After having circulated the room, the room air exits via the bottom interior opening of the outflow cavity (Fig. 3 (c)), and, finally, it is exhausted out through the top exterior opening of the outflow cavity (Fig. 3 (d)).

Fig. 3. Air passage through WIPS ((a): exterior inflow opening, (d): exterior outflow opening, (c): interior outflow opening, (b): interior inflow opening).

As a starting point, we consider the exterior and interior openings of the cavities to have the same dimensions as described in Table 1. Cavities are distinguished by the location of the cavity and its openings. As the initial design, the outflow cavity is positioned 1.4 m above the floor, while the inflow cavity located near the floor (Fig.2).

<table>
<thead>
<tr>
<th>Case</th>
<th>height (m)</th>
<th>width (m)</th>
<th>depth (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cavity</td>
<td>1.4</td>
<td>0.2</td>
<td>0.2</td>
</tr>
<tr>
<td>Cavity Opening</td>
<td>0.2</td>
<td>0.2</td>
<td>-</td>
</tr>
</tbody>
</table>
3 Modelling method

 Fluent 2021 R2 was used to solve the three-dimensional (3D) steady Reynolds-Averaged Navier–Stokes (RANS) equations with the RNG k-epsilon turbulence model for steady-state simulation. Our WIPS is designed to work mainly based on the buoyancy-driven effect. Therefore, the first step of analysis focuses on the ventilation performance due to the buoyancy without consideration of wind pressure. In other words, the influence of wind was excluded from the CFD model by (a) limiting the computational domain to the room size (Fig. 2 shows the computational domain size), and (b) considering 0 Pa pressure for pressure inlet (Fig. 3 (a)) and pressure outlet (Fig. 3 (d)) as boundary conditions. Incompressible and turbulent fluid with constant properties was selected to characterize the air within the domain.

The three-dimensional (3D) model was created using Ansys SpaceClaim modeling software. The face meshing method was applied to determine an appropriate analysis mesh for the cavity openings, and the body meshing method for the cavity units and the room. These meshing methods are on the basis of tetrahedron. Moreover, the grid-sensitivity analysis for three cases was performed by applying different element sizes. The results of the grid analysis confirmed that 0.04m, 0.06m, and 0.08m element sizes were sufficient for the cavity openings, cavity units, and room, respectively. Consequently, a total of 1,100,000 elements were generated and used for further simulation.

The initial conditions of the glass surface, room surfaces (ceiling, roof, east, west, and north walls), and south wall surface are fixed at 5°C, 15°C, and 22°C, respectively. The boundary conditions of the free stream temperature was specified as 5°C to represent 12 pm in February first. Additionally, a heat transfer coefficient of 6 W/m²K was set as the thermal properties of the glass used in both the cavity and the room. The radiation model employed for analysis was based on discrete ordinates (DO) and ray tracing solar model to reflect solar radiation distribution in Seoul at the selected time used for simulation. It is noted that additional internal heat sources such as occupants and plug-in equipment are not considered in this study. The converged results for the first day of February at 12 pm were obtained when the scaled residuals reached 10^-6 for continuity, 10^-4 for intensity, energy, x, y, and z-velocity, and 10^-5 for k and epsilon.

4 Airflow performance

This section investigates various design variables that can affect the overall performance of the WIPS. Design parameters considered in this study include material, cavity width, cavity height, cavity shape, cavity depth, and opening area. Several design alternatives associated with each design variable are developed and analyzed in the process of updating the initial design into a final design proposal. Since the fluid is considered incompressible with a constant density, the mass flow rate can be expressed as the multiplication of the volumetric flow rate and constant density. Thus, the volumetric flow rate (m³/s) is evaluated for analysis as it is used as inputs to building energy simulation models such as EnergyPlus.

