DOI: 10.2507/34th.daaam.proceedings.xxx

AERODYNAMIC COLLECTION EFFICIENCY: NUMERICAL SIMULATION AND ANALYSIS.

Mohammad Yousef Hdaib, Anas F A Elbarghthi, Musaddaq Azeem, Mohamed Abdelkader, Jiri Ragulik & Vaclav Dvorak

This Publication has to be referred as: M. Hdaib; A. Elbarghthi; M. Azeem; M. Abdelkader; J. Ragulik & V. Dvorak (2023). Aerodynamic collection efficiency: numerical simulation and analysis, pp.xxxx-xxxx, M. Hdaib (Ed.), Published by DAAAM International, ISBN 978-3-902734-xx-x, ISSN 1726-9679, Vienna, Austria
DOI: 10.2507/34th.daaam.proceedings.xxx

Abstract

The shortage of clean water resources is a common problem throughout the globe, which leads to an increasing interest in fog collection technologies. Therefore, finding new methods to optimize collection efficiency is essential. In this study, the aerodynamic efficiency of a fog collector was investigated numerically. A 3D Computational Fluid Dynamic (CFD) model was built based on the discrete phase modeling and Darcy-Forchheimer relation for Porous media. The Darcy-Forchheimer relation was fitted experimentally to find the porous media parameters, and the model was validated using results from the literature. Moreover, the velocity and pressure distribution around the fog collector were simulated and compared. The results revealed good agreement with experimental data with a high-pressure drop of 150 % and a velocity drop of 100 %.

Keywords: Fog collector; aerodynamic efficiency; CFD; Porous media.

1. Introduction

A fog collector is a device designed to extract clean water from fog droplets. It comprises three main components: mesh, gutter, and storage. The process of water harvesting occurs when droplets intersect the mesh and are directed to the storage via the gutter [1]. The performance of fog collection is assessed using the total collection efficiency, calculated as the product of three different efficiencies: aerodynamic collection efficiency, deposition efficiency, and drainage efficiency. This study focuses on the collector's resistance to airflow, represented by the aerodynamic collection efficiency (η_{ACE}). The aerodynamic collection efficiency indicates the proportion of water droplets in the upstream flow that could potentially impact the mesh fibers [2].

Several studies have developed theoretical models for aerodynamic collection efficiency [2]–[4]. In numerical simulations, porous media models have been employed to replace the mesh fibers [5]. This approach significantly reduces computation time by avoiding the structural complexities associated with mesh geometry. Carvajal et al. [5] conducted a large-scale simulation of a fog collector using the Darcy-Forchheimer porous medium model. They fitted experimental data from Bresci et al. [6] to the Darcy-Forchheimer model and applied discrete phase modeling to simulate water droplets in the flowing fog. Their results demonstrated good agreement with those obtained by Bresci et al. [6].

Recently, several studies have aimed to enhance the aerodynamic collection efficiency of fog collectors. For example, Azeem et al. [7] introduced a harp fog collector design with parallel filaments in the mesh, reducing solidity and minimizing pressure drop. Damak et al. [8] proposed a method involving electrical forces that could surpass aerodynamic drag forces by utilizing different electrical charges for the droplets and the mesh. Both of these methods were combined, as demonstrated by Sharifvaghefi et al. [9], resulting in a collection efficiency of 82%.

CFD analysis has been utilized in various engineering applications to enhance the aerodynamic behavior of vehicles. [10], [11]. In this study a 3D CFD wind tunnel was used to simulate the aerodynamic collection efficiency of a fog collector. Firstly, the Darcy-Frochheimer model was fitted based on the experiment done by Bresci et al. [6]. Then the acquired Parameters: permeability (α), and the inertial resistance factor (C_2) were used with the discrete phase modeling on Ansys fluent to simulate the velocity and pressure fields around the fog collector. Finally, the results were compared with experimental data from the literature.

