

EXPERIMENTAL INVESTIGATION OF CAVITY FLOW UNDER BUILDING INTEGRATED PHOTOVOLTAIC PANELS USING THERMOGRAPHY AND PARTICLE IMAGE VELOCIMETRY

Carmeliet J.*, Paterna E.¹ and Mirzaei P.A.²

*Author for correspondence

Chair of Building Physics,
ETH Zurich,
Zurich, 8093,
Switzerland,

E-mail: carmeliet@arch.ethz.ch

¹WSL, Davos, CH ²University of Nottingham, UK

ABSTRACT

An extensive experimental investigation demonstrates the impact of cavity airflow underneath photovoltaic (PV) panels integrated in the roof assemblies of buildings. The benefit of underside ventilation is seen in terms of an increased efficiency of photovoltaic panels due to lowering their operating temperature, resulting in less turn-off times as well as an improved hygrothermal and durability behavior of the panels.

We perform an extensive measurement campaign of the surface temperature using infrared thermography and of the airflow using particle image velocimetry. A novel setup was developed consisting of a building model with a mock PV panel and a solar simulator placed inside a large-scale atmospheric wind tunnel. A solar simulator is positioned in the tunnel to provide a range of various radiation intensities over the panels and the approaching upstream wind is well controlled in the wind tunnel. The top surface temperatures and air speeds above and below the panel are monitored simultaneously.

It is shown that, in general, the airflow within the cavity is faster compared to the free upstream air velocity, resulting in an increased heat exchange between the PV and the air cavity and a reduction of the PV surface temperatures. A stepped open arrangement of panels is shown to be more effective in reducing the surface temperatures comparing to a flat arrangement.

The results also show the presence of different interacting flow phenomena: natural convection due to irradiation, forced convection due to the upstream wind, cavity ventilation and surface convection, as well as the presence of complex 3D flows patterns (e.g. lateral eddies), which contribute to a highly non-uniform surface temperature distribution over the PV modules.

INTRODUCTION

Photovoltaic (PV) is one of the most efficient and environmentally clean technology to convert the intercepted solar energy to electrical energy. Its installation and maintenance is very easy, it has vast applications (e.g. water pumping, communications, satellites, power plants, etc.), and also a huge potential to produce up to megawatts of electricity.

Among the different adopted PV technologies, building integrated photovoltaic (BIPVs) is an emerging and popular technology for harnessing solar energy. In the BIPV sector, cheaper thin films are less popular due to lower efficiency, rapid performance degradation and shorter life time. However, the efficiency of c-Si PV panels drops at temperatures above 25 °C. Various correlations have been proposed for estimating this efficiency drop, for example 0.4-0.65%/K is reported by [1,2].

A simple cooling technique to avoid high surface temperatures is back-ventilation, inducing airflow in the cavity beneath the PV panels. The performance of this technique has been widely studied using numerical techniques such as computational fluid dynamics (CFD) [3,4].

This study simultaneously monitored the airflow above and below the PV panels. It was also investigated how airflow can decrease the surface temperature of the PV panels and hence improve performance, and how airflow can affect moisture ingress through weak points and interconnections of the PV panels. For the purpose of understanding further this behavior, a novel experimental setup was developed consisting of a building model with mock BIPV panels placed in a closed-loop atmospheric boundary layer wind tunnel, a solar simulator, a PIV system and an infrared camera for thermography of the surface temperature of the PV panels. The purpose of heating up the PV panel with the solar simulator rather than using surface heater was to observe the simultaneous flow on both sides of the PV while both wind-driven and buoyant mechanisms are contributing in its cooling.

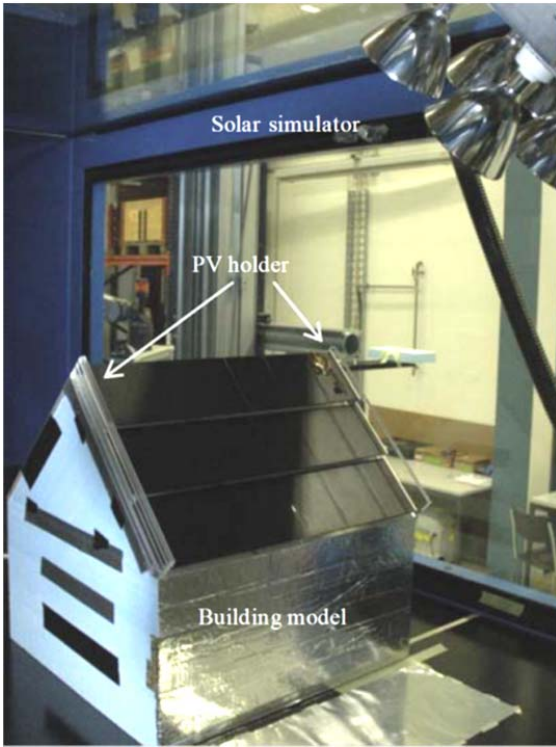


Figure 1 Photography of the building model and the solar simulator within the wind tunnel.

