Wing Aerodynamic Performance Analysis and Stall Prediction Using CANICE2D-NS Icing Code

K. Hasanzadeh 1, E. Laurendeau 2, F. Saeed 3, I. Paraschivoiu 4

124 Département de Génie Mécanique, École Polytechnique, C.P. 6079, Centre-Ville, Montréal (QC), H3C 3A7, Canada

3 Aerospace Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia

Email: Kazem.hasanzadeh@polymtl.ca

ABSTRACT

The paper presents results of iced airfoil performance in comparison to experimental data using a newly developed CFD based two-dimensional ice accretion and anti-icing simulation code, CANICE2D-NS. In CANICE2D-NS, a Serial Multi-Block (SMB) Navier-Stokes CFD code has been coupled with the CANICE2D panel/boundary-layer method based icing code. The new coupling is being used for the flow analysis and performance prediction of iced airfoils. Results reinforce the benefits and the robustness of CFD based CANICE2D-NS icing code in predicting ice shapes, aerodynamic performance parameters such as lift and drag coefficients as well as iced airfoil surface pressure coefficient distribution. The results show good agreement with experimental data both in terms of predicted ice shapes as well as aerodynamic analysis of predicted and experimental ice shapes.

1. Introduction

An iced airfoil, here referring to the ice generated on the leading edge of airfoil/wing, shows significant reduced lift and increased drag and consequently stalls at lower angle of attack compared to clean airfoils, which can result in accidents/incidents as previously reported [1].

There are mainly three ways to determine the icing effects on the aerodynamic performances, i.e. flight testing, wind tunnel testing and CFD simulations. These assessment methods differ in cost, simulation constraints for realistic test conditions, and accuracy (modeling errors). Flight tests are very expensive, but they are the best way to have the most realistic conditions and results, although freestream conditions are not easy to characterize. Wind-tunnel tests are less expensive albeit still out of reach of routine use, but they give additional flexibility to change and control tests conditions and icing parameters [2].

The cheapest method is therefore the application of CFD to model in-flight icing with a high flexibility to control the icing and flow conditions, but the accuracy of the approach depends on suitable turbulence models, mesh quality, and other aspects such as boundary layer transition. CFD modelling helps determine proper trends with respect to the various parameters. CFD flow simulations can predict the flow behaviour over simple ice shapes but lack sufficient accuracy when ice shapes are more complex, such as those with large ice horns and very rough surfaces. In these cases, there can be significant differences between the CFD predictions and the experimental results [3].

Grid generation methods, such as structured, unstructured, and hybrid grids, have been applied in icing problems, as described in the literature.
Structured-grid methods suffer in the aspect of flexibility and grid quality for complex shapes such as ice accretion, but the flow solver is typically more efficient and accurate. Grid generation for flow simulation on iced airfoils includes three basic steps: determination and smoothing of the iced surface; distribution of the grid points on the surface which could be modified based on its characteristic parameters such as curvature to help improve grid quality; and finally generation of the volume mesh using various approaches [4].

For icing CFD analyses, grid sensitivity studies can be helpful to find the optimal grid density for computation of the flow around the iced airfoil. The effects of grid quality and density can play a vital role when analyses involve cases of high angle of attack or of flow separation. Grid sensitivity is typically analyzed in the normal direction to the airfoil surface, stream-wise direction, minimum wall spacing (y+ value and expansion ratios), etc. Changes in the aerodynamic performance have been analyzed using grid sensitivity approaches and, as a consequence, optimal grid density and properties have been proposed to minimize the computation cost with respect to optimal simulation error [5].

Coupling the flow solution obtained using Navier-Stokes CFD methods to ice accretion codes, thereby replacing classical potential flow solvers, have been reported in the literature, such as the LEWICE-NS ice accretion code. Improved prediction of the final ice shape as a result of coupling the CFD flow solution to the particle trajectory and ice accretion thermodynamic modules has been reported in [6]. The main difficulty of the coupling for multi time-steps icing or multi-element icing is the complexity of grid generation for difficult geometries, automation of the meshing process and the deformation of the ice shape grids during the multiple time steps. Wall temperature boundary condition specification in the flow solver, which is related to the condition of warm glaze ice accretion, and ice roughness modeling and implementation in a CFD code, which mainly is related to the rough rime ice accretion, are the other aspects which need further investigations [6]. In order to gain better understanding of the underlying physics and support aircraft certification under icing conditions, the first step in the process is the development of a Navier-Stokes based icing simulation code. The investigation reported here is part of this larger effort.

In this paper, the general procedure in the new CFD-based ice accretion code, CANICE2D-NS, its validation, and analysis of aerodynamic performance degradation for several test cases are presented.

