

Design Optimisation for Cooling Building Integrated Photovoltaics using CFD

Abdel Rahman Elbakheit*, Dr. Guohui Gan**, Prof. Saffa B. Riffat***
School of the Built Environment, University of Nottingham, University Park,
Nottingham, NG7 2RD

*PhD student, Tel: +447780747214, Fax+441159513159, E-mail Laxaa1@nottingham.ac.uk

**Lecturer, Tel: +441159514876, Fax +441159513159, E-mail guohui.Gan@nottingham.ac.uk

***Director of IBT, Tel: +441159513158, Fax+441159513159, E-mail Saffa.Riffat@nottingham.ac.uk

Abstract: This paper presents the results from a simulation of the buoyancy effect occurring behind photovoltaic modules. Computational fluid dynamics (CFD) has been applied as a tool to predict airflow and temperatures in a cavity behind a heated panel. The results have been compared with experimental results published in the literature. Further discussion of using CFD in evaluating the performance of passive design of building envelopes integrated with PV is made. A number of design options were generated and evaluated..

NOMENCLATURE

C_p Specific heat, J/kg · K
 ΔT Temperature difference, K

m Mass flow rate, kg/s
 T Temperature, K

OVERVIEW

Statistics show that the annual consumption of electricity in estates in the UK is slightly higher than one and a half ^[1] in size compared to fuel consumption. It also shows that there is a trend to switch over from fuel consumption to electricity with a rapid growth of up to 11% from the period of 2002–2004. It is estimated that the building sector accounts to more than 70% of the annual consumption of electricity, whether in the construction, operation or activities inside them. This highlights the importance of this sector when considering measures to reduce energy consumption and exploit alternative solutions such as conservation, synthetic fuels, wind and solar power as part of a strategy to satisfy demand and reduce overall negative impact of energy use. One of the most promising renewable energy technologies is photovoltaic or PV.

1. INTRODUCTION

Designing buildings with PV presents a challenge to the designer to meet the demands for the physical form, environmental system, orientation, footprint and aesthetics of buildings. Among these challenges is the problem of overheating of PV panels when subjected to prolonged periods of solar radiation that reduces efficiency. Many approaches have been adopted to utilize this heat, such as hybrid systems, (i.e. combined generation of heat and electricity).

From the architectural point of view, approaches that involve passive cooling through buoyancy would

be valuable. The physical phenomena underlining the buoyancy effect rely on the density of air being reduced by heating, causing the warm (i.e. low density) air to rise and be replaced by cooler dense air from below. Wind induced pressure difference gives rise to a second force acting on the air near the PV panel. Forces that resist the flow are friction and end losses at both ends of the gap between the PV panel and wall or panel behind. The PV panel is heated by radiation and heat is transferred to the air gap by convection and radiation.

By thoroughly insulating the wall or panel opposite to the PV panel and the sides of the air gap, all the heat gain is delivered to the air in the gap by convection. The back panel will be heated by radiation, and heat will also be transferred to the air from it by convection.

The methodology adopted in this research involves validation of the results attainable from CFD by performing a simulation to a reference experiment for PV cooling with buoyancy. CFD is then used as a design tool to model different design configurations aiming to enhance the buoyancy behind the PV panel. The results obtained were compared in order to establish the best possible cooling options for a PV installation in a prototype house design.

2. PREVIOUS WORK

Experiments to investigate the buoyancy effect in an air gap similar to that found behind a PV panel have been undertaken by Moshfegh and Sandberg ^[2] as shown in figures 1 and 2 above. A theoretical investigation of the buoyancy flow in an air gap of

height $H=6.5\text{m}$ and cross section $w \times d = 1.64 \times 0.23\text{m}$ was also carried out. The aspect ratio (i.e. $H/d = 28$). The air gap was heated along its whole height with an electric heat source to mimic the heat-input from PV panels.

The back wall was thoroughly insulated so that the heat input was transferred to the air inside the channel. Tests were carried out for a range of heat fluxes: 20, 50, 100, 200 and 300 W/m^2 . The flow rate was measured using the constant tracer gas technique. Full details of the experiments and results obtained are given in references [3 – 4].

