International Journal of Multiphase Flow 81 (2016) 46–53

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Numerical simulation and analysis of the effect of rain and surface property on wind-turbine airfoil performance

Aiden C. Cohan[a,b,*], Hamid Arastoopour[a,b]

[a] Department of Mechanical, Materials and Aerospace Engineering, Illinois Institute of Technology (IIT), Chicago, IL 60616, USA
[b] Wanger Institute for Sustainable Energy Research (WISER), IIT, Chicago, IL 60616, USA

ARTICLE INFO

Article history:
Received 24 July 2015
Revised 20 November 2015
Accepted 14 January 2016
Available online 23 January 2016

Keywords:
Multiphase flow
Rain simulation
Wind turbines
VOF
DPM

ABSTRACT

The performance of wind turbines is significantly affected by the atmospheric condition of their operating environment. Because rain is a common phenomenon in many parts of the world, understanding its effect on the performance of wind turbines provides valuable information in determining the site for a new wind farm. We developed a multiphase computational fluid dynamics (CFD) model to estimate the effect of rain by simulating the actual physical process of rain droplets forming a water layer over the blades by coupling the Lagrangian Discrete Phase Model (DPM) and the Eulerian Volume of Fluid (VOF) models. We applied our model to a wind-turbine blade airfoil and studied the effect of rain for different rainfall rates in addition to the effect of surface tension and surface property of the airfoil. We observed that, at low rainfall rates, the performance of the airfoil is highly sensitive to the rainfall rate. However, if the rainfall rate is high enough to immerse most of the airfoil surface under water, a further increase in the rainfall rate does not have a substantial effect on the performance of the airfoil.

1. Introduction

Wind energy is a renewable, pollution-free alternative to conventional fossil fuel power plants. Energy of the wind is retrieved using wind turbines; the more efficient a wind turbine is, the more energy it can capture from the wind. As wind turbines operate in open environments, their performance is continuously affected by surrounding atmospheric conditions, one of which is rain. Rain affects the performance of a wind turbine by reducing the momentum of the boundary layer as a result of rain droplets forming a water layer that adheres to its surface and consequently changes the condition of its blades surface properties.

Although rain is a common phenomenon in certain regions of the word, little investigation has been conducted to date of the effect of rain on the performance of wind turbines. There have been experimental studies on the effect of rain, but mostly in aviation applications (Dunham, 1987; Hansman & Barsotti, 1985; Marchman et al., 1987 and Bezos et al., 1992). There have been less than a handful of experiments with wind turbine applications (Corrigan & DeMiglio, 1985 and Al et al., 2011). However, all of the existing studies suggest that rain can significantly downgrade the performance of airfoils, aircraft wings, and wind turbines. Therefore, there is a need for comprehensive study of the performance of wind turbines under rainy conditions and reduction of the turbine efficiency under such conditions.

Computational Fluid Dynamics (CFD) has provided a reliable tool to simulate and analyze the physical phenomena involving fluids without the need for costly experimentation. Many CFD models have been developed for multiphase flows (Gidaspow, 1994 and Arastoopour et al., 1982) and used to simulate industrial processes (Abbasi & Arastoopour, 2011; Yang et al., 2010 and Arastoopour, 2001; Ghadirian & Arastoopour, 2016). There have been attempts to numerically simulate the effect of rain using CFD (Valenine & Decker, 1995; Wan & Wu, 2004 and Yeom et al., 2012). However, all of the aforementioned studies have somehow circumvented the problem of modeling rain droplets forming a continuous water layer on the structure under study. Some studies assumed the effect of rain can be included by modifying the air density, while others assumed the rain droplets remain droplets after hitting the surface. However, several experiments (Zhang et al., 2014; Thompson et al., 1995 and Bezos et al., 1992) have confirmed that rain droplets form a very transient water layer after hitting the surface of the object under study.

Only in a recent study conducted by our research group (Cai et al., 2013) was the formation of a continuous water layer from the rain droplets modeled on a 2D wind turbine airfoil in pursuit of simulating the actual physical process, which is the droplets

* Corresponding author.
E-mail address: acohan1@hawk.iit.edu (A.C. Cohan).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2016.01.006
0301-9322/