4.1 Material

The WIPS material directly affects the room insulation and usable solar heat gain. Therefore, three different combinations of aluminum, glass, and unplasticized vinyl chloride remain (UPVC) were tested as provided in Table 2. In Case 1, the exterior aluminum was considered as it can be easily heated by solar radiation and increase the cavity air temperature by convection. In Cases 2 and 3, the exterior glass was considered as it allows solar radiation to enter the cavity and increases the surface temperature of the cavity. Aluminum was considered as another alternative as the high thermal conductivity of aluminum can quickly transfer absorbed solar heat into the air cavity. In addition, black-colored aluminum is assumed to have the solar absorption of 0.9. Except the exterior side, UPVC with a low thermal conductivity (W/mK) is considered to perform as an insulator for the room. It is noted that an internal emissivity of 0.9 and an absorptivity of 0.8 was considered in this study. Fig. 4 shows a view of the section from the cavities and its surfaces in the room. Parts e, f, g, and h in Fig. 4 show the cavity surface, which in Table 2 demonstrates the applied materials to these parts.

![Fig. 4. Parts of the system in relation to the room: e is the surface exposed to the outside, and f, g, and h are the cavity surfaces that are faced inside the room.](image)

<table>
<thead>
<tr>
<th>Part</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>e</td>
<td>Aluminum</td>
<td>Glass</td>
<td>Glass</td>
</tr>
<tr>
<td>f</td>
<td>UPVC</td>
<td>UPVC</td>
<td>Aluminum</td>
</tr>
<tr>
<td>g</td>
<td>UPVC</td>
<td>UPVC</td>
<td>UPVC</td>
</tr>
<tr>
<td>h</td>
<td>UPVC</td>
<td>UPVC</td>
<td>UPVC</td>
</tr>
</tbody>
</table>

Fig. 5 illustrates the results of the volume flow rate (m³/s) of Cases 1 to 3. The hatched and solid columns in this figure show the volume flow rate of the inflow and outflow cavities, respectively. Moreover, positive values (black color) indicate the intended flow rate whereas minus values (gray color) indicate the unintended volume flow rate in the unintended direction of the intended flow. Results show that using glass in Cases 2 and 3 for the external cavity surface improves...
the WIPS performance by reducing the unintended flow compared to Case 1 with aluminum. Case 1 showed the highest net volume flow rate but in an unintended direction. In contrast, Cases 2 and 3 showed a much smaller flow rate, but the magnitude of unintended flow was reduced by using glass as the exterior surface material. In both cases, the net volume flow rate was quite similar, 0.001 (m$^3$/s). Between Cases 2 and 3, as UPVC has lower thermal conductivity than aluminum, it functions as a better insulator for the room. Therefore, we selected Case 2 for further simulations.

The results (Fig. 6) illustrate that a wider inflow cavity width resulted in a higher net volume flow rate (Cases 5 and 6). When the inflow and outflow cavity widths were the same as in Cases 3 and 5, the intended and unintended airflows of each case were in a similar range, resulting in a low net volume flow rate. However, in Case 6, when the inflow cavity width is four times wider than the outflow cavity’s width, the unintended flow rate in the inflow cavity was reduced substantially. The unintended flow of 0.002 m$^3$/s still exists in the outflow cavity while the net volume flow rate is at the peak (0.003 m$^3$/s) (Fig. 6). Therefore, we selected Case 6 for the next step.

### 4.2 Width

Under the selected materials (in Case 2), we tested various cavity widths that could improve the system ventilation performance by absorbing more solar heat. Table 3 shows the width of the cavities for four cases, while the height, depth, and opening areas of the cavities were fixed at 1.4 m, 0.20 m, and 0.04 m$^2$, respectively.

<table>
<thead>
<tr>
<th>Case</th>
<th>Inflow cavity width [m]</th>
<th>Outflow cavity width [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.20</td>
<td>0.20</td>
</tr>
<tr>
<td>4</td>
<td>0.10</td>
<td>0.20</td>
</tr>
<tr>
<td>5</td>
<td>0.80</td>
<td>0.80</td>
</tr>
<tr>
<td>6</td>
<td>0.80</td>
<td>0.20</td>
</tr>
</tbody>
</table>

### 4.3 Height

The height of each cavity can influence the system’s buoyancy performance as it determines how the fresh air absorbs solar heat while passing through the cavities. Therefore, in Case 7, the height of the inflow cavity increased from 1.4 m to 2.8 m to potentially increase the buoyancy effect in the inflow cavity. Table 4 shows the considered heights and widths for simulations.