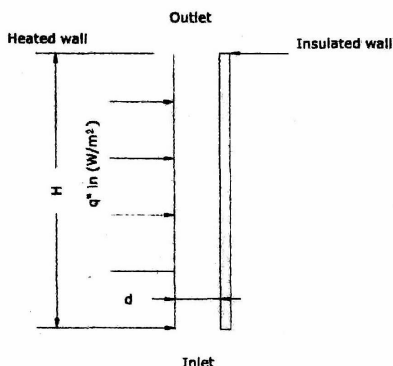


Figure 1 Geometry of parallel inlet and outlet. (Top)

3. CFD SIMULATION

In this study, FLUENT [5] is used to conduct a CFD simulation for the abovementioned experiments to examine its potential for predicting the behaviour of the system. The standard $k-\epsilon$ model has been used to represent turbulence. The results predicted by CFD were found to be in agreement with the experimental results as in figures 3 and 4 below.

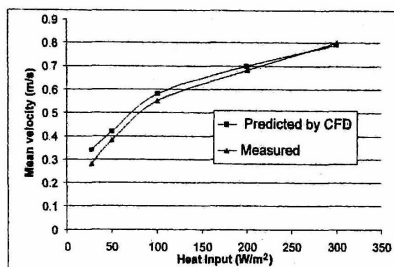


Figure 3 Comparison of experimental, and modelled data for the mean air velocity as a function of heat input for Parallel panels.

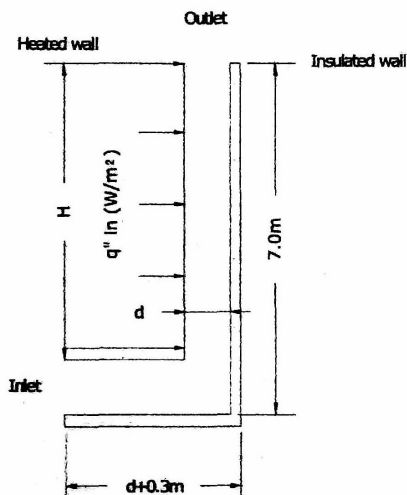


Figure 2 Geometry for L-shape experiment.
(Bottom)

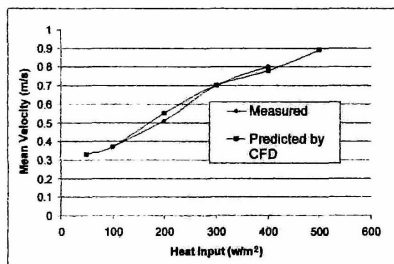


Figure 4 Comparison of experimental, and modelled data for the mean air velocity as a function of heat input for reversed L-shaped air Gap.

4. OPTIMIZATION OF DESIGN AND PERFORMANCE OF PV INTEGRATED INTO BUILDINGS

Considerable research has been directed towards idealized integration of PV installations in building envelopes such as Bahaj^[6] work. Most of these attempts were aimed towards acceptability, financial suitability, build-up, alylight versus electrical output and practical issues like maintenance. However for an optimum design and performance of PV integration, a better understanding has to be made to the technical and perational characteristics of PV. Special reference here can be made to the overheating problem. Such problem can be overcome by careful design measures and, by applying the theory of allowing passive cooling of PV through the buoyancy effect. The following study is made to provide informed design solutions for building integrated PVs for an exemplary design of a house.

4.1 HOUSE PROPOSAL

A feasibility study is conducted for the Zero Carbon village or the Eco Village for Nottingham University (latitude 52°N), which will provide additional residential facilities for students and staff of the university. Residence facilities are categorised in a number of different groups of buildings to suit different end users. One of the proposed categories is the detached house unit that provides accommodation for teaching staff.

The detached houses (figures 5 and 6) are basically four bedroom houses, with kitchen, two bathrooms and a lounge. In order to study the effect of overheating of PV while operation a number of design options were made aiming to cool PV panels with the buoyancy effect.

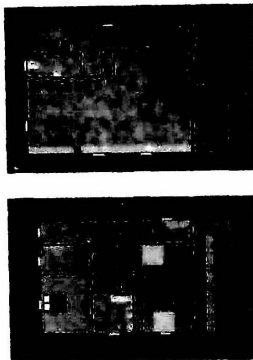


Figure 5 Plans of the Ground and first floors, Top to Bottom.

