Simulation of the Convective Heat Transfer and Working Temperature Field of a Photovoltaic Module using COMSOL Multiphysics

E. Ruiz-Reina[*1] and M. Sidrach-de-Cardona[1]
[1]Departamento de Física Aplicada II, Universidad de Málaga.
*Corresponding author: Escuela de Ingenierías, Campus de Teatinos, s/n, 29071, Málaga, Spain, eruizr@uma.es

Abstract: The aim of this work is the numerical study, by finite element analysis using COMSOL Multiphysics, of the convective heat transfer and working temperature field of a photovoltaic module under different wind conditions.

Keywords: Photovoltaic Modules, Temperature, Heat Flux, Convection Cooling.

1. Introduction

It is well-known that a great portion of the solar radiation absorbed by a photovoltaic module (typically 85% of the incident radiation) is not converted into electrical energy, but it is wasted by the increase of the module's temperature, reducing its efficiency by heat transfer with the surrounding medium[1,2]. In this contribution we perform a numerical study, using COMSOL Multiphysics, of the convective heat transfer and working temperature field of a photovoltaic module under different wind conditions.

The working temperature of photovoltaic modules depends on different environmental factors as the ambient temperature, the solar irradiation, the relative humidity, the direction and speed of the wind; and physical factors as the construction materials and particular installation of the module[1].

For ease, we have chosen a very simple module with a unique Si cell. A schematic of the simulation system is shown if Figure 1. We suppose that the module is installed forming 30º with the horizontal axis. All the thermal and fluid parameters of the materials involved have been found in the literature[3].

2. Use of COMSOL Multiphysics

2.1 Geometry

The geometry of the system is divided into five subdomains:

Solid Subdomains:

1. The glass of the cover. Width: 250 mm. Thickness: 3 mm.

2. The Silicon cell. Width: 125 mm. Thickness: 0.4 mm.

3. The EVA (ethylene vinyl acetate) film. Width: 250 mm. Thickness: 0.8 mm.

4. The Tedlar back film. Width: 250 mm. Thickness: 0.05 mm. White reflective color.

Gaseous Subdomain:

5. The air that surrounds the module. Width: 37 cm. Height: 14 cm.

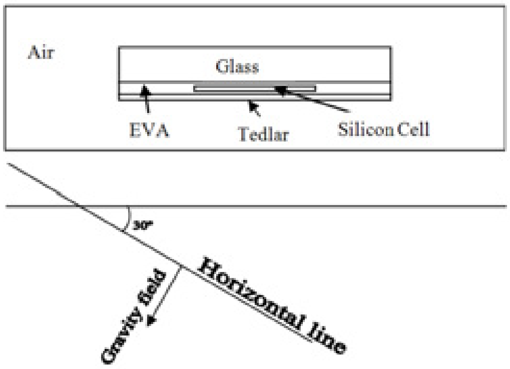

Figure 1. A schematic of the problem. For clarity, the distances are not in real scale.

2.2 Thermal equations

For this equations we use the General Heat Transfer application mode. The equations to be solved in every subdomain are:

Gaseous Subdomain: Heat conduction and convection,

$$\vec{\nabla}(-k\vec{\nabla}T) = Q - \rho c_p \vec{u} \cdot \vec{\nabla}T \qquad (1)$$

where T is the absolute temperature, k is the air thermal conductivity, Q is the heat generation term ($Q = 0$ in our case), $\vec{u}$ is the velocity field, ρ is the density and c_p is the specific heat capacity at constant pressure. The velocity field is coupled with the Navier-Stokes equations (see below) that are treated in the Weakly Compressible Navier-Stokes application mode.

The density and thermal conductivity of the air are related with the temperature by the expressions

$$\rho = 1.013 \cdot 10^5 \frac{28.8 \cdot 10^{-3}}{8.314T} \quad (2)$$

$$k = 10^{-3.723 + 0.865 \log T} \quad (3)$$

where the temperature is introduced in Kelvin, the density is given in kg/m^3 and the thermal conductivity in W/(m·K). Equation (2) comes from the equation of state of ideal gases applied to the case of air. The specific heat capacity at constant pressure of the air is taken as constant and equal to $1.1 \cdot 10^3$ J/(kg·K).

Solid Subdomains: Only heat conduction,

$$\vec{\nabla}(-k\vec{\nabla}T) = Q \quad (4)$$

where Q is zero for the Tedlar, glass and EVA subdomain, and has a value of 1.005 MW/m^3 for the silicon cell subdomain, which corresponds to an incident irradiation of 1000 W/m^2 homogeneously distributed minus a 15 % efficiency of electrical conversion.

