Solar Energy 147 (2017) 151–163

Contents lists available at ScienceDirect

Solar Energy

journal homepage: www.elsevier.com/locate/solener

Prediction of the surface temperature of building-integrated photovoltaics: Development of a high accuracy correlation using computational fluid dynamics

Ruijun Zhang [a], Parham A. Mirzaei [a,*], Jan Carmeliet [b,c]

[a] Architecture and Built Environment Department, The University of Nottingham, Nottingham NG7 2RD, UK

[b] Laboratory for Building Science and Technology, Swiss Federal Laboratories for Materials Testing and Research (Empa), Uberlandstrasse 129, 8600 Dubendorf, Switzerland

[c] Swiss Federal Institute of Technology Zurich (ETHZ), Wolfgang-Pauli-Strasse 15, 8093 Zurich, Switzerland

ARTICLE INFO

Article history:
Received 2 August 2016
Received in revised form 6 March 2017
Accepted 8 March 2017
Available online 22 March 2017

Keywords:
Building
Photovoltaics
CFD
Cavity cooling
Wind-driven
Surface temperature

ABSTRACT

Building-integrated photovoltaic (BIPV) panels are generally expected to operate for over 25 years to be viewed as an economically viable technology. Overheating is known to be one of the major deficiencies in reaching the targeted lifespan goals. Alongside the thermal degradation, the operational efficiency of the silicon-based solar panel drops when the surface temperature exceeds certain thresholds close to 25 °C. Wind-driven cooling, therefore, is widely recommended to decrease the surface temperature of PV panels using cavity cooling through their rear surfaces. Wind-driven flow can predominantly contribute to cavity cooling if a suitable design for the installation of the BIPV systems is considered.

In general, various correlations in the form of $Nu = CRe^a$ are adapted from heat convection of flat-plates to calculate the heat removal from the BIPV surfaces. However, these correlations demonstrate a high discrepancy with realistic conditions due to a more complex flow around BIPVs in comparison with the flat-plate scenarios. This study offers a significantly more reliable correlation using computational fluid dynamics (CFD) technique to visualize and thus investigate the flow characteristics around and beneath BIPVs. The CFD model is comprehensively validated against a particle velocimetry and a thermography study by Mirzaei et al. (2014) and Mirzaei and Carmeliet (2013b). The velocity field shows a very good agreement with the experimental results while the average surface temperature has a 6.0 % discrepancy in comparison with the thermography study. Unlike the former correlations, the coefficients are not constant numbers, but a function of the airflow velocity, in the newly proposed correlation, which is in the form of $Nu_L = 0.1513Re_L^{0.7065}$.

© 2017 Elsevier Ltd. All rights reserved.

1. Introduction

The utilization of photovoltaics (PV) has been continuously growing within the power sector and shows a phenomenal increase among all renewable energy sources over the last five years (Renewable Energy Policy Network for the 21st Century (REN21), 2014). Building-integrated photovoltaics (BIPV) systems, in particular, are one of the most promising applications of solar power technologies and offer considerable potential in responding to building energy demands. Roof-mounted applications of BIPV are currently holding the dominant position in all BIPV markets with a share of 80%. The rest of the market is mainly focused on facade integrated technologies (Krawietz, 2011). A typical roof-mounted BIPV system is assessed to be capable of supplying 14.5–57.8% of a building's energy demands, depending on the local available solar yields, mounting geometry and climatic weather conditions (International Energy Agency (IEA), 2002).

The electrical performance of a solar power panel can be predicted from a linear expression with known reference data measured at standard testing conditions (STC) – where solar radiation is 1000 W/m^2 at ambient temperature, T_a = 25 °C. However, the efficiency of the silicon-based PV panels, as the dominant type of photovoltaic technology in the market, drops inversely with increasing cell temperature, also known as operating temperature. The decline ratio is addressed in many studies and is most likely to vary from 0.1 to 0.5%/°C (Skoplaki and Palyvos, 2009). This indicates that cooling is becoming an essential technique to maintain the BIPV electrical performance, especially in hotter climates.

A variety of strategies has been proposed to enhance heat removal from solar cells, including the circulation of water flow

* Corresponding author.
E-mail address: parham.mirzaei_ahranjani@nottingham.ac.uk (P.A. Mirzaei).

http://dx.doi.org/10.1016/j.solener.2017.03.023
0038-092X/© 2017 Elsevier Ltd. All rights reserved.

through the BIPV's front surface, utilization of hybrid systems with thermal collectors, and using forced ventilation through the cavity (Krauter, 2004; Enteria and Akbarzadeh, 2013). Natural winds around the stand-off mounted BIPV can also assist the cooling by placing a sufficient air cavity to remedy the lack of convection at the rear side. Skoplaki and Palyvos (2009) summarize the current analytical correlations used for the determination of the operating temperature.

The main challenge in predicting the thermal behavior of the BIPV corresponds to the complex airflow regimes around these panels. Many studies in this area have been carried out both experimentally and numerically. For example, the Nominal Operating Cell Temperature (NOCT) defined in nominal terrestrial environment conditions and the Sandia National Laboratories (SNL) model are two common mathematical correlations that have been developed from empirical datasets (King et al., 2004). The latter is preferable to the former as it encompasses both the wind effect and the solar radiation intensity (King et al., 2004). Nonetheless, both of these models prove weak in understanding the effects of wind direction and terrain characteristics on the wind profile as well as the influence of the cavity size of the mounted BIPV. D'Orazio et al. (2014) assessed these two models by comparing them with in situ experiments for three different roof installations: fully integrated and stand-off by 0.2 m and 0.04 m cavity sizes. The SNL model overestimates the back surface temperature in all scenarios. On the other hand, for all scenarios calculated by NOCT model, overestimations of the heat removal from rear side were found to be significant on sunny and breezy days while on a typical windy day, the predicted values were lower than measurements for stand-off BIPV. The largest deviations between the NOCT model and site measurement were 12 °C and 8 °C, respectively. The NOCT model was around 2.5% more accurate in its projection of the annual energy production in comparison with the SNL model. It was also recommended by D'Orazio et al. (2014) that a 0.04 m cavity gap is enough to supply sufficient cooling to the BIPVs in a typical Mediterranean climate.