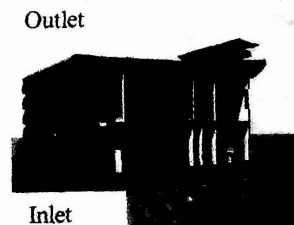
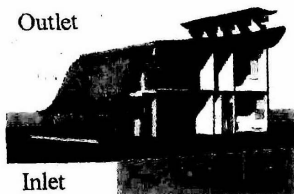
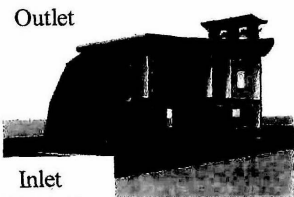
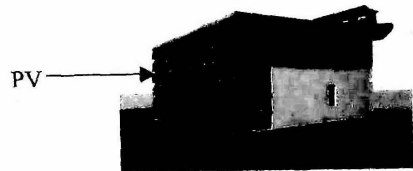
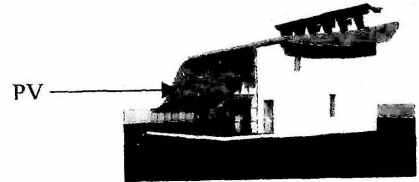
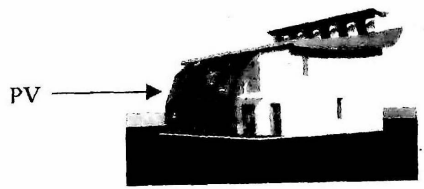


Figure 6, General outside view of design options 1,2&3 from Top to bottom and Sectional perspective view through the PV of the same designs.

PV is integrated into the sunspace at the far left side of each design option. Each case was simulated using CFD to predict how well it would respond to the problem of overheating of PV. The heat flux for PV is assumed as 360W/m^2 , which is the condition prevailing in an inclined wall mounted PV module based on clear sky radiation data for London at noon on June 21^[7].

4.2 PV architectural integration

From the design of the proposed house, the south facing facade was the major position for integrating solar-based renewable energy technologies. As a rule of thumb for optimum panel orientation, PV is positioned with a slope equals to the latitude minus 20 degrees^[7] for the UK. Therefore the proposed PV arrays are tilted at about 32° . An insulated, high emissivity back panel is fixed behind each PV installation to trap the heat in the gap behind the PV as well as to enable the air to carry the heat. PV are installed in a southerly sunspace, dimensions are 9m long, 3.2m wide and 5.6m high as part of an inclined curtain wall design, supported with aluminium mullions and transoms.

5. COMPARISON OF PERFORMANCE

In order to select a design that would deliver the optimum performance a comparison between three design options is made. Figure 7 depicts the predicted PV temperatures for simple passive architectural design of forms as described earlier.

The anticipated cooling capacity of each design layout can be estimated using the following equation.

$$\text{Cooling capacity} = m \cdot C_p \cdot \Delta T$$

That is mass flow rate X specific heat X temperature difference.

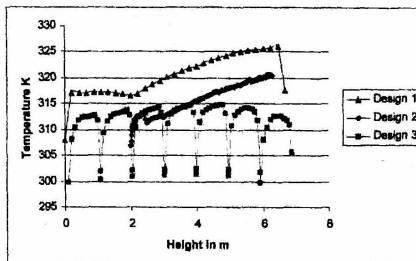


Figure 7 Comparison of PV

performance for the three design options, at operating temperature of 293K.

Mean resulting PV temperatures Are 321K, 316K and 311K for design options 1,2 and 3 respectively.

From CFD simulation the predicted mass flow rates and temperature differences between inlet and outlet air in the sunspace as a result of PV integration are as follows:

Table 1 Comparison of Case Results, at operating temperature of 293K and duct system.