EXPERIMENTAL SETUP

A 1:20 scale building model was fabricated from extruded polystyrene board, as shown in Figure 1. The mock BIPV system was installed on the roof of the model. The thickness of the panel is 2 mm. The panels could be arranged as flat or stepped. A solar simulator was used to heat the mock BIPV panel up to 110°C. The solar simulator consisted of a 2 x 3 array of 250 W infrared lamps connected to a power supply allowing an adjustable heat flux. In each case, the surface of the panel was subjected to a radiation intensity of 200 W/m².

A solar simulator is positioned in the tunnel to provide a range of various radiation intensities over the panels and the approaching upstream wind is well controlled in the wind tunnel.

The PIV measurements were conducted using a high-speed 12 bit CMOS camera with a resolution of 2016 x 2016 pixels. The flow was seeded by particles produced by a Laskin-nozzle type atomizer.

The airflow around the panel was measured for various scenarios as follows:

- Flat and stepped PV configurations were examined
- The cavity was tested in open and closed configurations
- The upstream velocity was set to three values: 0.5, 1 and 2 m/s, corresponding respectively to Re numbers of 2×10^4 , 4×10^4 and 8×10^4 .

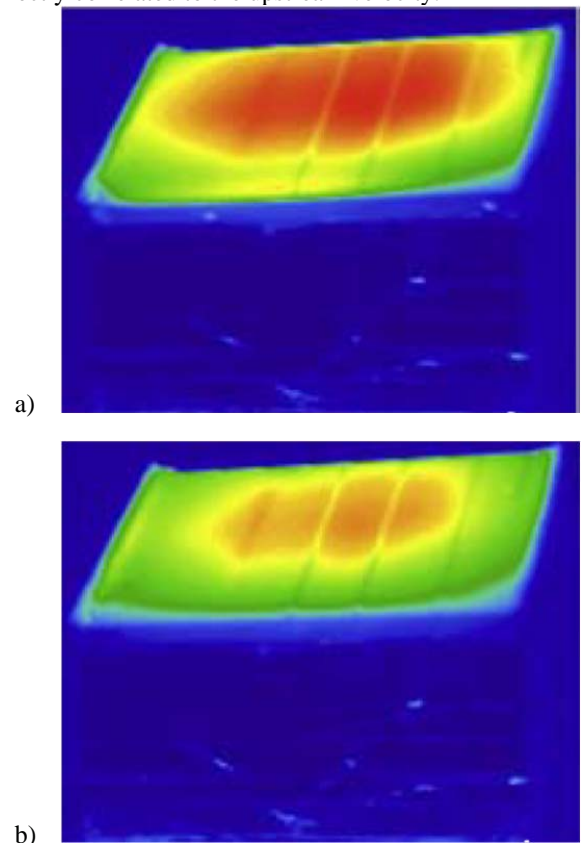
RESULTS AND DISCUSSION

Thermography

Figure 2 shows the surface temperature of the PV panels for closed and open cavity configurations, and stepped and flat arrangements of panels at the upstream air speed of 0.5 m/s.

In Figure 2a, the cavity was closed, thus the flow above the panel was the only means of convective heat exchange. The center of the roof is at quite high temperature and the edges are cooled significantly due to the air flow. In Figure 2b, we see the impact of adding an air cavity under the panels. The heat removal from the panel is increased due to this additional airflow. Obviously, lower temperatures can be obtained. With the cavity closed, the temperature distribution in the stepped arrangement was very similar to the flat arrangement, as shown in Figure 2c. For the stepped case with the cavity open, the surface temperature decreased dramatically, succeeding in the removal of the high temperatures in the central part of the roof, Figure 2d.

We also carried out experiments at the air speed of 1 m/s and 2 m/s (results not shown here). The maximum temperature decreases for higher upstream velocities as wind-driven convection increased considerably. The temperature decrease was directly correlated to the upstream velocity.



