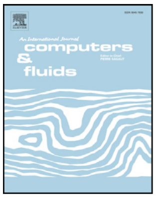
Computers and Fluids 161 (2018) 189–198

Contents lists available at ScienceDirect

Computers and Fluids

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



Numerical algorithms for infinite swept wing ice accretion

P. Lavoie*, S. Bourgault-Côté, E. Laurendeau

Department of Mechanical Engineering, Polytechnique Montréal, C.P. 6079, succ. Centre-ville, Montréal (Qc), H3C 3A7, Canada

ARTICLE INFO

Article history:
Received 28 March 2017
Revised 3 October 2017
Accepted 23 November 2017
Available online 24 November 2017

Keywords:
Ice accretion
Swept wing
Convective heat transfer
Multi-layer icing
Conservative ice growth

ABSTRACT

A method to simulate ice accretion on swept wings using a two-dimensional Navier-Stokes airflow solver, an Eulerian droplets field formulation and a PDE based thermodynamic model is presented. The current approach, called 2.5D in this paper, is developed by applying a rotation of the system of coordinates to allow simulations on a cut normal to the leading edge. The crossflow velocity component in the simulation plane is computed with the third momentum equation simplified by assuming null derivatives in the spanwise direction. The consistency of the ice accretion software, NSCODE-ICE, is first verified by a grid convergence study on 2D icing cases and an icing layers convergence study on 2D and 2.5D icing simulations. The validation of the approach is performed on an unswept two-dimensional glaze ice case and on a rime ice case. The current 2.5D approach is then validated on an infinite MS-317 swept wing and compared to results from usual semi-empirical infinite swept wing corrections. The new approach succeeds to model the stagnation line where other correction methods fail, leading to a better representation of the convective heat transfer in this area.

1. Introduction

Ice accretion on aircraft poses a serious safety hazard by inducing a degradation in aerodynamic performance, by adding weight and by affecting the aircraft's probes and sensors. Two-dimensional numerical tools to predict ice accretion on airfoils have been widely studied and used in the industry in the past decades (e.g. CANICE, LEWICE). However, ice accretion on modern aircraft exhibits strong 3D effects [1,2], for instance on swept wings, hence motivating the development of three-dimensional ice accretion tools (e.g. FENSAP-ICE [3], ONERA3D [4], NSMB-ICE [5]). Although full 3D icing simulations are necessary to obtain an accurate prediction of the ice accretion, they are time consuming and too costly for insertion into design loops, or even certification process [6]. More efficient tools are needed since many configurations and icing conditions have to be tested.

One solution is to apply a numerical correction to the 2D icing simulation results. For instance, Dorsch and Brun [7] presented a hingewise correction to the freestream conditions by performing the simulation on a plane normal to the leading edge of the wing. Correction for the flow velocity and collection efficiency were provided. Pueyo [8] revised Dorsch's method and suggested a streamwise approach to correct the droplet collection efficiency for the sweep angle by considering the orientation of the wall cells compared to the flow velocity direction. Two other numerical corrections are added to account for finite wings by correcting the angle-of-attack and the liquid water content (*LWC*) following comparisons with 3D airflow and droplets results. These approaches were proven to enhance the results compared to a purely 2D simulation and are much faster than full 3D icing methods.

Another option is to use a quasi-3D method that couples 3D low-fidelity flow solutions (e.g. Vortex Lattice Method) with 2D viscous solutions. The 2D viscous solutions are computed on sections along the span and the results are used to add a viscous correction at each section of the 3D wing by interpolating the lift coefficient. This approach accounts for the effect of a finite wing, sweep angle, twist, and local viscous effects. For instance, Silva et al. [9] present a quasi-3D method coupling the Non-Linear Modified Lifting Line Method and the 2D viscous results from RANS equations. The method thus relies on the accuracy of the 2D viscous results.

The three-level correction method presented by Pueyo [8] does not limit itself to the correction of the collection efficiency. The local angle of attack and liquid water content (*LWC*) are corrected according to 3D airflow and droplet trajectories results. Hence, the method can also account for the effect of a finite wing, sweep angle, twist and viscous effects, but at higher cost since a 3D solution is required. It also accounts for the presence of other bodies near the wing, such as nacelles.

In the methods discussed above [7–9], crossflow effects are approximately accounted for. For instance, the 2D leading edge stag-

---

* Corresponding author.
E-mail address: pierre.lavoie@polymtl.ca (P. Lavoie).

https://doi.org/10.1016/j.compfluid.2017.11.015

nation point does not transform into a stagnation line characteristic of swept wings.

In this paper, a two-dimensional icing prediction tool is extended to accurately solve ice accretion on an infinite swept wing. The proposed approach reduces the empiricism found in standard 2D correction methods by solving the three dimensional systems of equations simplified for infinite swept wing geometries to allow simulation on a topologically two-dimensional domain [10]. A crossflow equation is thus accounted for in the airflow and droplet solvers to directly compute the flow solution, the droplet collection efficiency, the convective heat transfer coefficient and the runback water behavior. Based on the infinite wing hypothesis, the method assumes constant variables in the spanwise direction and uses the classical geometric transformation to solve on a wing section normal to the leading edge. As this approach simulates 3D phenomena on a 2D grid, it is referred as 2.5D throughout this paper.

This paper first presents the icing software used for the simulations and describes the basic equations and the infinite swept wing corrections applied to each module. Second, the numerical consistency of the software is verified through a grid convergence study and an icing layers convergence study. Third, validation of the software is performed on two-dimensional unswept icing cases. Fourth, the infinite swept wing (2.5D) method is validated with experimental data on the MS-317 infinite swept wing. A discussion will follow as to compare with Dorsch's [7] and Pueyo's [8] standard sweep correction methods.