Parameters	Design 1	Design 2	Design 3
Cooling(W)	840	808.5	460
Inlet/Outlet Size(m)	0.30	0.90	0.30
Duct width (m)	0.30	0.90	0.90
Area(m ²)	8	8	8
Mass flow rate(kg/s)	0.21	0.385	0.20
Temperature Difference (K)	4	2	2.3
Mean PV Temperature (K)	321	316	311

The different examined design options have the same area of PV, same overall height and provide the same architectural role. However, from figure 5 and Table 1 design option 3 has the coolest predicted mean PV temperatures of 311K followed by design 2 with 316K while design 1 has the highest mean PV

temperature of 321K. The predicted cooling capacity of each design option is 840W, 808.5W and 460W for the first, second and third designs respectively. The PV panel height in Designs 1 and 3 starts from the ground level, yet design 1 is higher in temperatures than design 3. However, because the first option has a continuous unbroken length it has a build-up effect on PV temperatures. Therefore the longer the length of the PV panels the more likely there is to be an accumulation of heat in the upper parts. One way of solving this problem is by introducing a number or series of inlets across the height of the panel, similar to the idea used in the third option. Design option 2 has a PV panel that starts from 2m height on top of a window level. This option also experiences the same built up effect of temperatures due to the long stretch of the panel, however because of the higher mass flow rate, which is the result of the shape of the panel and the position of the duct behind it we get lower temperatures than the first option. This emphasises the influence of the design shape in reducing PV temperatures.

Since PV performance (i.e. efficiency) depends on the temperature of the panel, we can conclude that design 3 is the optimum option because it produced the lowest temperature of 311K(38 °C) followed by Design 2 and design 1.

Table 2 Comparison of Case Results, at operating temperature of 293K without duct system.

Parameters	Design 1	Design 2
Cooling(W)	696.3	740
Inlet/Outlet Size,(m)	0.30	0.90
Duct width,(m)	0.30	0.90
Area(m ²)	8	8
Mass flowrate (kg/s)	0.21	0.37
Temperature Difference(K)	3.3	2
Mean PV Temperature(K)	318	316

Table 2 above describes the prediction of PV performance without using an insulated duct behind the PV panel. While similar results to those of the ducted system are observed in design 2. Design 1 results are 3-degrees lower in the un-ducted than with ducted system. The addition of a duct or surface behind the PV panel would increase the resistance to the flow. However, the results of design 2 show that the position of the duct can be optimised to the level where the resistance to the flow is minimal. Further optimisation for the duct position would be needed for design 1.

6. CONCLUSIONS

This investigation has shown that CFD FLUENT provides agreement with experimental results on the effect of buoyancy behind PV panels. FLUENT has been used as a design tool for optimisation of passively cooled PV panels through various parametric studies (i.e. Design shape, use of duct systems etc). PV cooling with buoyancy is an economic way to ensure efficient production of electricity. This process can be further enhanced through careful design of building envelopes in such a way that allow for this process to take place. The use of ducting or insulated surface behind the PV panels can assist the building requirement for thermal comfort, in terms of the desired U-values. Care must be taken to optimise the position of this panel in order to minimize the resistance to the flow. Otherwise the resistance to the flow will cause a build up of heat and reduce the cooling effect.

As the PV panel alone can't provide the full requirement needed by building envelope (i.e. required thermal properties etc.). The use of an insulated ducting system can provide the necessary backup here. The ducting system can further contribute to the ventilation requirement.

Acknowledgements:

The first author would like to extend his thanks to Dr. Robin Wilson for his valuable advice and comments through various stages of preparing this paper.

REFERENCES

- [1] Building Estates Management unit, published August 2004, updated Nov. 2004.
www.homeoffice.gov.uk/doc/energytargets.html.
- [2] Flow and heat transfer in the air gap behind photovoltaic panels, by Moshfegh B. and Sandberg M. Renewable and Sustainable Energy Reviews, 2(1998), 287-301.
- [3] Moshfegh B. and Sandberg M. The investigation of fluid flow and heat transfer in a vertical channel heated from one side by PV element. Part I - Numerical Study, Proceedings of the fourth Renewable Energy Congress, Denver, USA 1996. Volume 8, Issues 1-5, 8 May 1996, P (248-253).
- [4] Sandberg M. Moshfegh B. The investigation of fluid flow and heat transfer in a vertical channel heated from one side by PV element. Part II - Experimental study, Proceedings of the fourth Renewable Energy Congress, Denver, USA 1996.
- [5] FLUENT, user's Guide, Fluent Inc. USA; 1995
- [6] Photovoltaic roofing: Issues of design and integration into buildings, Abubakr S. Bahaj. Sustainable energy research group, civil and environmental engineering Department, University of Southampton, Hampshire, SO17 1BJ, UK, 2003.
- [7] Photovoltaic and architecture, Edited by Randall Thomas and Max Fordham & partners, Spon Press, published 2001, P-11.