# INVESTIGATION OF RAIN EFFECTS ON NACA0012 AIRFOIL WITH OPENFOAM

LIU NINGYU, ZHAO XIANG, SHAO JIANGYAN, SHU CHANG
*National University of Singapore, tslln@nus.edu.sg*

***Keywords:*** *rain effects, NACA0012 airfoil, DPM model, surface film model*

## 1. Introduction

Heavy rain can cause severe aerodynamic performance degradation to airfoil, particularly during take-off and landing stage. It has been deemed as a critical cause to many aviation accidents [1]. Rain condition can lead to aircraft aerodynamic penalties in several ways. For instance, raindrops can stick on and roughen the surface of wings. This phenomenon can induce changes in critical aerodynamic performance indicators such as pressure distribution as well as drag and lift force [2][3]. Thus, rainfall is considered as an important aircraft safety hazard. Nevertheless, accurate assessment of rain effect on aerodynamic performance poses many difficulties to both experimental and numerical study and it is still an open topic.

To analyse this problem numerically, it is noticed that OpenFOAM provides a powerful platform for simulation of fluid mechanics [4]. On the other hand, it was found that there is no built-in application in OpenFOAM for simulation of rain effect on airfoil that involves complex interactions among droplets, air flow and the structures, which is the focus of the present work. To achieve this aim, we developed an incompressible Eulerian-Lagrangian solver within OpenFOAM framework, named parcelFilmFoam. This solver integrates several OpenFOAM built-in libraries and applications. It is validated against benchmark data first, and then used to investigate aerodynamic performance of NACA0012 airfoil with dry air and heavy rain conditions.

## 2. Methodology

The main part of the solver for continuous phase is based on the OpenFOAM built-in solver named pisoFoam. Meanwhile, the Lagrangian library is linked to the main solver to implement Discrete Particle Model (DPM). Additionally, a thin film model is applied. DPM interacts with continuous phase and thin film through the source terms in the corresponding equations of main solver and surface film solver. They are introduced respectively in the following subsections.

### 2.1 Main Solver

To simulate incompressible viscous flow, there are semi-implicit method for pressure-linked equations (SIMPLE), pressure-implicit split-operator algorithms [5] (PISO) and a combination of SIMPLE and PISO algorithms (PIMPLE). In OpenFOAM, the corresponding built-in incompressible solvers are simpleFoam, pisoFoam and pimpleFoam. Among them, simpleFoam is steady-state solver, while both pisoFoam and pimpleFoam are transient solvers. pimpleFoam is featured as a large time-step transient solver for incompressible, turbulent flow. However, the Eulerian-Lagrangian solver we are developing needs to couple the detailed dynamics of DPM and thin film models with the flow field, and it requires small time-step. Thus, instead of using pimpleFoam, pisoFoam is adopted as the reference to build the solver for continuous phase.

### 2.2 DPM Model

The standard Lagrangian DPM is based on a translational force balance that is formulated for an individual particle. In this model, each particle represents a parcel of particles, and a DPM parcel is subject to gravity, drag force, and other forces when more complicated physics is considered. The particle trajectory is calculated by integrating the particle force balance equation. In this work, the drag force and gravity play the most important role.

$$\frac{du_p}{dt} = \frac{3\mu C_d Re_p}{4\rho_p D_p^2}\left(u - u_p\right) + g \tag{1}$$

Here, $u$ and $u_p$ are the air and particle velocity respectively; $g$ denotes the gravity acceleration; $\rho_p$ and $D_p$ are the density and diameter of the particle; $Re_p$ is the particle Reynolds number and $C_d$ is drag coefficient. Literature on similar approaches can be found in [6].

In Eulerian-Lagrangian approach, there are one-way coupled model [7] and two-way coupled model [8]. The one-way coupled model assumes that the motion of particles is affected by the continuous phase, while the continuous phase is not affected by the presence of particles. A two-way coupled model assumes that there is a two-way exchange of mass and momentum between the continuous phase and the particles. In the developed solver, the dynamics of the particle and the flow field are two-way coupled.