2. Aero-icing framework

A two-dimensional aero-icing software, NSCODE-ICE [11], is under development at Polytechnique Montréal since 2012. This software supports the methods presented in this paper. It is composed of five modules: a mesh generation module able to handle complex ice shapes, NSGRID2D [12], a Reynolds-Averaged Navier-Stokes (RANS) airflow solver, NSCODE [13], an Eulerian droplet solver [14], a thermodynamic solver [15] and a geometry evolution solver. NSCODE-ICE is able to perform multi-layer icing and deal with multi-elements airfoils via the use of an overset mesh approach [13].

Multi-layer ice accretion simulations on airfoils are performed with a decoupled approach by running the five modules sequentially in a loop. Mesh regeneration is performed for each new layer on the accreted geometries. This multi-layer process approximates the unsteady nature of ice accretion with a quasi-steady approach [16], as the different modules are run in steady mode without contributions from previous simulations except for the wall temperature and the water mass accumulation. Following the assumption that a thin layer of ice does not significantly impacts the solution of the other modules, the quasi-steady assumption is valid.

2.1. Navier–Stokes flow solver

2.1.1. 2D solver

NSCODE is a finite volume cell-centered RANS solver running on multiblock structured grids in two dimensions. Overset grids are also supported. It is developed to run in parallel on shared memory (openMP). It incorpotates various accelerations techniques, time integration scheme and turbulence models [17]. Also, unsteady simulations can be performed using a dual-time stepping scheme or a non linear frequency domain scheme (NLFD) [13].

In this paper, all airflow results are solved as steady flows using an explicit five stages Runge–Kutta scheme with three level multigrid [18]. Turbulence is modeled with the Spalart–Allmaras model and Boeing's extension [19] to account for the surface roughness, which is used in the heat transfer coefficient computation.

2.1.2. Infinite swept wing solver

The airflow solver of NSCODE was extended to simulate infinite swept wings and validated in a previous paper [10]. The goal is to obtain the effect of the sweep angle by solving the third momentum equation instead of relying on a semi-empirical correction. The process to obtain a coherent solution with three velocity components on a planar grid follows. First, the simulated section must be normal to the wing leading edge and the 2D mesh is thus modified by applying a rotation around the vertical axis (z in this paper) which affects the geometry and the freestream boundary condition, as presented in Eqs. (1) and (2).

$$\begin{aligned} x' &= x\cos\Lambda_{geo} \\ y' &= y \end{aligned} \tag{1}$$

$$\begin{aligned} u'_{a,\infty} &= U_\infty \cos\alpha\cos\Lambda \\ v'_{a,\infty} &= U_\infty \cos\alpha\sin\Lambda \\ w'_{a,\infty} &= U_\infty \sin\alpha \end{aligned} \tag{2}$$

A noticeable effect is a reduction of the airfoil chord, which can also be considered as an increase of the thickness to chord ratio. Then, the infinite wing hypothesis is applied on the 3D RANS equations to cancel all derivatives relative to the spanwise direction (y'). This allows to recover the 2D RANS system with the addition of the so called crossflow equation, presented in Eq. (3), which solves the crossflow velocity component v'_a.

$$\begin{aligned} \frac{\partial\rho v'_a}{\partial t} + \frac{\partial\rho u'_a v'_a}{\partial x'} + \frac{\partial\rho w'_a v'_a}{\partial z'} &= \frac{\partial\tau_{x'y'}}{\partial x'} + \frac{\partial\tau_{y'z'}}{\partial z'} \\ &= \frac{\partial}{\partial x'}\left(\mu\frac{\partial v'_a}{\partial x'}\right) + \frac{\partial}{\partial z'}\left(\mu\frac{\partial v'_a}{\partial z'}\right) \end{aligned} \tag{3}$$

The crossflow velocity is thus non zero in the rotated simulation plane and some three-dimensional flow features appear, such as stagnation lines and separation lines. Oblique shocks can also be captured [10], although the test cases presented in this paper do not highlight this feature. In particular, the stagnation line has a strong effect on the convective heat transfer in the stagnation region as the velocity is non null for swept wings, as it is presented later in this paper. It should be noted that semi-empirical sweep correction methods for two-dimensional simulations cannot capture this effect [4], which tends to reduce the accuracy of such ice accretion simulations.

Although the current paper only addresses the 2.5D method applied to the simulation of an infinite swept wing, any combination of side wind and geometrical sweep can be represented by this approach. In such a case, Eq. (1) is only affected by the geometrical sweep Λ_{geo} while Eq. (2) also includes the effect of side wind ($\Lambda = \Lambda_{geo} + \Lambda_\beta$). The 2.5D method is also applicable to multi-element configurations (e.g. flap, main element and slat) as demonstrated by [13] who showed its impact on the lift curve slope including stall and post stall regions of a three elements airfoil.

For the numerical method, a loosely coupled approach was found to be sufficient and efficient to solve the non-conservative crossflow equation with a first-order accurate upwind spatial discretization scheme using the ADI solver of the Spalart–Allmaras one equation turbulence model [20]. Thus 2.5D simulations add at most 20% more computing time than the 2D formulation, as only one more equation is solved. Here, the first-order accurate spatial discretization is selected to match the methodology in [20], allowing the ADI solver from the turbulence model to be re-used. This approach is interesting for a rapid implementation of the 2.5D method in an already existing 2D code. An alternative approach is to implement a 2nd order advection scheme for the crossflow equation, which can be tailored on the main flow