Simulation of Fluid Flow in an Artificial Soiling Device

E. Klimm*, S. Kochersperger, K.-A. Weiss
Fraunhofer Institute for Solar Energy Systems, Heidenhofstrasse 2, 79110 Freiburg
*Corresponding author: elisabeth.klimm@ise.fraunhofer.de

1. Introduction

Solar systems are used around the globe, especially in sunny and dry regions, to convert radiation from the sun into electricity or heat. Soiling, as dust deposition onto surfaces is commonly called, hinders this conversion of the solar energy by blocking the incidenting sunlight. The dust is carried by the wind, settles out of the wind onto surfaces, and causes soiling. This occurs more often in environments with loose soil coverings such as deserts. This is problematic because it is desirable to locate solar arrays in desert environments due to the huge amount of sunlight and space available. To reduce the costs of produced electricity, it is not desirable to have efficiency deteriorate throughout the operation of the solar array due to soiling. It is important to understand the effects of sand and dust on solar arrays because of the large solar potential that exists in desert regions around the world. It is also important to understand the technical and economic risk, which dust poses to the solar performance and yield.

In this study the turbulent fluid flow mechanics in a closed artificial soiling system are modelled with COMSOL Multiphysics®. The closed artificial soiling device is designed to simulate homogeneous soiling of outdoor environments in a reproducible manner for better comparison of particle surface interactions and investigation of the physical soiling fundamentals via experimentation.

1.1 Motivation

A simplified approach to simulate the soiling process can help to understand the interactions of environment, particle and surface better. So far, to the knowledge of the authors, only a few simulation studies have been conducted that model the soiling process. None of these studies focus on analyzing the fundamental physical forces and interactions at play in the soiling process. This study aims to reduce this gap in the soiling research field by focusing on the basics of soiling in a well-controlled closed system by elaborating on the turbulent fluid flow and particle transport forces that are fundamental to understanding soiling. This model and the associated physics are the first approach for real-world modelling of soiling in outdoor environments. One major advantage in using a small-scale, well controlled set-up, such as the dusting device, is that the computer simulation results can be validated easily through experimentation.

1.2 Investigative Approach

Finite Element Method (FEM) simulations of an in-house developed artificial dusting device are created using COMSOL Multiphysics® in 2D and 3D. The results of the simulation can be verified with fluid flow experiments conducted with the artificial dusting device.

2. Use of COMSOL Multiphysics® Software

Fluid flow experiments conducted with the aritificial dusting device were compared with the simualtion created in COMSOL Multiphysics®. The system was carefully analyzed to determine which physics models provided by COMSOL were appropriate to use in the model. To accomplish this: the Knudsen number, Reynolds number, and Mach numbers were used to characterize the fluid behavior in the device. After the initial characterization of the fluid, it was determined that the $k-\varepsilon$ turbulence model provided by COMSOL Multiphysics® is the model that is most suitable to simulate the fluid flow in the artificial dusting device.

This turbulence model solves three additional equations along with the standard Navier-Stokes equation to be able to account for turbulence accurately.

$$\boldsymbol{\mu_T = \rho C_\mu \frac{k^2}{\varepsilon}} \quad (1)$$

$$\boldsymbol{\rho \frac{\partial k}{\partial t} + \rho u \cdot \nabla k = \nabla \cdot \left(\left(\mu + \frac{\mu_T}{\sigma_k} \nabla k \right) + P_k - \rho \varepsilon \right)} \quad (2)$$

$$\boldsymbol{\rho\frac{\partial\varepsilon}{\partial t}+\rho u\cdot\nabla\varepsilon=\nabla\cdot\left(\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\nabla\varepsilon\right)+C_{\varepsilon 1}\frac{\varepsilon}{k}P_k-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}\right)} \tag{3}$$

This model uses a formulation of the turbulent viscosity, μ_T, that introduces two new parameters: the turbulent kinetic energy (k), and the turbulent dissipation (ε). Two additional transport equations to define k and epsilon are used to supplement what the Reynolds Averaged Navier-Stokes equation cannot express.

Two simulations were created to simplify the modelling process and to aid in cutting down on simulation time. The first simulation was a 2D simulation, even though a 2D representation of the experimental set-up does not accurately represent the system, it requires less time to run and aids in making sure that the solver can converge with the given inputs before moving onto the time-intensive, complex 3D model. This means that more time was spent on computation and analysis of results for the 3D model, shown in Figure 1. The quality of the mesh, the wall resolution, stabilization methods, turbulence models and characterization of the fluid flow were all taken into account while developing the simulations.