The thermal conductivities k of the glass, EVA, silicon and Tedlar are 1.7 W/(m·K), 0.23498 W/(m·K), 148 W/(m·K) and 0.1583 W/(m·K), respectively. The specific heat capacity at constant pressure c_p of the glass, EVA, silicon and Tedlar are 780.33 J/(kg·K), 3135 J/(kg·K), 710.08 J/(kg·K) and 1090 J/(kg·K), respectively.

2.3 Fluid equations

The fluid equations are only applicable in the gaseous subdomain (air). They are the corresponding to the Weakly Compressible Navier-Stokes application mode:

$$\rho\vec{u} \cdot \vec{\nabla}\vec{u} = \vec{\nabla}\left[-p\mathbf{I} + \eta\left(\vec{\nabla}\vec{u} + (\vec{\nabla}\vec{u})^{\mathrm{T}}\right) - \frac{2}{3}\eta(\vec{\nabla} \cdot \vec{u})\mathbf{I}\right] + \vec{F} \quad (5)$$

$$\vec{\nabla} \cdot (\rho\vec{u}) = 0$$

where the suprescript T denotes the transpose matrix, p is the pressure, η is the dynamic viscosity of the air, $\mathbf{I}$ is the identity matrix and $\vec{F}$ is the body force in any point of the fluid. The air viscosity is related with the temperature by the expression

$$\eta = 6 \cdot 10^{-6} + 4 \cdot 10^{-8} T \quad (6)$$

where the temperature is introduced in Kelvin and the dynamic viscosity is given in Pa·s.

The body force $\vec{F}$ acting on the fluid is the buoyancy force due to the dependence of the density ρ of the air with the temperature by equation (2). In the case of the geometry depicted in figure 1, we have

$$\vec{F} = \frac{1}{2} g(\rho_0 - \rho)(\hat{i} + \sqrt{3}\hat{j}) \quad (7)$$

where g = 9.81 m/s^2 is the value of the gravity field and ρ_0 = 1.17 kg/m^3 is the density of the air at ambient temperature 300 K.

2.4 Boundary conditions

In figure 2 we show schematically the geometry with a specification of the relevant boundaries.

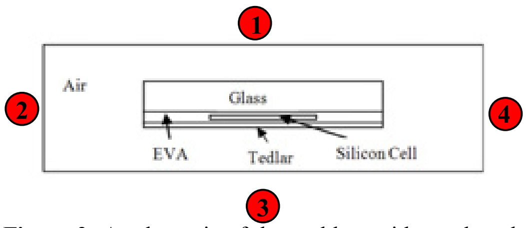

Figure 2. A schematic of the problem with numbered boundaries. The distances are not in real scale.

The boundary conditions are as follows:

Boundaries 1 and 2. There is a wind in the horizontal line that incides over the glass surface of the module. The boundary type in the Weakly Compressible Navier-Stokes application mode is "Inlet" with boundary condition "Velocity". The air velocity in these boundaries is

$$u_0 = v_{ini}\cos\left(\frac{\pi}{3}\right) = \frac{v_{ini}}{2}$$
$$v_0 = -v_{ini}\sin\left(\frac{\pi}{3}\right) = -\frac{\sqrt{3}v_{ini}}{2} \quad (8)$$

where v_{ini} is the inlet wind speed. The boundary condition in the General Heat Transfer application mode is "Temperature" with a value of 300 K, the ambient temperature.

Boundaries 3 and 4. The boundary type in the Weakly Compressible Navier-Stokes application mode is "Stress" with boundary condition "Normal Stress". The boundary condition in the General Heat Transfer application mode is "Convective Flux". With these conditions we try to model an unbounded domain.

Inner boundaries. The boundary type in the Weakly Compressible Navier-Stokes application mode is "Wall" with boundary condition "No Slip". The boundary condition in the General Heat Transfer application mode is "Continuity".

3. Results

The problem seems to be highly non-linear. A direct resolution of the problem in a single step was found to be impossible. Searching for an alternative possibility, we performed a parametric resolution starting from a well-behaved situation. We select the air viscosity as a varying parameter and the first iteration was made with a viscosity value one hundred times greater than the real one. The reason behind this election is that, for a high viscosity fluid, the flow is laminar and the problem is much easier to solve. The fluid viscosity value is then diminished in ten steps until it reaches the real air viscosity value at the final iteration.