<table>
<thead>
<tr>
<th>Case</th>
<th>Inflow cavity’s height [m]</th>
<th>Outflow cavity’s height [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1.40</td>
<td>1.40</td>
</tr>
<tr>
<td>7</td>
<td>2.80</td>
<td>1.40</td>
</tr>
</tbody>
</table>
By doubling the inflow cavity height from 1.40 meters to 2.80 meters, the net volume flow rate also increased from 0.003 m³/s to 0.006 m³/s (Fig. 7). Furthermore, in Case 7, both cavities showed an increase in the intended volume flow rate, while the outflow cavity showed a substantial decrease in the unintended volume flow rate. This higher net airflow with much reduced unintended flow can be explained due to the higher buoyancy effect in the inflow cavity because of the higher height. Therefore, Case 7 was selected for further analysis.

In addition to the size of air cavity, the shape of air cavity can impact the pressure distribution within the cavity and, consequently, volumetric flow rate. For incompressible fluid, the Venturi effect states that fluid pressure decreases when passing through a constricted section of the pipe, which increases air velocity; vice versa for a non-constricted section. Li, Luo, Sandberg, and Liu [9] studied passage ventilation between two buildings with respect to the Venturi effect. Their results showed that a converging passage could promote ventilation more than diverging passage. In order to reflect the Venturi effect, we designed Case 9 illustrated in Fig. 8, which has a converging air cavity with tilted surfaces, which can potentially increase the air velocity based on the Venturi effect. We also created the other case, Case 8, shown in Fig. 8, which has a diverging air cavity (increasing the cavity area from the inflow to the outflow), which may increase pressure differences in the cavity.

The results of shape analysis in Fig. 9 show that having a diverging inflow cavity of WIPS, resulted in higher unintended flow in the outflow cavity and, consequently, decrease the net volume flow rate. However, in Case 9, having the converging inflow cavity increased the net volume flow rate to 0.008 (m³/s), while no unintended flow was observed in the WIPS. Hence, the shape of Case 9 was considered for the next analysis.
4.5 Depth

It should be considered that by changing the shape of the inflow cavity, the system has surpassed the edge of the wall and intruded into the indoor zone. Therefore, to decrease the intrusion of the inflow cavity in the area of the room, the designed WIPS system has been analyzed by having less depth for the inflow cavity. Fig. 10 shows the dimension of the room sections, including the inflow cavity with different depths. Case 9 is the designed system in the last section, and depth is decreased by 10 (cm) in Case 10.

![Diagram showing dimension of room sections with different inflow cavity depths](image)

**Fig. 10.** Section from the inflow cavity shows the dimension of the cavity’s depth at the bottom and top.

Fig. 11 illustrates the volume flow rates of the two cases with varying depth. The results show that decreasing the depth decreased the volume flow and rate and dropped the value to 0.007 (m³/s) by creating unintended flow in the outflow cavity. Hence, Case 9 was selected for further analysis.

![Graph showing volume flow rates for cases with different cavity depths](image)

**Fig. 11.** Net volume flow rate (m³/s), intended volume flow rate (+) (the flow in the intended direction for each cavity) and unintended (-) (the flow in the unintended direction for each cavity) for cases with different cavity depths.

4.6 Opening area

So far, all the opening areas of inflow and outflow cavities were set to be the same (0.04 m²) for all the previous cases. In order to analyze the effect of opening area, the opening area of the inflow cavity was increased from 0.04 m² to 0.08 m². Tecle, Bitsuamlak, and Jiru [10] investigated low-rise building natural ventilation and found that the ventilation rate is 1.5–2.5 times higher when opening ratio (inlet to outlet) is more than 1 than when the ratio is less than 1. Therefore, this study investigated the opening ratio by adding another case of increasing the opening size of the inflow cavity only. Table 5 shows the dimension of the openings for the two cases with different opening areas associated with the inflow air cavity. It should be noted that for each cavity, the interior and exterior openings are considered identical.