2. Methodology

The icing simulation code CANICE2D has been developed at École Polytechnique de Montréal as part of a collaborative R&D activity funded by Bombardier Aerospace and NSERC. It should be noted that the version presented here differs from the version used at Bombardier (named CANICE-BA) and the results and conclusions presented here do not apply to CANICE-BA. CANICE2D [7-13] contains an inviscid flow solver based (panel method) and icing and anti-icing resolution modules. The potential flow solution is used to determine the water droplet trajectory and droplet impingement distribution via the Lagrangian approach. An integral boundary layer formulation is implemented in CANICE to determine the local heat transfer coefficient, skin friction, and near-body flow characteristics. The modified Messenger model [14, 15] is used for ice accretion thermodynamic analysis. The thermodynamic model incorporates roughness, runback and water splash/ice shed models based on a water-bead model [16]. The ice shape and the amount of runback water are determined from the thermodynamic analysis.

The CANICE2D-NS code is a version of CANICE2D that has been configured to work with the Serial Multi-Block CFD code (SMB3D), a three dimensional Euler/Navier-Stokes flow solver developed by Badcock et al. [17,18]. SMB3D is a cell-centered finite-volume based code and can be applied for compressible steady and unsteady flow simulation. A wide range of turbulence models such as k-ω, k-ε, k-ω-SST, Spalart-Allmaras, hybrid (DES based on Spalart-Allmaras) have been implemented and validated. Parallel processing version of SMB3D (PMB3D) has been configured to decompose the structured multi-block grids solution on different processors using MPI and PVM. SMB3D and PMB3D have been used for a wide variety of aerospace applications [19-21]. The two codes can use the ICEM grid generation software for structured multi-block mesh generation.
CANICE2D-NS has been coupled with PMB in two modes. Mode 1 makes use of the PMB3D flow field solution, i.e. velocity, pressure, and density values, instead of the panel method solution in determining the particle trajectories, impingement efficiency, and boundary layer parameters. Mode 2, in addition to mode 1, also makes use of PMB3D based skin friction coefficients in the thermodynamic module. Note that the heat transfer coefficients are calculated within thermodynamic module through empirical methods. The effect of the two coupling methods is discussed in the next section.

CFD modelling for icing simulations is impeded by the absence of a robust grid generation process which creates high quality Navier-Stokes mesh for severe concave/convex glaze ice shapes. Automation of the grid generation is another equally important aspect for performing multi-time steps during icing simulations. An alternate approach is to perform grid deformation. However, in the present work, grid generation or re-generation rather than grid deformation was used for highly deformed glaze ice shapes.

In the present study, an automated grid generation process using the ICEM CFD mesh generator was therefore developed for use in conjunction with CANICE2D-NS. Elliptic smoothing equation using two back ground and fore ground control functions in a select manner is used to generate structured multi-block grids around ice shapes. The number of foreground smoothing iterations has been increased for the leading edge block containing the ice grids to obtain a better mesh quality. Grid parameters, wall distance, and number of elements are based on sensitivity analysis, computation cost vs. time consideration, and literature data. The one-equation Spalart-Allmaras turbulence model was selected for efficiency and accuracy in icing flow simulation as reported in the literature [5,6,22,24].

3. Results and Discussion

In this section, different icing simulation cases are presented with the objective to validate CANICE2D-NS by comparing its results with known experimental data [23]. All the selected test cases are of the glaze ice type, suitable for icing code verification. In addition they have been tested before and their experimental and simulation data have been cited in the literature which is essential for the work validation.

Case Study NACA0012 run 408

Figure 1 shows a multi time-steps glaze ice (NACA0012 Run 408, LEWICE validation study [23]) predicted by CANICE2D-NS mode 2 and compared to CANICE2D, LEWICE, and experimental data. The case study data is included in Table 1. The CANICE2D multi time step ice shape shows a smaller height prediction of the upper and lower ice horns. Increase in ice shape complexity can generate multiple stagnation points that create difficulties in boundary layer prediction, heat transfer coefficient calculation, and runback water estimation.

CANICE2D-NS 9 time-steps ice shape shows better prediction of the upper ice horn, but still underestimates the lower ice horn. This underestimation could come from the heat transfer coefficient prediction and roughness effects. The smoothed structured multi-block mesh generated for the ninth ice layer of case 408, with CANICE2D-NS mode 2, and using ICEM and its elliptic smoothing utility is shown in Figure 2. The mesh includes around 24,000 nodes and wall spacing of 2E-6 chord. SMB3D solver convergence data for 9 time-steps is shown in Figure 3. The flow solution at each time step has been used as the initial value for the next step, which reduces the convergence time. Lift and Drag coefficient degradation as a result of ice accretion at a constant angle of attack of 3.5° is shown in Figures 4 and 5, which show the decrease in lift and increase in drag as ice grows on the airfoil via the multi time-steps, in a quasi-steady manner.

Considering the effects of icing on performance, a close match between predicted and experimental ice shapes would mean that the predicted performance would be closer to that observed experimentally. Since CANICE2D-NS shows better prediction of ice shapes, the predicted performance under icing is also seen to be closer to the test data. This is clearly demonstrated in this study as shown later in this section.

