Applied Energy 91 (2012) 103–115

Contents lists available at SciVerse ScienceDirect

Applied Energy

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

Flow and heat transfer inside a PV/T collector for building application

Olympia Zogou, Herricos Stapountzis *

University of Thessaly, Mechanical Engineering Department, Volos, Greece

ARTICLE INFO

Article history:
Received 2 June 2011
Received in revised form 9 September 2011
Accepted 9 September 2011
Available online 7 October 2011

Keywords:
Building integrated photovoltaics
Flow field
PV cooling
Hot wire anemometry
Flow visualization
CFD

ABSTRACT

In certain building applications, the PV installation is extended to cover also south- or west-facing walls, taking care to circulate cooling air to the back of the panels. The cooling effect maintains a high conversion efficiency of the PV panels and the heated air may be exploited by the HVAC or service water heating system. Sizing and design of the double façade system is critical to its energetic performance. In this paper, the results from flow visualization and hot wire anemometry measurements performed on the basic structural module of a double-skin photovoltaic (PV/T) façade are discussed. The concept and its feasibility have been presented in a previous paper. The results of transient outdoor measurements with the testing device have been reported elsewhere. The air flow and turbulence field inside the cavity is analyzed in the present paper by means of indoor measurements with the testing device. The results are combined with CFD computations to calculate heat transfer coefficients and improve our understanding and modeling of the specific PV/T concept.

1. Introduction

During the last decade, building photovoltaic installations are extended to cover also south- or west-facing walls, taking care to circulate cooling air to the back of the panels. The cooling effect maintains a high conversion efficiency of the PV panels and the heated air may be exploited by the HVAC or service water heating system. Design optimization for this type of applications must be based on a sound understanding of the air flow and heat transfer inside the cavity behind the PV panels. A number of research works is already present in the specialized literature on this broad subject.

Sandberg and Moshfegh [1] analyzed the mass flowrate, velocity, temperature rise and location of neutral height in air gaps behind solar cells located on vertical façades. The flow was assumed to be turbulent or laminar and behave as bulk flow. For turbulent flow, the mass flow rate, velocity and volumetric flow rate follow a power-law relation with the effective heat input raised to 1/3. The temperature increase between the inlet and the outlet is proportional to the heat input raised to 2/3. Laminar flow might occur in very narrow air gaps. At the laminar flow regime, for special cases, there is a simple relation between the heat input and flow variables. For a given heat input, the maximum flow rate is obtained by increasing the height of the air gap until a balance between friction and buoyancy is obtained.

A procedure for modeling double façades made of glass layers with a ventilated mid-pane shading device—comprising a spectral optical and a computational fluid dynamic (CFD) model—is described by Manz [2]. The simulation results are compared with data derived from an experimental investigation of a single-story glass double façade with free convection, incorporated in an outdoor test facility. For a given set of layers, total solar energy transmittance may vary by a factor of five. However, use of these models even for free convection only is questionable, because of the complexity induced by gaps between cavities, airflows through shading devices, etc. The recirculations that frequently occur in real façade cavities are inherently impossible to model using a piston-flow approach.

Pappas and Zhai [3] briefly reviewed the primary parameters for a double skin façade (DSF) design. This research has developed an iterative modeling process with integrated CFD and building energy simulation program to analyze the thermal performance of double skin façade with buoyancy-driven airflow. The model was validated using measured data from Dirk Saelens taken at the Vliet test cell in Leuven, Belgium, and errors are calculated with root mean differences for air flow rate prediction of 2.7 m^3/h (or 9%), and 2.0 °C (or 15%) for temperature stratification.

Fossa et al. [4] studied the natural convection in an open channel in order to investigate the physical mechanisms which influence the thermal behavior of a double-skin photovoltaic (PV) façade. To this end, a series of vertical heaters is cooled by natural convection by air flowing between two parallel walls. Different heating configurations are analyzed, including the uniform heating mode and two different configurations of non-uniform heating. The procedure allows the wall surface temperature, local heat transfer coefficient and local and average Nu numbers to be inferred.

* Corresponding author. Tel.: +30 24210 74003; fax: +30 24210 74052.
E-mail address: erikos@uth.gr (H. Stapountzis).