2. Theory

The physical model was created based on a numerical wind tunnel approach proposed by Carvajal et al. [5]. The realizable $k-\varepsilon$ model was used because it predicts the flow with high pressure gradients and circulations accurately [5]. The following assumptions were used:

- The flow is considered turbulent, steady, and incompressible.
- The system of differential equations was solved with time-independent properties (steady).
- Negligible heat transfer between the fog and collector was considered.

For the mentioned assumptions, the set of differential equations to be solved numerically is reduced to the following forms:

- Continuity:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

- Momentum:

$$u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[(\nu + \nu_t)\frac{\partial u_i}{\partial x_j}\right] + S_D \tag{2}$$

Where ν and ν_t are the kinematic and eddy kinematic viscosities, respectively. While The last term S_D represents the darcy-Frochhiemer equation which takes the form in the x-direction (flow direction) as:

$$(S_D)_x = -\rho\left(\frac{\nu}{\alpha}u_x + C_2\frac{|u|u_x}{2}\right) \tag{3}$$

Where α, and C_2 are the before-mentioned porous medium constants.

- Turbulent Kinetic energy (k):

$$u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_i}\left[(\nu + \nu_t)\frac{\partial k}{\partial x_j}\right] + G_k - \varepsilon \tag{4}$$

Where G_k represents the turbulence due to the mean velocity gradients.

- Rate of viscous dissipation (ε):

$$u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_i}\left[(\nu + \nu_t)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 S\varepsilon - C_2\frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} \tag{5}$$

Where ρ is the air density, and S is the mean rate-of-strain tensor. A specified number of particles were dispersed on a surface at a distance of 2.5h downstream of the collector, where h is the collector height and the surface is parallel to the fog collector. The dispersed particles must be on a surface parallel to the fog collector and have the same line of action.

Then the aerodynamic efficiency was calculated as shown in equation (6), Where N represents the number of particles, and s represents the solidity.

$$\eta_{ACE} = \frac{N_{collected}}{N_{released}} s \tag{6}$$

3. Results

The dimensions of the simulated collector are 12 meters in width and 4 meters in height. The wind tunnel maintains an undisturbed velocity at a constant 7 m/s. The fog collector was substituted with a porous jump, and this jump exhibits a pressure drop as indicated below:

$$(P1 - P2) = \Delta n \rho \left(\frac{\nu u_2}{\alpha} + C_2 \frac{u_2^2}{2} \right) \tag{7}$$

$P1$ and $P2$ represent the pressure before and after the fog collector, respectively. Δn stands for the thickness of the collector, and u_2 corresponds to the downstream velocity.

Initially, the created model was validated using experimental data. Figure (1) Depicts the vertical velocity profiles downstream of the fog collector. The x-axis on the graph represents the dimensionless velocity (u/u_∞), whereas the y-axis indicates the dimensionless length (y/h), with "y" signifying the vertical distance from the ground and "h" representing the collector height (in the case of Bresci, h=4 meters). According to Figure (1.a), the results show good agreement with Bresci's dataset at a distance of 10 meters downstream of the collector ($x = 10\,meters$). The discrepancies can be attributed to the substantial simplifications applied to the model. These simplifications include considering the porous jump as a flat surface, despite its actual behavior of bending due to accumulated droplets and its weight. Moreover, the model assumes a constant pressure drop, whereas in reality, the pressure drop increases as droplets accumulate. Figure (1.b) depicts the evolving velocity profiles at distances exceeding 10 meters, illustrating a smooth flow development.