<table>
<thead>
<tr>
<th>Case</th>
<th>Inflow cavity opening size [m]</th>
<th>Outflow cavity opening size [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Width</td>
<td>height</td>
</tr>
<tr>
<td>9</td>
<td>0.80</td>
<td>0.05</td>
</tr>
<tr>
<td>11</td>
<td>0.80</td>
<td>0.10</td>
</tr>
</tbody>
</table>

**Table 5.** Dimensions of openings for each cavity.

![Graph showing volume flow rates for cases with different cavity opening areas](image)

**Fig. 12.** Net volume flow rate (m³/s), intended volume flow rate (+) (the flow in the intended direction for each cavity) and unintended (-) (the flow in the unintended direction for each cavity) for cases with different cavity’s opening area.
5 Thermal performance

In addition to the flow rate through the cavities, this section investigates the preheating effect of the air cavities in terms of the temperature difference between the air at the lower part of the cavity and the upper part. Fig. 13 shows the sectional areas in the cavity used to calculate the area-weighted average of air temperature from the CFD results. In short, the temperature differences were calculated for the inflow cavity based on sectional areas (p)-(s) and for the outflow cavity based on sectional areas (m)-(n).

Fig. 13. Sectional areas used to calculate the WIPS thermal performance.

Fig. 14 illustrates the results of temperature differences in the inflow cavity and outflow cavity. Results show that the preheating effect was the least in Case 1 and Case 4, with aluminum material used for the outside surface and smaller cavity width (inflow to outflow), both of which resulted in less solar heat in the inflow cavity. Furthermore, Cases 1 - 5 showed negative temperature differences in the outflow cavity, which resulted from the unintended flow from the outside. Case 8 shows the highest preheating effect in the inflow cavity, followed by Case 7. In comparison to Cases 7 and 8, Case 9 shows a reduced pre-heating effect because Case 9 creates a higher air flow rate between the outside and inside than the other cases. Regarding the airflow direction, Case 9 shows a consistent airflow pattern with no unintended flow in the outflow cavity, unlike the other two cases that show noticeable unintended flow in the outflow cavity.

6 Conclusion

This paper presented a new WIPS and conducted computational fluid dynamic (CFD) simulations to evaluate the influence of design parameters (width, height, shape, depth, and opening area) of WIPS. Zero (Pa) pressure difference was applied to the pressure inlet and outlet boundary condition to exclude the wind effect. The 3D steady RANS approach was employed with the simulation’s RNG k–epsilon model. The results showed that all considered design parameters could either assist or hinder the airflow of the WIPS.

Analyzing materials with different transparency and conductivity shows that by utilizing glass material for the exterior surface, and UPVC material for other cavity’s surfaces, the system can gain more solar heat while UPVC acts as an insulator for the room. Furthermore, by analyzing the width and height of the system, it revealed that using 0.8 (m) width and 2.8 (m) height for the inflow cavity compared to the outflow cavity with 0.2 (m) width and 1.4 (m) height can increase the volume flow rate to 0.006 (m³/s). Also, analyzing the shape of the cavity in Case 9 illustrates that by applying a converged shape to the inflow cavity, the velocity of the air increased based on the Venturi effect and increased the volumetric flow rate to 0.008 for WIPS. The thermal performance of WIPS illustrates that unintended flow will decrease the heat gain in the cavity. Case 8, with the diverged shape, shows the greatest buoyancy effect, while Case 4, with a wider outflow cavity than the inflow cavity, showed the lowest effect.

This paper focused on the winter operation using 0 (Pa) pressure difference and assumed that the testing...
room is located on the ground floor. The buoyancy-driven air flow within the air cavities may be influenced substantially by the wind direction and speed. If the WIPS is installed in the façade at higher floors, the surrounding atmospheric air pressure would be higher than the case settings in this study. A further study is under way to examine the WIPS performance under different pressure differences due to varying wind directions, speeds, and heights of installation.

This work is supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT) (No.2020R1A5A1018153) and the Korea Agency for Infrastructure Technology Advancement (KAIA) grant funded by the Ministry of Land, Infrastructure and Transport (Grant KAIA22HSCT-C157909-03).

References

3. Siegenia company, AEROMAT VT with heat recovery, siegenia, German.