List of symbols

ρ	density of the fluid
p	pressure of the fluid
u_i	velocity of the fluid
μ	viscosity of the mixture
k	turbulent kinetic energy
ω	specific dissipation rate
$\widetilde{G}_k$	generation of k
G_ω	generation of ω
Y_k	dissipation of k
Y_ω	dissipation of ω
Γ_k	effective diffusivity of k
Γ_ω	effective diffusivity of ω
D_ω	cross-diffusion term
u_{Pi}	velocity of the parcel
m_P	mass of the parcel
C_{D_d}	Droplet drag coefficient
ρ_P	densities of the droplets
g_i	gravitational acceleration
α_q	volume fraction of the qth phase
ρ_q	density of the qth phase
$\boldsymbol{u}_q$	velocity of the qth phase
S_{α_q}	mass source term for the qth phase
F_i	momentum source term
V	volume of the computational cell
Δt	time-step of the continuous phase solver
u_{ri}	droplet relative velocity to continuous fluid phase
u_{fi}	velocity of the continuous fluid phase
C_L	airfoil lift coefficient
C_D	Airfoil drag coefficient
F_{st}	surface tension force
σ	surface tension

forming a water layer. In that study, the mass of the droplets was added to the water layer as a source term in its continuity equation by coupling the Lagrangian Discrete Phase Model (DPM) and Eulerian Volume of Fluid (VOF) methods. A similar approach has been used by Arienti et al. (2011) to model wall film formation and breakup.

In this study, we further improved the Cai et al. (2013) model by realizing that the rain droplets have momentum accompanying their mass, and this momentum is also transferred to the water layer by the impinging droplets. Therefore, our numerical model should also account for this phenomenon. Accordingly, we modified the Cai et al. (2013) model to add the momentum of the droplets, in addition to their mass, to the water layer formed by them.

We also included surface tension in our simulation and studied its effect through different surface properties (wettability) of the wind-turbine airfoil to see its effect on lift and drag coefficients. In addition, we performed simulations to examine the effect of rainfall rate on the performance of the airfoil. We ran our simulation for five different rainfall rates, ranging from light to heavy, to represent all types of rainy conditions encountered in the environment.

2. Numerical modeling

In this section, we present the governing equations of two multiphase models that we coupled to simulate the formation of a water layer from rain droplets, as well as the details of how the coupling was carried out. The two multiphase models are the Lagrangian Discrete Phase Model (DPM) and the Eulerian Volume of Fluid (VOF) model. DPM is a suitable model for simulating the motion of rain droplets before they enter the water layer, while VOF is appropriate for simulation of air and the water layer.

In order to simultaneously take advantage of the promising aspects of both models in simulating different parts of the process of water layer formation from rain droplets, we divided the computational domain into two subdomains. The first subdomain is the Lagrangian rain droplets and the second subdomain is the Eulerian free-stream air and water layer formed by droplets on the airfoil's surface. For the first subdomain, we used DPM to simulate the motion of rain droplets before they enter the water layer. For the second subdomain, the Eulerian VOF model was used to simulate the air that carries the rain droplets and also to capture the water layer formation and the interface between the two phases. We then used a coupled Lagrangian-Eulerian model to connect these two subdomains and carry out the simulation. The details of the coupling will be explained later in this section after we present the features and governing equations of the DPM and VOF models.

In the Lagrangian DPM model used to simulate the motion of rain droplets before they enter the water film, a droplet is assumed to be point mass and the fluid phase is treated as a continuum, while the dispersed phase is solved by tracking the droplets through the calculated flow field. The flow field can be obtained from any Eulerian model. In our simulations, the VOF model was used to calculate the flow field. Additionally, in order to render the simulation computationally affordable, we used the parcel approach to calculate droplet trajectories. In this approach, each parcel represents several droplets, all having the same diameter, density, and velocity. However, if the mass of the parcel is needed in any multiphase calculation, it should be manually set equal to the mass of an individual droplet times the number of droplets that the parcel is representing. The momentum equation for the droplets in the DPM written in a Lagrangian reference frame may be written as (ANSYS, 2014):

$$\frac{du_{Pi}}{dt} = \beta(u_i - u_{Pi}) + g_i\left(\frac{\rho_D - \rho}{\rho_D}\right) \tag{1}$$

where u_{Pi} and u_i are parcel and fluid phase velocity in the i-direction, respectively. $\beta(u_i - u_{Pi})$ is the drag force per unit particle mass and g_i is the gravitational acceleration. ρ_D and ρ are densities of the droplets and the fluid phase, respectively. β has a unit of 1/s and is defined as:

$$\beta = \frac{18\mu}{\rho_P d_P^2}\frac{C_{D_d} Re}{24} \tag{2}$$

C_{D_d} is the droplet drag coefficient for which we used the spherical drag law (Morsi & Alexander, 1972). Re is the relative Reynolds number, which is defined as:

$$Re \equiv \frac{\rho d_P |u_i - u_{Pi}|}{\mu} \tag{3}$$

The VOF model is a surface-tracking technique where the fluids share a single set of momentum equations, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. The fields for all variables and properties are shared by the phases and therefore properties appearing in the transport equations of the VOF model are determined by the presence of the component phases in each control volume and represent volume-averaged values. The tracking of the interface between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases using the geometric reconstruction scheme. For the qth phase, this continuity equation has the following form:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla.(\alpha_q \rho_q \boldsymbol{u}_q) = S_{\alpha_q} \tag{4}$$