Application of different Lagrangian Particle Tracking techniques for water impingement

F. Petrosino 1, D. De Rosa 2 and G. Mingione 3
CIRA Italian Aerospace Research Center, Capua (CE), Italy

Abstract. Water impingement on aerodynamic surfaces represents how much flow rate water mass impact on the surfaces, respect to total mass flow rate. This parameter, together with Liquid Water Content, is important in ice prediction since it defines water mass distribution on solid surface, by which depends several other problems like freezing fraction, anti-icing system, etc. In this work we focus on the lagrangian method to calculate droplets’ trajectory, comparing the results against 2D and 3D experimental data.

1. Introduction
In the framework of icing on aircraft study, the first phenomena to investigate is the impingement of water droplets on the surfaces of aircraft. Numerical approaches are usually implied to support the experimental testing and to provide fast responses when designing ice protection systems. In order to compute the impingement efficiency, we need to follow the water particles presents in the air flow field, from their starting positions to the hitting positions on the surfaces of an aircraft. This can be done using two approaches: Eulerian approach that means fix the observation volume and solve the transport equation of the water inside the main air flow, Lagrangian approach that means follow each water particle solving the equations of motion for the particle interacting with the main air field. In CIRA both approaches are developed and implemented in 2D and 3D tools, validated trough the literature and experimental test cases. In this work, we focus on the use of lagrangian approach. A comparison between Lagrangian model implemented in CIRA software Multi-Ice[4] and in open-source framework OpenFOAM[3], has been made. The robustness of the methodology and the accuracy of the approach are discussed. The method has been used on classical two- and three-dimensional test cases for which experimental data are available in literature. The results are compared with experiments and also using different numerical aerodynamic solutions.

2. Lagrangian impingement procedure
Collection Efficiency represents how much flow rate water mass impinging on the surface, respect to total mass flow rate: this parameter is important in ice prediction since it defines water mass distribution on solid surface, by which depends the freezing fraction, and the impact limits, defined by the two limit impinging droplets. The droplets’ trajectory is usually calculated
writing the second dynamic law and solving the ordinary differential equation system. The assumptions are:

- the droplets are spherical and of the same size;
- droplets do not deform, and they do not collide or coalesce each other;
- droplets are influenced only by aerodynamic and gravitational force;
- water concentration is small enough to assume they do not influence the aerodynamic field.

Referring to 2D case, the scheme of single droplet’s velocity decomposition is shown in figure 1: it is present a relative velocity $V_{\text{rel}}$, composition of air velocity $V_a$ and droplet’s velocity $V_d$; $u_{\text{rel}}$ and $w_{\text{rel}}$ are the relative velocity components respect to $x$ and $z$ axes.

![Diagram of droplet velocity decomposition](image)

**Figure 1.** Droplet velocity decomposition scheme.

Aerodynamic and gravitational forces act on the droplet; the aerodynamic force can be written as

$$F_a = \frac{1}{2} \rho_a C_D A_d V_{\text{rel}}^2$$

in which $A_d$ is the droplet surface and $C_D$ is the drag coefficient; the latter assumes different expressions according to the Reynolds droplet number ($Re_d = \frac{V_{\text{rel}} d}{\nu}$).

$$\begin{cases} 
C_D = \frac{24}{Re_d} (1 + 0.15 Re_d^{0.687}) & Re_d \leq 1000 \\
C_D = 0.4 & Re_d > 1000 
\end{cases}$$

Decomposition of aerodynamic force along $x$ and $z$ axes is

$$F_{ax} = \frac{1}{2} \rho_a C_D A_d u_{\text{rel}}^2$$

$$F_{az} = \frac{1}{2} \rho_a C_D A_d w_{\text{rel}}^2$$

The gravitation force has only one component, along the $z$ axis, and its expression is

$$F_g = m_d g \left( 1 - \frac{\rho_a}{\rho_w} \right)$$

in which $m_d$ is the droplet mass, equal to $\rho_w V_p$. The motion equations along the considered axes are:

$$\begin{cases} 
\frac{du_d}{dt} = \frac{F_{ax}}{m_d} \\
\frac{dw_d}{dt} = \frac{F_{az} + F_g}{m_d} 
\end{cases}$$
System 5 is an ordinary differential equation, which can be solved using a Runge-Kutta method, this is the procedure which has been implemented in lagrangian CIRA solvers and in OpenFOAM solvers.

In two-dimensions, the Collection Efficiency can be calculated as a ratio between two linear density [6] (impinging mass per arc length on total mass per length upstream): since the hypothesis particles have the same size has been made, it can be calculated simply as the ratio between the distance upstream and the distance on solid surface, as shown in the left part of figure 2. In the three-dimensional case, using the same procedure of the two-dimensional case, it is possible to define two surfaces density, for impinging at total mass, and calculate collection efficiency as the ratio between them, as shown in the right part of figure 2.

![Figure 2](image)

**Figure 2.** Definition of collection efficiency for 2D case (left) and 3D case (right).

### 3. Impingement on 2D airfoil NACA0012

The first case is a flow around a NACA0012 airfoil. The free-stream flow velocity is 44.39 m/s and the angle of attack is 0 degrees. The median diameter of the droplets is 20 µm. The collection efficiency is compared with experimental results presented in Morency et al [5]. In the left part of figure 3, the impingement coefficient computed with Multi-Ice and OpenFOAM is compared. Multi-Ice has an aerodynamic module that compute the air velocity using a panel method, while OpenFOAM compute the air flow field using a complete CFD approach, in both the codes the droplet motion equations are implemented and solved in a similar way. As shown, the results have small differences in the stagnation point, due to velocity of air different results. Multi-Ice has the ability to use an external flow field for the air velocity, we used the flow field made by OpenFOAM and another made in the same conditions by the in-house CFD code ZEN. Both the codes use similar numerical approach, but OpenFOAM is based on an unstructured grid, while ZEN uses structured mesh. This difference lead to greater differences in the stagnation region when the grid is less regular as in the unstructured case.