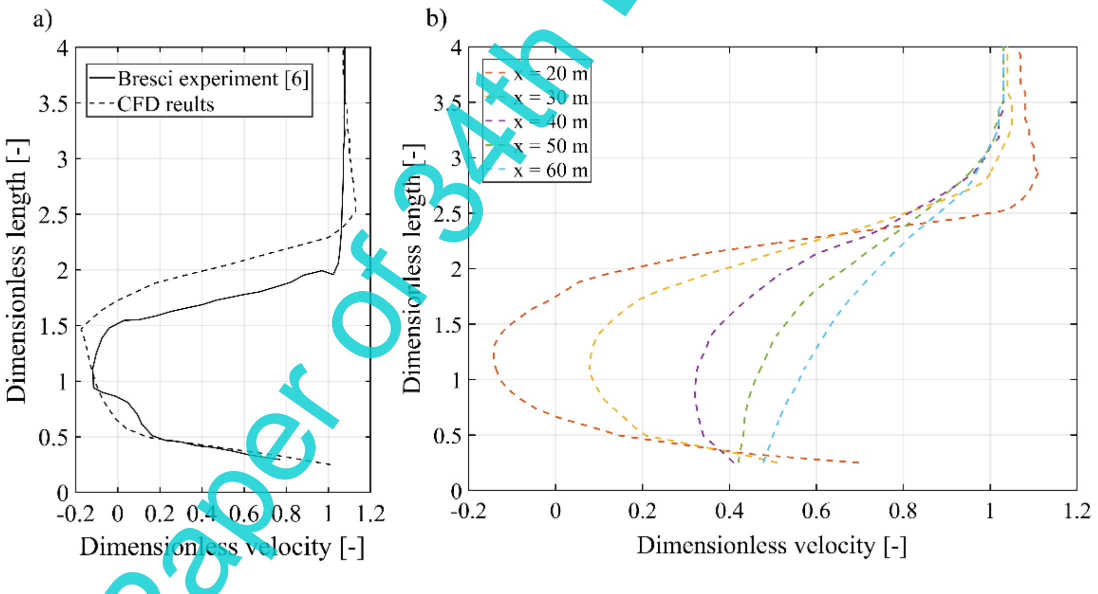

Fig. 1. Depicts vertical velocity profiles downstream of the fog collector. a) The validation using Bresci's dataset at $x = 10\,meters$, b) Vertical velocity profiles downstream of the collector at distances $x = 20, 30, 40, 50$ and $60\,meters$.

Figure (2) illustrates the difference between the distributed and undistributed flows around the fog collector for the top and side views. The results show good agreement with the CFD investigation carried out by Carvajal et al. [5]. According to Figure (2.a), the fog collector exhibits a significant pressure drop of 150 % due to its relatively high solidity (approximately 78%). It's important to recognize that this solidity increases as droplets accumulate on the collector leading to an even higher pressure drop. Based on the achieved results, only 42% of the released particles were collected by the porous jump leading to an aerodynamic efficiency of 33%. As shown in Figure (2.b), the high-pressure drop disrupts the flow influencing particles to pass by without being collected.

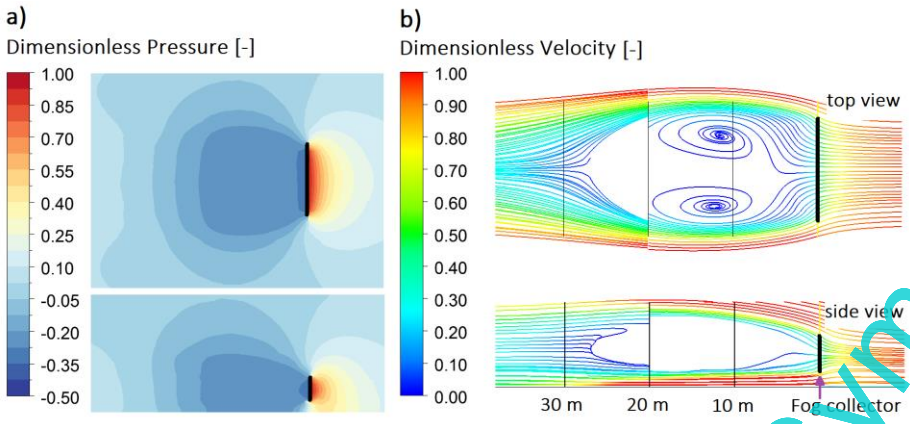

Fig. 2. Aerodynamic behavior around the fog collector. a) Dimensionless velocity streamlines, b) Dimensionless Pressure distribution $\left(\frac{P}{0.5\rho U_\infty^2}\right)$.

