FENSAP-ICE: a comprehensive 3D Simulation Tool for In-flight Icing

Wagdi G. Habashi*, François Morency and Héloïse Beaugendre
Computational Fluid Dynamics Laboratory, McGill University

1. Introduction

Ice accretion simulations are usually based on 2D and quasi-3D inviscid panel or Euler flow computations for the air, on Lagrangian techniques for droplet impingement, and on a control volume analysis of the mass and heat transfer for ice accretion. Current computational fluid dynamics (CFD) technologies could overcome many of the limitations of these approaches such as their limited ability to handle compressibility, three-dimensionality and flow recirculation and/or separation. It is only at the cost of solving models based on partial differential equations, that a comprehensive three-dimensional approach to icing simulation becomes possible.

Our modern approach views the simulation of icing as the solution of the compressible Navier-Stokes equations (in our case with FENSAP*: Finite Element Navier-Stokes Analysis Package), the computation of the collection efficiency distribution by an Eulerian method with DROP3D, and the prediction of the ice accretion shape by a shallow-water icing model with ICE3D. As shown on figure 1, the three modules are set in a complete interactive loop. In addition, to calculate the heat loads for de-icing or anti-icing, a conduction module CHT3D has been developed to calculate the heat flux in the metal skin, together with the convection in the flow external and internal to the wing, in a fully conjugate heat transfer mode.

The following methods are used for the numerical solution of the icing problem:

- Each of the three systems of PDEs (air/droplets/ice) is solved independently from the others; selected variables are passed between solvers.

- Both the flow and droplet solvers are weak-Galerkin finite element models, on structured, unstructured and hybrid meshes.

- The distortion of the solid surfaces caused by ice growth is introduced automatically and transparently into the flow solver through an Arbitrary Lagrangian Eulerian (ALE) scheme.

![Fig.1 Code interactions with ICE3D](image-url)
2. DROP3D

Lagrangian particle-tracking techniques are widely used to compute the droplet impingement on aircraft parts. DROP3D, on the other hand, is an Eulerian model for airflow containing water droplets. Suitable variables are then computed only at the nodes of the discrete domain where the airflow variables are known, so no particles have to be tracked as they go through the mesh as in the Lagrangian approach. On complex or 3D geometries, the well-known difficulties encountered by a Lagrangian method (long calculations times, difficulty near walls, difficulty in determining impingement limits, sparsely seeded particles, etc.) are completely absent from the Eulerian approach.

2.1 Mathematical model

The Eulerian model used for the impingement calculations has been introduced for 3D applications in Bourgault et al. This is essentially a two-fluid model consisting of the Navier-Stokes or Euler equations for the air, augmented by new droplet-related continuity and momentum equations. The latter are, in non-dimensional form, respectively:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}_d) = 0 \quad (1)$$

$$\frac{\partial \mathbf{u}_d}{\partial t} + \mathbf{u}_d \cdot \nabla \mathbf{u}_d = \frac{C_D}{24K} \mathbf{Re}_d (\mathbf{u}_a - \mathbf{u}_d) + \left( 1 - \frac{\rho_d}{\rho_a} \right) \frac{1}{Fr^2} \mathbf{g} \quad (2)$$

where \( \alpha(x, t) \) and \( \mathbf{u}_d(x, t) \) are mean values of the water volume fraction and of the droplet velocity, respectively, over a fluid element at location \( x \) at time \( t \). The first term on the right-hand-side of Eq. (2) represents the air drag force on droplets, while the second term represents the buoyancy and gravity forces. The non-dimensional air velocity, \( \mathbf{u}_a \), is obtained by solving either the Navier-Stokes equations or the Euler equations, both through FENSAP.

The two-fluid model assumes spherical droplets of a single and uniform size, usually chosen to be the median volume diameter (MVD) of the sample size distribution. The spherical droplet approximation is valid for droplet Reynolds numbers below 500. No collision or mixing between the droplets is accounted for, as it can be shown not to be important for icing situations. An empirical equation gives the drag coefficient for spherical droplets:

$$C_D = \begin{cases} 
24 / \mathbf{Re}_d & \text{for } \mathbf{Re}_d \leq 1300 \\
0.4 & \text{for } \mathbf{Re}_d > 1300
\end{cases} \quad (3)$$

A finite element Galerkin formulation is used to numerically discretize the equations, with a streamline upwinding Petrov-Galerkin (SUPG) term added. Details of the numerical method and validations can be found in Bourgault et al.

2.2 Results

Once an airflow solution is obtained for a given geometry, DROP3D can be used for the calculation of the droplet impingement limits and the mass of water captured by the body. DROP3D has been used successfully to compute results on simple as well as on rather complex geometries. In figure 2, the collection efficiency distribution around the nose and cockpit of a Convair-580 is plotted as an example of results. An inviscid airflow solution is first calculated around the nose, then DROP3D is used to calculate collection efficiency for several MVD sizes, only three being shown here, ranging from small sizes to Supercooled Large Droplets (SLD).
3. ICE3D

Once the friction force and the heat fluxes are known from the airflow solution (FENSAP) and the mass rate of water caught is known from the impingement module (DROP3D), the ice accretion rate can be assessed.

3.1 Mathematical model

A new equilibrium model was introduced in the paper by Bourgault et al.\(^5\) to predict the ice accretion and water runback on the surface. Figure 3 shows the heat and mass transfer phenomena taken into account by the model.