0306-2619/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.apenergy.2011.09.019

Nomenclature

A	heat transfer area (m^2)
c_p	heat capacity (kJ/kg K)
d	the depth between the plates defining the flow (m)
D_h	hydraulic diameter (m)
g	gravity acceleration (m^2/s)
h	convection coefficient, air-to-channel wall (W/m^2 K)
H	height of channel (m)
k	thermal conductivity (W/m K)
$\dot{m}$	mass flow rate (kg/s)
Nu	Nusselt number of the fluid in the channel
Pr	Prandtl number for the fluid in the duct
Q	heat flow rate (W)
Ra	Rayleigh number of the fluid in the channel
Re	$Re = \frac{4\dot{m}}{\pi\mu D_h}$ Reynolds number (channel)
T	temperature (K)
u'	RMS velocity fluctuation
U	velocity (m/s)

Greek symbols

α	thermal diffusivity (m^2/s)
α_{sl}	the slope of the duct (vertical = 90°)
β	volumetric coefficient of expansion (T^{-1})
Δp	pressure drop, pressure head (Pa)
δ	declination angle ($-23.45° < \delta < 23.45°$)
λ	thermal Conductivity of glass (W/m K)
μ	viscosity (Pa s)
ν	kinematic viscosity (m^2/s)
φ	latitude
ω	hour angle

Subscripts

air	air flowing in the duct
a	ambient

Abbreviations

BIPVT	building integrated photovoltaic/thermal
CEN	Comité Européen de normalization
fps	frames per second
GDF	glass double façades
EN	European norm
NOCT	normal operating cell temperature
PV	photovoltaic
STC	standard test conditions
RMS	root mean square

Infield et al. [5] investigated different approaches to estimate thermal performance of ventilated photovoltaic (PV) façades. Heat loss and radiation gain factors have been employed to take account of the energy transfer to the façade ventilation air. Four terms describing ventilation gains and transmission losses in terms of irradiance and temperature components are defined, which characterize the performance of the façade in total. This approach has been applied to the ventilated PV façade of the public library at Mataro, Spain. The Mataro building is marginally less energy efficient than a similar construction with a conventional windowed south wall. It does, however, have other advantages in terms of the quality of the interior space, the contribution of natural light, and the aesthetically pleasing integration of PV. The PV module temperature generally does not exceed 45 °C.

Balocco [6] provide a steady state calculation model to simulate and study the energy performance of a ventilated façade, and compare different typologies of façade systems. The results show that it is possible to obtain a sensible solar cooling effect when the air cavity width of the chimney is wider than 7 cm.

Mei et al. [7] studied the thermal performance of a specific type of ventilated PV façade, consisting of a PV panel, an air gap and an inner double glazing. A dynamic thermal model based on TRNSYS was developed for a building with an integrated ventilated PV façade/solar air collector system. The building model developed has been validated against experimental data from a 6.5 m high PV façade on the Mataro Library. Based on the building model, the heating and cooling loads for the building, in different locations, have been estimated. From both measurement and simulation, it can be seen that the PV façade outlet air temperature reaches around 50 °C in summer and 40 °C in winter. Twelve percentage of heating energy can be saved using the pre-heated ventilation of the air for the building location in Barcelona during winter.

Manz et al. [8] developed a procedure for modeling glass double façades comprising a spectral optical and a computational fluid dynamic model. Simulation results are compared with data derived from an experimental investigation of two mechanically ventilated glass double façades (GDFs) built in an outdoor test facility. It is shown that simple models assuming piston-flows can lead to inaccurate results. Hence, a combination of experiment and simulation is considered the most reliable approach for analyzing glazed double facades.

Liao et al. [9] studied the heat transfer and fluid flow in a building – integrated photovoltaic – thermal system intended for single story applications with a two-dimensional CFD model. The k–ε turbulence model is employed for the flow and convective heat transfer in the cavity, including buoyancy effect. Long-wave radiation between boundary surfaces is also modeled. Although the main case of interest is forced convection with fan-induced airflow, at low pressure differentials, there is a buoyancy-induced velocity peak near the PV surface. The heat transfer coefficients were predicted for the two cavity surfaces, and the convective coefficient was determined through a combination of CFD simulations and experimental measurements that provided boundary conditions; the convective coefficients were generally higher than expected by other correlations. Measurements of the velocity profiles in the BIPV system cavity using a particle image velocimetry system were in good agreement with CFD model predictions. Correlations have been developed for the convective heat transfer coefficients.

Infield et al. [10] proved that four parameters can be used to adequately characterize the thermal performance of partially transparent ventilated PV façades.

Yun et al. [11] investigated the complex interrelationship between a ventilated PV façade and the overall energy performance of a building. They analyzed a theoretical ventilated photovoltaic (PV) façade, which functions as a pre-heating device in winter and a natural ventilation system in summer and reduces PV module temperatures.

Solanki et al. [12] report on the design, fabrication and performance assessment of a PV/T solar air heater. A simplified 1-D thermal model of the device was developed, and a test method proposed as a standard indoor test procedure for thermal and electrical testing of series connected PV/T collectors. Typical electrical efficiencies of the order of 10% and thermal efficiency of the order of 35% are reported.

Sarhaddi et al. [13] developed an analytical model to investigate the thermal and electrical performance of a solar photovoltaic thermal (PV/T) air collector. Some corrections are done on heat loss coefficients in order to improve the thermal model. The results of