4. Conclusion

In this study, the aerodynamic collection efficiency of a fog collector was computed numerically. Initially, the Darcy-Forchheimer relation for Porous media was fitted to experimental data to find the porous media constants. Then, the Ansys fluent was used to perform the CFD simulation. The CFD results showed a high-pressure drop of 150 % and a velocity drop of 100 %. For future analysis, more consideration shall be considered regarding the porous media model because a lot of the collector structural parameters are fitted solely by three constants. Moreover, a transient simulation can be considered to investigate the change in pressure drop while the droplets accumulate on the collector.

5. Acknowledgments

This work was supported by the Student Grant Competition of the Technical University of Liberec under project No. SGS-2023-5323.

6. References

[1] M. Azeem, M. T. Noman, J. Wiener, M. Petru, and P. Louda, “Structural design of efficient fog collectors: A review,” *Environ. Technol. Innov.*, vol. 20, p. 101169, 2020, doi: 10.1016/j.eti.2020.101169.

[2] J. de D. Rivera, “Aerodynamic collection efficiency of fog water collectors,” *Atmos. Res.*, vol. 102, no. 3, pp. 335–342, 2011, doi: 10.1016/j.atmosres.2011.08.005.

[3] K. Steiros and M. Hultmark, “Drag on flat plates of arbitrary porosity,” *J. Fluid Mech.*, vol. 853, pp. 853R31-853R311, 2018, doi: 10.1017/jfm.2018.621.

[4] J.-K. Koo and D. F. James, “Fluid flow around and through a screen,” *J. Fluid Mech.*, vol. 60, no. 3, pp. 513–538, 1973, doi: 10.1017/S0022112073000327.

[5] D. Carvajal, L. Silva-Llanca, D. Larraguibel, and B. González, “On the aerodynamic fog collection efficiency of fog water collectors via three-dimensional numerical simulations,” *Atmos. Res.*, vol. 245, no. April, p. 105123, 2020, doi: 10.1016/j.atmosres.2020.105123.

[6] E. Bresci, “Wake characterization downstream of a fog collector,” *Atmos. Res.*, vol. 64, no. 1–4, pp. 217–225, Sep. 2002, doi: 10.1016/S0169-8095(02)00093-5.

[7] M. Azeem *et al.*, “Optimal Design of Multilayer Fog Collectors,” *ACS Appl. Mater. Interfaces*, vol. 12, no. 6, pp. 7736–7743, 2020, doi: 10.1021/acsami.9b19727.

[8] M. Damak and K. K. Varanasi, “A P P L I E D S C I E N C E S A N D E N G I N E E R I N G Electrostatically driven fog collection using space charge injection,” no. D, pp. 1–9, 2018, [Online]. Available: http://advances.sciencemag.org/.

[9] S. Sharifvaghefi and H. Kazerooni, “Fog harvesting: combination and comparison of different methods to maximize the collection efficiency,” *SN Appl. Sci.*, vol. 3, no. 4, pp. 1–11, 2021, doi: 10.1007/s42452-021-04518-3.

[10] K. Tuzharov, S. Iliev, and D. Gunev, “Aerodynamic study of the environmental vehicle model with the flow simulation CFD program,” *Ann. DAAAM Proc. Int. DAAAM Symp.*, vol. 29, no. 1, pp. 0495–0504, 2018, doi: 10.2507/29th.daaam.proceedings.072.

[11] Z. Čarija, I. Lučin, B. Lučin, and L. Grbčić, “Investigation of numerical simulation parameters on fluid flow around trash-racks,” *Ann. DAAAM Proc. Int. DAAAM Symp.*, vol. 29, no. 1, pp. 1046–1052, 2018, doi: 10.2507/29th.daaam.proceedings.149.