![Fig.3 Heat and mass balance in a thin film.](image)

The velocity \( u_f \) of the water in the film is a function of coordinates \( x = (x_1, x_2) \) on the surface and \( y \) normal to the surface. A simplifying assumption consists in taking a linear profile for \( u_f(x, y) \), with a zero velocity imposed at the wall, i.e:

\[
    u_f(x, y) = \frac{y}{\mu_w} \tau_{wall}(x, y) \tag{4}
\]
where $\tau_{\text{wall}}$, the shear stress from the air, is the main driving force for the water film. For very thin films, the terms of order higher than one in the velocity profile are negligible. In icing or anti-icing simulation, film thickness is seldom above 10 $\mu$m. By averaging along the thickness of the film, a mean velocity is given as follows:

$$\bar{u}_f(x, y) = \frac{1}{h_f} \int_0^{h_f} u_f(x, y) dy = \frac{h_f}{2\mu_w} \tau_{\text{wall}}(x, y)$$  \hspace{1cm} (5)

The resulting system of partial differential equations is the following:

**Mass Conservation**

$$\rho \left[ \frac{\partial h_f}{\partial t} + \text{div}(\bar{u}_f) \right] = U_\infty LWC \beta - \dot{m}_\text{evap} - \dot{m}_\text{ice}$$  \hspace{1cm} (6)

where the three terms on the right hand side correspond, respectively, to the mass transfer by water droplets impingement (source for the film), the evaporation and the ice accretion (sinks for the film).
Energy Conservation

\[
\rho_w \left[ \frac{\partial h f C_w \tilde{T}}{\partial t} + \text{div}(u f h f C_w \tilde{T}) \right] = \left[ C_w \tilde{T}_{d,\infty} + \frac{1}{2} \|u_d\|^2 \right] \times U_\infty LWC \beta - 0.5 \left( L_{\text{evap}} + L_{\text{subl}} \right) \tilde{Y}_{\text{evap}} + \left( L_{\text{fusion}} - C_{\text{ice}} \tilde{T} \right) \tilde{Y}_{\text{ice}} + \sigma (T_\infty^4 - T^4) + \tilde{Q}_h
\]

where the first three terms on the right hand side model, respectively, the heat transfer caused by the supercooled water droplets impingement, the evaporation and the ice accretion. The last two terms represent the radiative and convective heat transfer.

The coefficients \( \rho_w, C_w, C_{\text{ice}}, L_{\text{evap}}, L_{\text{subl}}, L_{\text{fusion}} \) represent physical properties of water, while \( \tilde{T}_{d,\infty}, U_\infty, LWC, \sigma, \) and \( T_\infty \) are airflow and droplet parameters specified by the user. The ambient icing conditions completely determine those values.

The Eulerian droplet module provides local values for the collection efficiency \( \beta \) and the droplet impact velocity \( u_d \). The flow solver provides the local wall shear stress \( \tau_{\text{wall}} \) and the convective heat flux \( \tilde{Q}_h \). The evaporative mass flux is recovered from the convective heat flux using a parametric model\(^7\). There remains three unknowns: the film thickness \( h_f \), the equilibrium temperature \( T \) within the air/water film/ice/wall interface, and the instantaneous mass accumulation of ice \( \tilde{m}_{\text{ice}} \). Compatibility relations are needed to close the system. One way to write them is the following:

\[
\begin{align*}
    h_f &\geq 0 \quad (8a) \\
    \tilde{m}_{\text{ice}} &\geq 0 \quad (8b) \\
    h_f \tilde{T} &\geq 0 \quad (8c) \\
    \tilde{m}_{\text{ice}} \tilde{T} &\geq 0 \quad (8d)
\end{align*}
\]

3.2 Results

Ice accretion computation is carried out through a full interaction between FENSAP, DROP3D and ICE3D. A turbulent airflow solution is first calculated on a clean (non-iced) geometry, then DROP3D computes the droplet impingement and ICE3D the nodes’ displacement due to ice accretion. A new solution and mesh are then automatically generated by FENSAP with the ALE package. Figure 5 shows an example of a 260-second accretion on a NACA 0012 airfoil.
4. Conclusions

FENSAP-ICE is a comprehensive icing code for in-flight ice simulations in 3D, as well as in 2D. It is built in a modular fashion with four modules (three are described in this paper), each having a specific task but with all being linked. The first module, FENSAP, takes care of the airflow computation by either Euler or Navier-Stokes models. DROP3D uses the calculated airflow for droplet impingement calculations. Finally, ICE3D uses shear forces and heat fluxes from the airflow calculation of FENSAP and the water catch from DROP3D to yield the 2D shape of the ice on the 3D surface. FENSAP-ICE has been successfully used to predict collection efficiencies on 2D and 3D bodies. The ice calculation module is in the final phases of its validation process, with quite promising results.

* CORRESPONDING AUTHOR:
Dr. Wagdi G. Habashi  
Director, CFD Lab  
Department of Mechanical Engineering  
McGill University  
688 Sherbrooke West, Suite 715  
Montreal, QC, Canada H3A 2S6

Tel: +1 (514) 398-3747, Admin. Asst: 6005, Fax: 2203  
Wagdi.Habashi@mcgill.ca
6. References