### 2.3 Surface Film Model

In order to model film layers with OpenFOAM, a dedicated mesh region for surface film needs to be extruded from the airfoil surface. The surface film is modelled inside the extruded mesh region. In this work, a thin film assumption is used, namely the velocity normal to the mesh wall is assumed to be zero and the wall-tangential diffusion is considered negligible compared to wall-normal diffusion. The surface film flow is solved with the continuity equation and the momentum equation, with a film layer thickness $\delta_f$ included.

$$\frac{\partial \rho \delta_f}{\partial t} + \nabla \cdot \left(\rho \delta_f U\right) = S_{imp} + S_{splash} + S_{sep} \tag{2}$$

$$\frac{\partial \rho \delta_f U}{\partial t} + \nabla \cdot \left(\rho \delta_f U U\right) = -\delta \nabla p + S_{\rho \delta_f U} \tag{3}$$

The continuity equation contains several source terms as shown in Eqn. (2). They include the mass added to the film layer due to impinging particles, as well as the mass deducted from the film layer due to rebounding, splashing particles, and separation of the film layer [9].

In the present case, the Weber number (We), which is defined as a measure of fluid's inertia to its surface tension, is used to decide the type of particle-film interaction. The particle-film interaction is classified as “stick” if We<5, “rebound” for 5<We<10, and “splash” when We>10.

## 3. Validation of Numerical Modelling

The parcelFilmFoam is used to simulate rain effects on NACA0012 airfoil. In this solver, the rain effects are investigated by simulation of the coupled dynamics between the particle, film and the flow field. The results will be validated by the measurement data from Hansman and Craig [10], as well as the numerical simulation results from Ismail et al. [11].

A 2D mesh model is built. The chord of airfoil is 1, and the calculation domain is 250×200×1 in meter. The total cell number is about 64,000. The inlet flow velocity is 18m/s. The angle of attack (AOA) is varied between 0° to 12°, and SA turbulent model is used.

### 3.1 Validation of Results without Rain

For the NACA0012 airfoil, there are many reliable benchmark data under dry air condition at various Reynolds number, (Re). Here, Re is defined as a ratio of fluid's inertial force to its viscous force. At high Re (Re=3.E6), the results have been verified based on the well-known benchmark data from Abbott and von Doenhoff [12]. The results indicate that, with the current setup, Cd (drag coefficient) is slightly overestimated, while Cl (lift coefficient) agrees well with the benchmark data, as shown in Figure 1. The results at a lower Re (Re=3.E5) are also included in the same plots, to illustrate the variation of force coefficients versus Reynolds number. It can be observed that the lower Re can lead to higher Cd and lower Cl.