### 4. Impingement on 2D airfoil NACA652–415

The second test case is the impingement efficiency distribution for NACA 652–415 airfoil. The chord of the airfoil is 36 inches, freestream velocity 175 mph, angle of attack 0 and 4 degrees, droplets diameter 21 µm. The experimental results are available in the NASA report made by Papadakis et al [2], shown in 4.

The experimental data for the 0 degrees angle of attack case, presents the maximum impingement very close to the leading edge. As the angle of attack was increased to 4 degrees, as shown in figure 4, the location of the maximum impingement moved toward the lower surface.
Figure 3. Collection efficiency for NACA0012 test case, on the left comparison between panel method and CFD, on the right comparison using different CFD flow fields.

Figure 4. Experimental collection efficiency for NACA642-415 test case, angle of attack 0 and 4 degrees.

The impingement limits moved toward the leading edge on the upper surface and toward the trailing edge on the lower surface with respect to the 0 degrees case.

Figure 5. Collection efficiency for NACA642-415 test case, on the left comparison between panel method and CFD, on the right comparison using different CFD flow fields, angle of attack 0 degrees.

In figure 3 the collection efficiency computed with Multi-Ice and OpenFOAM for the case with angle of attack of 0 degrees is compared. The aerodynamic simulated with panel method
lead to better results than the CFD results. As in the previous test case, using two different flow field from structured and unstructured solvers in Multi-Ice, the solution with a structured computational domain is better than an unstructured one. Similar results are in figure 6, for the case with angle of attack of 4 degrees. In this case, the impingement results using OpenFOAM or ZEN flow field are equivalent, and in good agreement with experiments.

![Figure 6](image1.png)

Figure 6. Collection efficiency for NACA642-415 test case, on the left comparison between panel method and CFD, on the right comparison using different CFD flow fields, angle of attack 4 degrees.

5. Impingement on 3D wing

The three-dimensional test case is a full-scale reflection plane tail model consisting of the outboard portion of a general aviation business jet tail. The tail tip consists of a semi cylindrical cap. The tail airfoil is a symmetric 8% thick NACA64A008 section and it is kept constant from root to tip. The mean aerodynamic chord is 0.956 meters. The subsonic flow around the test case was at Mach number 0.23, Reynolds value $5.03 \times 10^6$ based on the mean aerodynamic chord and 0 degrees angle of incidence. A cloud of water droplets having a droplets diameter of 20.36 microns is considered. The case was extensively studied in the NASA Glenn Icing Research Tunnel as for aerodynamic performance and impingement efficiency [1].

![Figure 7](image2.png)

Figure 7. Collection efficiency for NACA64A008 test case, on the left angle of attack 0 degrees, on the right 6 degrees.

Multi-Ice is a 2D code, in this case only OpenFOAM was used, with the results shown in figure 7. The impingement limits are in good agreement with experiments, while the peak values
are lower than expected. As for the previous results, the aerodynamic flow field lead to this underestimation.

6. Conclusion
Different Lagrangian tools for water droplets impingement prediction have been tested in the framework of a panel aerodynamic solver and CFD solver. The method has been applied to classical two- and three-dimensional test cases for which experimental data are available in literature. The Multi-Ice tool shows very good agreement with experiments, using the panel solver for aerodynamic flow field computation, or CFD solution from structured solver. The impingement method implemented in OpenFOAM has good agreement with respect to experimental measurements in terms of impingement limits, while it underestimates the peak value in almost any condition. The OpenFOAM implementation is able to handle also 3D cases, a capability not available in the Multi-Ice tool. Also in this case, the results are in line with the 2D solutions. Further research will be focused on analyzing the three-dimensional simulation in supercooled large droplets (SLD) regime through advanced wall-droplet interaction models.

Funding Sources
This project has received funding from the Clean Sky 2 Joint Undertaking under the European Union’s Horizon 2020 research and innovation programme under grant agreement No 945521.

References
[1] Papadakis M. et al. “Experimental Investigation of Water Droplet Impingement on Airfoils, Finite Wings and an S-Duct Engine Inlet”. In: NASA/TM-2002-211700 ()
[2] Papadakis M. et al. “Large and Small Droplet Impingement Data on Airfoils and Two Simulated Ice Shapes”. In: Glenn Research Center, NASA/TM-2007-213959 ()
[3] Ljungskog E. “Description of reactingParcelFilmFoam”. In: CFD with OpenSource software course at Chamlers University of Technology (2014).
[4] Brandi V. Mingione G. “A Code for the Evaluation of Ice Accretion on Multi-Element Airfoils”. In: CIRA TN-96-089 (1996).
[5] Paraschivoiu I. Morency F. Tesok F. “Anti-Icing System Simulation using CANICE”. In: Journal of Aircraft, Vol. 36, No. 6, pp. 999-1006 (1999).
[6] G. Poots et al. “Aircraft icing”. In: Philosophical Transactions of the Royal Society of London. Series A: Mathematical, Physical and Engineering Sciences 358.1776 (2000), pp. 2873-2911.