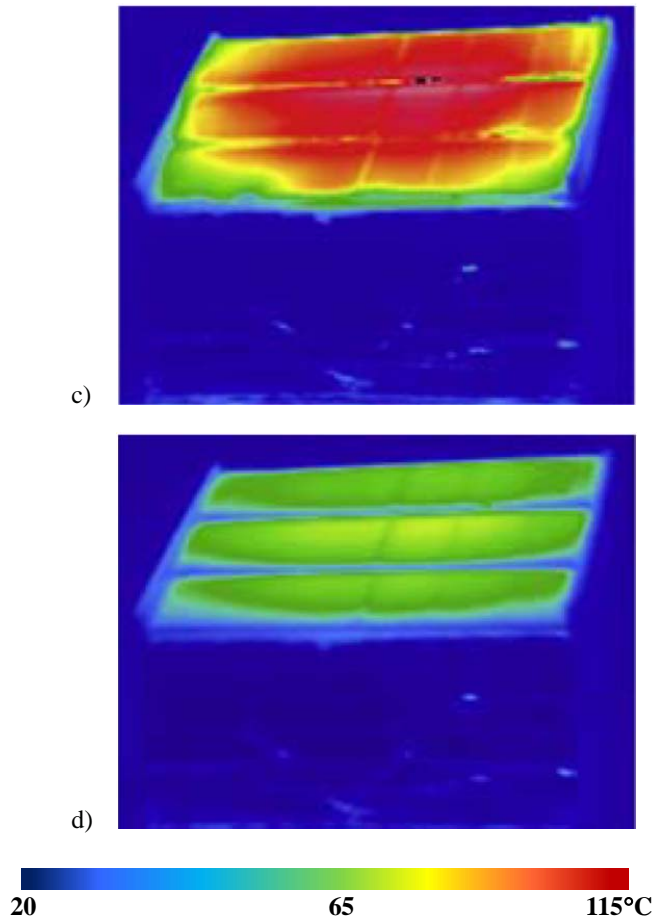


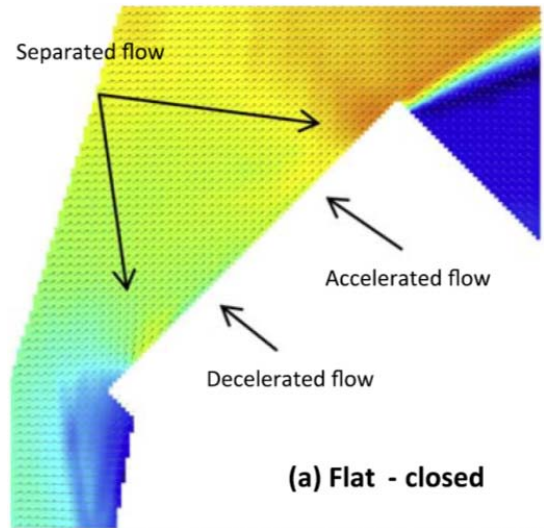
Figure 2 Thermography of the surface of the PV panels subjected to an upstream velocity of 0.5 m/s in four configurations: a) flat alignment and closed cavity, b) flat alignment and opened cavity, c) stepped panels and closed cavity, and d) stepped panels and open cavity.

Impact of the cavity

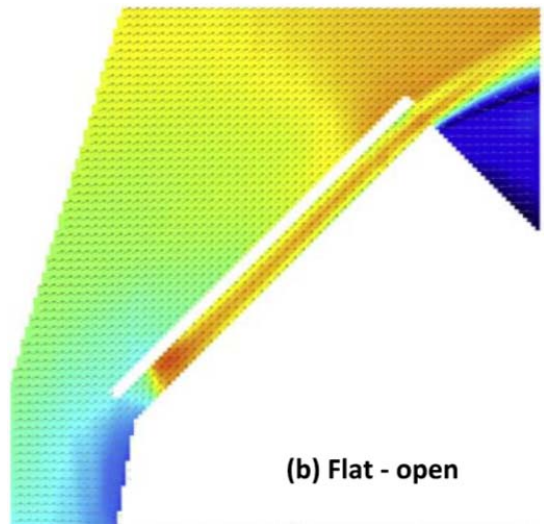
Figure 3 shows the mean velocity vector fields for upstream velocity of 0.5m/s for the four configurations, i.e. (a) flat, closed cavity, (b) flat, open cavity, (c) stepped, closed cavity, and (d) stepped, open cavity.

For all cases, the general flow pattern included two regions that appear stationary, at the windward wall of the building and leeward side of the inclined roof. The flow had a complex, turbulent 3D motion in these regions, which averaged to very low net flow.