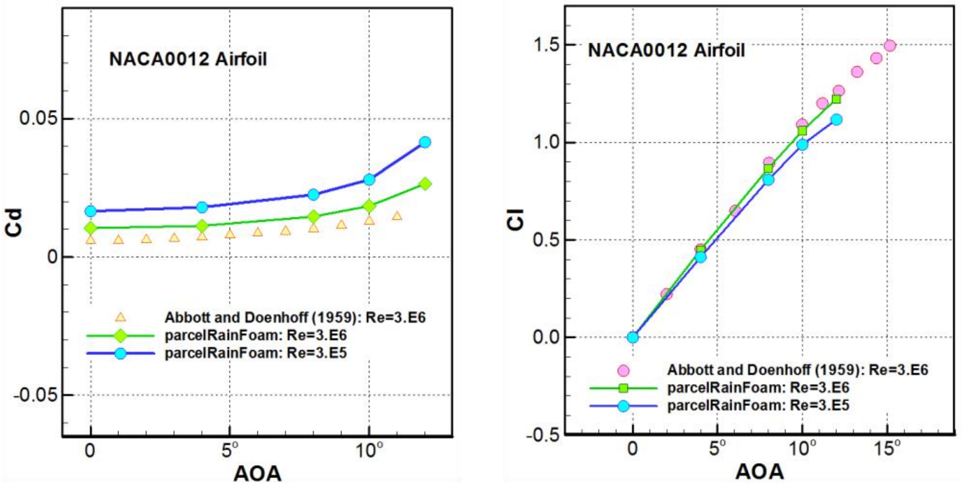

**Figure 1: Validation of force coefficients for NACA0012 without particle injection**

### 3.2 Validation of Results with Rain

To consider rain effects, the particle properties and the particle injection setting are needed. The particle properties are: density is 1000kg/m³ and diameter is 0.5mm. The particle injection settings are: the particle velocity is $u_p = 18m/s$ and $v_p = -3m/s$ in horizontal and vertical directions, and the liquid water content (LWC) is 30 g/$m^3$. The results considering rain effects simulated by parcelFilmFoam are compared with the experimental data from Hansman and Craig [10], and the numerical results from Ismail et al. [11], as shown in Figure 2.

The experimental data provided a lower Cl and a much higher Cd even for the dry air cases. As Hansman and Craig [10] claimed, their experimental data are the raw data and no correction was made. To demonstrate the difference, the experimental results with necessary corrections at low Re for dry air from Eastman et al. [13] are presented in the figures. In addition, the numerical simulation by ANSYS Fluent from Ismail et al. [11], with scale-down model (chord= 0.14m) and both the particle (diameter of 0.46mm) and the film effects are also shown. It can be seen that the present results of Cd and Cl by parcelFilmFoam are comparative to those of Ismail et al. [11] (The present simulation using normal scale, chord= 1m, and particle diameter 0.5mm). The “in-house Fortran Code” and the “Fluent” data in Figure 2 are obtained by using the same setting as that in parcelFilmFoam. They are included in plots for cross-comparison.