Similar investigations on the effect of the cavity gap have been carried out in several simulation studies. For example, Guiavarch and Peuportier (2006) used a commercial tool, COMFIE, to test the dynamic performance of three different BIPV installation methods on roofs: rack mounted, stand-off and shingling without an air cavity. Mono-crystalline and amorphous silicon solar cells were examined in two climates, Paris and Nice, with a vertical façade application for a social residential building and an inclined roof application for a single family house. Annual PV productivity was forecast to have a 6% increase with the excess heat from the back ventilation employed for space preheating purposes. Shingling was found as the least preferable option having both low yields and efficiency. In another study, Mei et al. (2003) utilized a building energy simulation tool (TRNSYS) to model the thermal condition of façade integrated photovoltaic panel with forced air cavity ventilation. The intention was to use the air heated up in the cavity for heating purposes during winter time.

Computational fluid dynamics (CFD) was broadly used to investigate the cavity cooling, taking into account the BIPVs performance by detailed representations of velocity, temperature and turbulence fields. An example is shown in the research by Li and Karava (2012) where they recommended the Renormalization Group (RNG) k–ε model as turbulence model to provide a better overall performance in comparison with other turbulence models for an unglazed transparent collector with PV/T systems under forced convection. Controversially, Getu et al. (2014) indicated that although k–ε models could provide a more accurate prediction for temperature of the air between insulation located on the building prototype roof and the PV/T panel comparing with the k–ω model. The latter showed its strength in prediction of the temperature distribution. The utilized k–ε model was based on the assumption of the presence of high turbulence, which leads to less agreement with the experimental scenario conduced in lower airflow velocities. The drawback of k–ω model was mentioned to be its instability, depending on the free stream ω value generated by the leading-edge effect as discussed by Liao et al. (2007). Liao et al. (2007) conducted a CFD study to model the cavity cooling performance of a façade with integrated hybrid solar/thermal system. Experimental measurements were obtained using particle image velocimetry (PIV) for the validation of the CFD model. By using of computational results, a regression relation was proposed for the surface heat transfer (convection coefficient) in addition to a correlation with Nusselt number (*Nu*) against average air speed and cavity size. The predicted channel flow velocity, however, was higher than the measured values, also resulting into stronger predicted turbulence in comparison with the measurements.

Wilson and Paul (2011) ran a series of simulations for different air cavity sizes and tilt angles. The BIPV was tested by alteration of the tilt angle from 15° to vertical placement followed by nine cavity aspect ratios (cavity length to its height), ranging from 4.8 to 120, at upstream flow velocities of 0, 1, 2 and 3 m/s. The optimum mounting option for the BIPV system was found to be a 90° inclined panel with a large air cavity under buoyancy dominant ventilation. The maximum electrical efficiency was observed to be about 10.7–10.9% though this number could be further improved by 0.5–1% with mixed mode convection. A noteworthy observation was that the BIPV operating temperature was more sensible to inclination in the context of natural convection, but changed little under mixed cavity ventilation. In a similar study, Gan (2009a, 2009b) developed a CFD model to explore the thermal performance of the BIPV in different mounting geometries, including roof pitch, cavity size and number of PVs. Unlike the study by Wilson and Paul (2011), the flow regime was assumed to be turbulent rather than laminar. The conducted parametric study revealed that cavity cooling cannot be improved after a certain threshold for the air cavity size. Moreover, stepped multi-panels were recommended as a preferable arrangement to achieve better cavity air circulations in comparison with a long single panel. A high risk of hot spot occurrence near the top edge of the panels was also observed with a maximum temperature being detected as over 85 °C above the ambient temperature of London during the summer.

In another CFD study, Kouyunbaba et al. (2013) validated a model to simulate the hourly performance of a façade integrated photovoltaic system in combination with a Trombe Wall using in situ measurements. The computational results were validated to predict temperature profiles of the system in correlation with its power output using the recorded datasets.

Jubayer et al. (2010) developed a 3D CFD model of a BIPV/T system integrated into a 30° inclined roof of a low rise building. The investigation was mainly focused on the velocity field by comparison of the forced convective heat transfer using the Nusselt (*Nu*) number normalized by Reynolds (*Re*) number and studying various roof inclinations, wind angles, upstream roughness and turbulent intensities (Karava et al., 2012). It was observed that turbulent kinetic energy (TKE) generally decreases with distance above the surface and also with the distance from the leading edge. Moreover, it was concluded that the buoyancy dominant flows, with Richardson (*Ri*) number within the range of 0.9–7, were likely to provide a 14% improvement in convective heat transfer.

As it was discussed in the mentioned studies, previous CFD researches of BIPV mostly focused on the cavity region with a fixed parallel flow, and only minimally include the impact of the entrance flow when wind is entering as a non-parallel flow. In other words, the microclimate around and within the BIPV's cavity can play a significant role in the heat removal mechanism from

Download English Version:

https://daneshyari.com/en/article/5451197

Download Persian Version:

https://daneshyari.com/article/5451197

Daneshyari.com