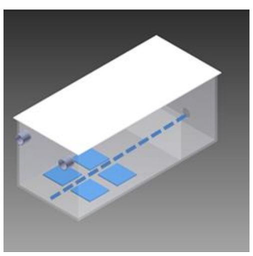

Figure 1. CAD rendering of the experimental set-up showing the plane of symmetry in the box

The creation of the best mesh for the simulation was done using an iterative process to determine which combination of mesh types and sizes gives the best results. The mesh was created manually with special focus on meshing areas where the most turbulence should occur. For this system, the most turbulence should occur in the front of the device before the small wall that divides the main chamber. The mesh element quality statistics were also used to help determine which mesh size should be used.

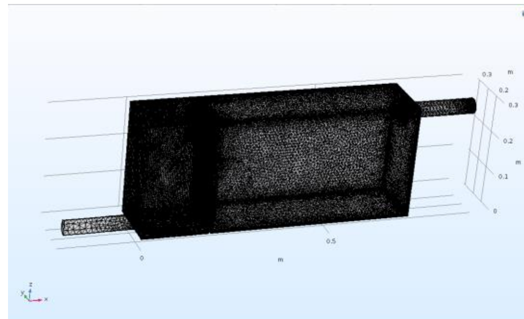

Figure 2. 3D Meshing of the dusting device after being cut along the plane of symmetry in the device

Another helpful tool to determine the quality of the mesh is the wall resolution. After running the full simulation, COMSOL automatically generates a plot that shows the mesh resolution. The mesh should ideally have the lowest number for the mesh resolution, which indicates a better mesh. Figure 3 shows the wall resolution for two different mesh sizes tested for the 3D simulation.

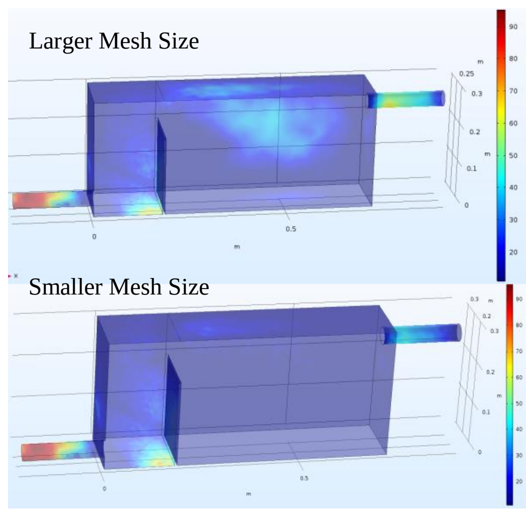

Figure 3. Wall Resolution Plots for the 3D model

For the 3D simulation this wall resolution plot determined which mesh size was better, the smaller mesh was better resolved in this case enough to merit the longer computation time that a smaller mesh size would require.

3. Results

The results show the fluid flow streams through the dusting device at three different inlet velocities. The inlet velocities chosen for modelling represent commonly occurring wind speeds in the desert environment of Fraunhofer ISE's outdoor PV test field in the Negev Desert in Israel, which is an example of an environment where soiling occurs. The chosen inlet velocities were: 1, 5, and 10 m/s; where 1 m/s represent the average and 10 m/s the maximum wind speed that occurs in the Negev Desert environment. The results for an inlet velocity of 5 m/s is verified experimentally in the lab; when using an stable inlet air pressure the velocity of 5 m/s is measured at the outlet. Figure 4 shows the result obtained by the model with inlet $\upsilon = 5$ m/s.

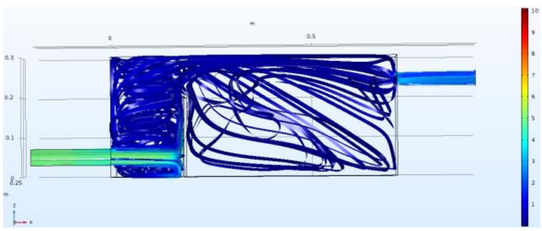

Figure 4. Fluid flow with an inlet velocity of 5 m/s.

Because of the presence of the wall inside of the box, it is expected that the turbulence would be highest between the inlet and the wall at the front of the box. This was seen experimentally and in the simulation. Experiments were performed with standardized dust (Arizona test dust SAE J726) so that the dust clouds in the box were visible. Videos of the dust distribution in the chamber were taken to compare with to the simulation results. The results showed the expected fluid flow with the red dust being uplifted in turbulent eddies in the pre-chamber and then traveling along the top of the box towards the outlet, as seen in Figure 4. Comparing the videos and the simulation results allowed for a confirmation that the simulation was accurate.

4. Conclusion

The results of the simulation matched closely to the results of the experiments conducted in the lab, which validates the results obtained from the simulation. This proves that through the creation of the numerical simulation, the basic physics of the fluid flow through the experimental set-up is understood well enough to follow up with the simulation of simplified dust particle movement in the soiling device.

A first step towards a deeper understanding into the physics of soiling was achieved and will be transferred to a macro-scale outdoor test field.