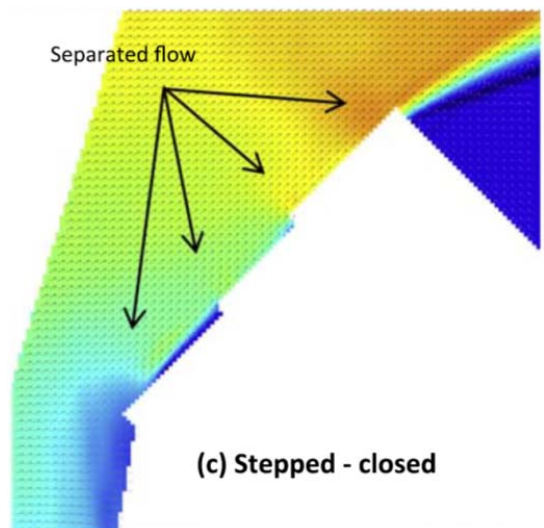
With the cavity closed, the flow separated from the leading and trailing edges of the panel. Small corner eddies occurred at the point of separation (shown with the arrows), which may have contributed to non-uniform temperatures on the panel surface. The degree of separation slightly increased for the higher upstream velocities of 1 and 2 m/s, not shown here.



(a) Flat - closed



(b) Flat - open



(c) Stepped - closed

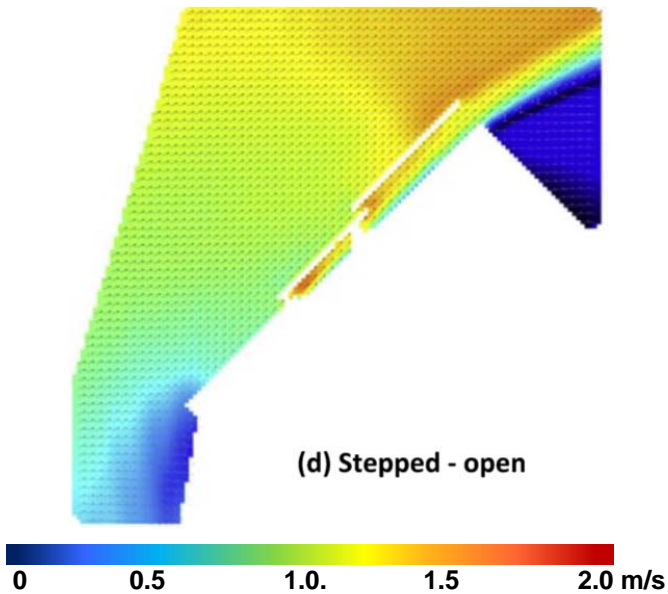


Figure 3 Particle image velocimetry results of the mean velocity vector fields for an upstream velocity of 0.5 m/s, for the four configurations.

CONCLUSION

The impact of the airflow regime on heat removal from PV panels was clearly demonstrated for both open and closed cavities. Airflow through the cavities can be between 1.25 and 1.35 faster than the upstream velocities. Consequently the surface temperature of the PV panels is considerable lower with an open cavity.

The findings of this study can be used to formulate recommendations for the installation of roof and façade integrated PV panels, including the promotion of cavity ventilation in order to reduce hygrothermal loading. The velocity field data can be further used to validate numerical models.

REFERENCES

- [1] Hasan M.A. and Sumathy K., Photovoltaic thermal module concepts and their performance analysis: a review, *Renew. Sustain. Energy Rev.*, Vol. 14, 2010, pp. 1845-1859
- [2] Kumar R. and Rosen M.A., A critical review of photovoltaic-thermal solar collectors for air heating, *Appl. Energy*, Vol. 88, 2011, pp. 3603-3614.
- [3] Corbin C.D. and Zhai Z.J., Experimental and numerical investigation on thermal and electrical performance of a building integrated photovoltaic-thermal collector system, *Energy Build.*, Vol 42, 2010, 76-82.
- [4] Yoo S.-H. and Manz H. Available remodeling simulation for a BIVP as a shading device, *Sol. Energy Mater. Sol. Cells*, Vol 95, 2011, 394-397.
- [5] Mirzaei P., Paterna E., Carmeliet J, Investigation of the role of cavity airflow on performance of building-integrated photovoltaic panels, *Solar Energy*, Vol. 107, 2014, pp. 510-522
- [6] Mirzaei P.A. and Carmeliet J., Influence of the underneath cavity on buoyant-forced cooling of the integrated photovoltaic panels in