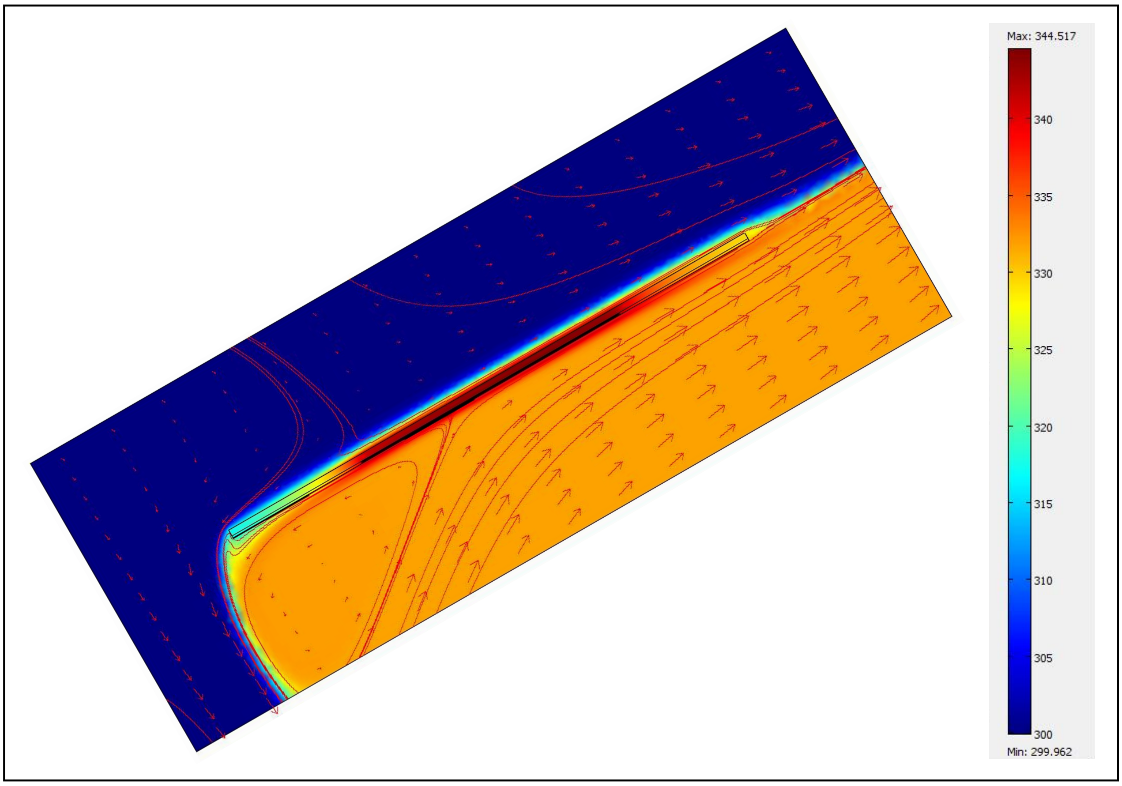

Figure 3. Temperature and velocity fields under a wind speed of 0.01 m/s. Color surface plot: temperature. Arrows and streamlines: velocity field.

In figure 3 we show a plot of the calculation of the temperature and velocity fields under a wind speed of 0.01 m/s.

The wind comes from the upper and left boundaries. The color surface plot represents the temperature and the arrows and streamlines correspond to the velocity field.

From this plot, we can observe some features:

- The temperature of the module at its centre is about 40 ºC higher than the ambient temperature.
- The temperature difference from the upper and lower borders of the module is about 10 ºC.
- There is an air vortex, with roughly 8 cm diameter, behind the module.

The results about the temperature of the module agree well with the values founded in experiments[2,3].

4. Conclusions

In this contribution, we have shown the utilization of COMSOL Multiphysics in the thermal simulation of a working photovoltaic module. The convention cooling by air is a difficult problem to solve, but using the parametric solver we can obtain reliable solutions.

From the numerical results, we can extract a lot of information about the temperature fields and heat transfer with the ambient medium. In particular, this procedure could be applied in the determination of the convective heat coefficients of photovoltaic modules made with different technologies, under several installation conditions and wind speeds. This will be a future work.

5. References

1. M. Mattei, G. Norton, C. Cristofari, M. Muselli and P. Poggi, *Renewable Energy*, **31**, 553-567 (2006).
2. K. Emery, Measurement and Characterization of Solar Cells and Modules, in *Handbook of Photovoltaic Science and Technology*, A. Luque and S. Hegedus, eds. (2003).
3. M. C. Alonso García and J. L. Balenzategui, *Renewable Energy*, **29**, 1997-2010 (2004).

6. Acknowledgements

Financial support for this work by Junta de Andalucía, Spain, Project P07-RNM-02504, co-financed with FEDER funds by the European Union, and photovoltaic modules manufacturer Isofotón, contract number 8.06/5.57.2619-1, are gratefully acknowledged.