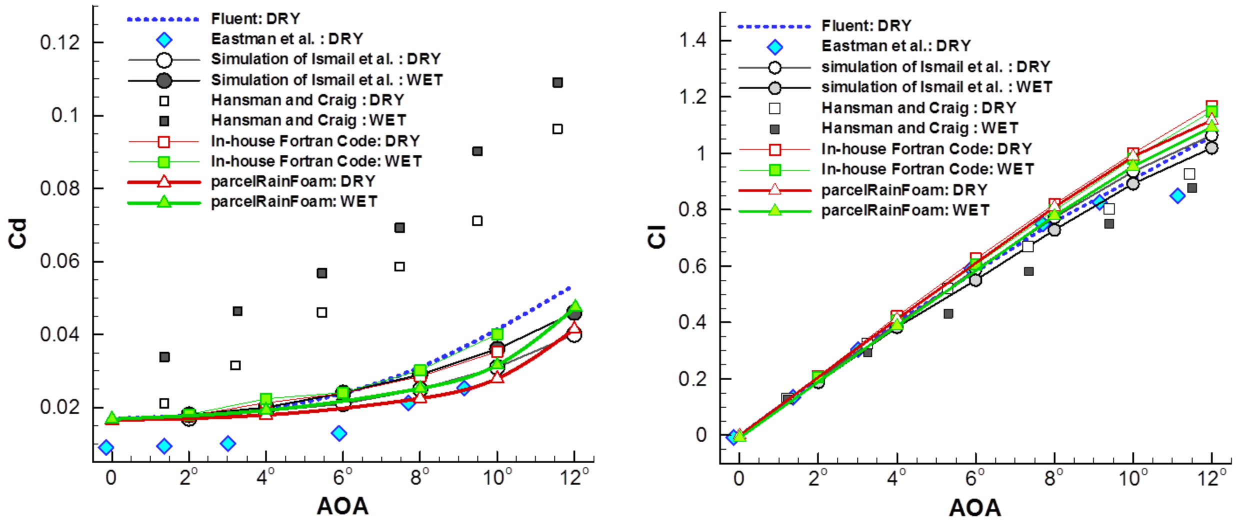

**Figure 2: Validation of force coefficients with particle injection at Re=3.E5**

## 4. Results and Discussion

A direct application of the unsteady solvers, such as pisoFoam, or parcelFilmFoam, may lead to obvious oscillation and even diverging of the simulation, especially for high AOA cases. To overcome this problem, a three-step simulation procedure is designed. It includes: i) obtain the steady flow field using simpleFoam; ii) obtain the unsteady flow without rain by parcelFilmFoam; iii) run parcelFilmFoam with DPM model and film model.

### 4.1 Results of DPM Model

To investigate the rain effects (with particle inject) on NACA0012 airfoil, the force coefficients with AOA=4° is recorded at each time step, as shown in Figure 3.

The particle injection starts at t=2s, and the particles start to hit the airfoil at t=2.17s. It can be seen that before the particles hitting the wing, there is a slight drag decrease and lift rising. The plot shows that rain (upon impact of particles) can cause the drag increase and lift decrease. At a later stage, raindrops (modelled as particles) can surround the airfoil and also present in its wake region. In this phase, the drag and lift coefficient become stable, and the final force coefficients can be obtained by averaging their value in this phase.

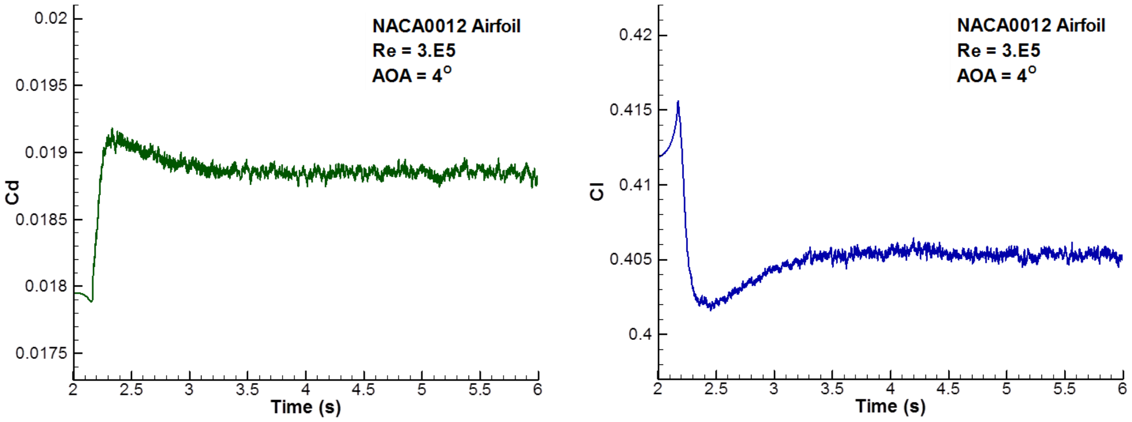

**Figure 3: Force Coefficient profiles at Re=3.E5 and AOA=4°**

### 4.2 Results of Surface Film

Two-dimensional flow is considered here. This means that the wall film velocity will be along the surface according to the thin film assumption. The film results are presented in term of the film velocity, $u_f$ and the film thickness, $\delta_f$ as shown in Figure 4. The raindrops exchange mass and momentum with the surface film during impact. When there is thin film on airfoil, it can in turn affect the motion of the impacting particles in several ways. For example, raindrops can stick on the surface of airfoil, and they can become part of the film and contribute to its mass and momentum. On the other hand, when the film is too thick or the curvature of film is too sharp at certain place (such as the place around the trailing edge), the water film can break into drops, and the drops will leave the film layer.

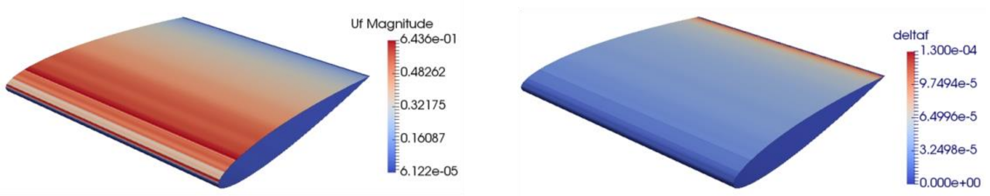

**Figure 4: Film velocity and film thickness at Re=3.E5, AOA=4° and t=4.8s**

## 5. Conclusions

In this work, based on OpenFOAM platform, a new incompressible Eulerian-Lagrangian solver, parcelFilmFoam, is developed. It is applied to simulate the aerodynamic performances of NACA0012 airfoil under rain conditions. Firstly, the solver is validated against the benchmark data from both experiments and numerical simulations. The aerodynamic efficiency degradation of NACA0012 airfoil in the rain is then investigated by parcelFilmFoam. Quantitative variations in drag and lift coefficients of the airfoil under rain condition are presented in this work. For the future work, we will keep on investigating the underlying physics as well as the influence of the interactions among raindrops, water film, air flow and the wing surface. In the meantime, we will also further explore the potential of OpenFOAM, to optimize the new solver, and to extend its applications to different types of airfoils, wings, and aircrafts.

## References

[1] Y. Cao, Z. Wu and Z. Xu, Effects of rainfall on aircraft aerodynamics. Progress in Aerospace Sciences, 71 (2014) 85-127.

[2] J. Valentine and R. Decker, A Lagrangian-Eulerian scheme for flow around an airfoil in rain. International journal of multiphase flow, 21 (1995) 639-648.

[3] R. Zhang and Y.-H. Cao, Study of aerodynamic characteristics of an airfoil in rain. Journal of Aerospace Power, 25 (2010) 2064-2069.

[4] C.J. Greenshields, OpenFOAM: The OpenFOAM foundation user guide version 5.0. OpenFOAM Foundation Ltd. 24th July 2017.

[5] R.I. Issa, Solution of the implicitly discretised fluid flow equations by operator-splitting. Journal of Computational Physics, 62 (1986) 40-65.

[6] A.B. Yu, B. Wright, Z.Y. Zhou, H.P. Zhu and P. Zulli, Discrete particle simulation of gassolids flow in a blast-furnace. Computers & Chemical Engineering, 32 (2008)1760-1772.

[7] J.R. Valentine and R.A. Decker, Tracking of raindrops in flow over an airfoil. Journal of Aircraft, 32 (1995) 100-105.

[8] J.R. Valentine and R.A. Decker, A Lagrangian-Eulerian scheme for flow around an airfoil in rain. International journal of multiphase flow, 21 (1995) 639-648.

[9] D.W. Stanton and C. Rutland, Multidimensional modeling of thin liquid films and spray-wall interactions resulting from impinging sprays. International Journal of Heat and Mass Transfer, 41 (1998) 3037-3054.

[10] R.J. Hansman and A.P. Craig, Low Reynolds number tests of NACA 64-210, NACA 0012, and Wortmann FX67-K170 airfoils in rain. Journal of Aircraft, 24 (1987) 559-566.

[11] M. Ismail, Z. Wu, A. Bakar and S. Tariq, Aerodynamic characteristics of airfoil cruise landing and high lift configurations in simulated rain environment. Journal of Aerospace Engineering, 28 (2014) 04014131.

[12] I.H. Abbott and A.E. von Doenhoff, Theory of wing section: including a summary of airfoil data. New York: Dover Publication, (1959).

[13] N. Eastman, N. Jacobs and A. Sherman, Airfoil section characteristics as affected by variations of the Reynolds number. NACA TR-586 (1937) 231.