Smoothed grid generated for the experimental ice shapes (case 408) using ICEM is shown in Figure 6. Aerodynamic performance analysis is
performed using a batch process in conjunction with the SMB3D flow solver to determine lift and drag coefficients as a function of the airfoil angle of attack. The Spallart-Almaras one-equation turbulence model is used in all the flow simulations herein. The lift curve and the drag polar for several ice shapes are shown in Figures 7 and 8 and compared to the clean airfoil. The CFD simulation of the clean airfoil shows a stall angle close to 13°. For the experimental ice shape CFD simulation the stall angle is close to 7°. CANICE2D ice shape CFD simulation stall angle is close to 9° and CANICE2D-NS ice shape stall angle is close to 8°. The lift and drag coefficient predictions as a function of the AOA for the CANICE2D-NS ice shapes are shown in Figure 7 and 8. Pressure coefficients comparison for the clean airfoil and different predicted ice shapes for angle of attack of 8° are shown in Figures 9 and 10. It can be observed from Figure 10, that the pressure distribution for the CANICE2D-NS ice shape is closer to the pressure distribution predicted for the experimental ice shape.

![Figure 1: Ice shape comparison for NACA0012 run 408](image1.png)

![Figure 2: ICEM mesh generated for 9th ice layer of CANICE2D-NS mode 2 (case 408)](image2.png)

![Figure 3: SMB convergence for 9 time-steps of CANICE2D-NS mode 2 (case 408)](image3.png)

![Figure 4: $C_L$ convergence for 9 time-steps of CANICE2D-NS mode 2 (case 408)](image4.png)

**Table 1: NACA0012 run 408 test conditions**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord Length</td>
<td>0.5334 m</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>3.5 deg</td>
</tr>
<tr>
<td>Flight Speed</td>
<td>102.8 m/s</td>
</tr>
<tr>
<td>Static Temperature</td>
<td>262.04 K</td>
</tr>
<tr>
<td>Mac Number</td>
<td>0.3169</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>4.44E6</td>
</tr>
<tr>
<td>LWC</td>
<td>0.86 g/m³</td>
</tr>
<tr>
<td>MVD</td>
<td>20 µm</td>
</tr>
<tr>
<td>Icing Duration</td>
<td>270 seconds</td>
</tr>
<tr>
<td>CANICE2D-NS</td>
<td>9 30-second steps</td>
</tr>
<tr>
<td>Icing Time-Steps</td>
<td></td>
</tr>
</tbody>
</table>
Figure 5: $C_D$ convergence for 9 time-steps of CANICE2D-NS mode 2 (case 408)

Figure 6: ICEM mesh generated for experimental ice shape (case 408)

Figure 7: $C_L$ comparison (case 408)

Figure 8: $C_D$ comparison (case 408)

Figure 9: $C_p$ comparison (AOA=8°) (case 408)

Figure 10: $C_p$ comparison, zoom view (AOA=8°) (case 408)
Case Study NACA0012 run 411

Figure 11 shows the comparison of CANICE2D-NS mode 2 multi time-step ice shapes with that from CANICE2D panel-method single time-step and LEWICE results, as well as experimental data case NACA0012 run 411 [23]. The test conditions are listed in Table 2.

Table 2: NACA0012 run 411 test conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord Length</td>
<td>0.5334 m</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>3.5 deg</td>
</tr>
<tr>
<td>Flight Speed</td>
<td>102.8 m/s</td>
</tr>
<tr>
<td>Static Temperature</td>
<td>262.04 K</td>
</tr>
<tr>
<td>Mac Number</td>
<td>0.3169</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>4.4E6</td>
</tr>
<tr>
<td>LWC</td>
<td>0.55 g/m³</td>
</tr>
<tr>
<td>MVD</td>
<td>20 µm</td>
</tr>
<tr>
<td>Icing Duration</td>
<td>14 Minutes</td>
</tr>
<tr>
<td>CANICE2D-NS</td>
<td>14 60-second steps</td>
</tr>
<tr>
<td>Icing Time-Steps</td>
<td></td>
</tr>
</tbody>
</table>

CANICE2D-NS ice shape was obtained after 14 60-second time steps of ice accretion simulation. The mesh generated after the 14th time step ice shape for this case 411 using CANICE2D-NS mode 2 is shown in Figure 12. The CANICE2D result for the same case was run in single time-step icing mode since the panel method breaks down due to increasing complexity of ice shape during the multi time-step simulation. As a consequence, single time-step CANICE2D run predicts excessive ice accretion thus emphasizing the importance of multi time-steps in ice accretion simulation process as the flow field parameters change with the ice growth. Ice shapes predicted by CANICE2D-NS were found to be closer to the experimental ice shapes as compared to CANICE2D. The structured grid has been generated using ICEM, and includes almost 24,000 nodes and a non-dimensional wall spacing of 2E-6 chord. The convergence data of the SMB3D code for 14th ice accretion time step is shown in Figure 13. Aerodynamic degradation from the effects of ice accretion at a constant angle of attack is presented in Figures 14 and 15.

The grid generated for case 411 experimental ice shapes is shown in Figure 16. The computed aerodynamic performance in terms of lift and drag coefficient as a function of angle of attack using the SMB3D solver for clean airfoil and predicted ice shapes are shown in Figures 17 and 18. Clean airfoil CFD stall angle is close to 13°. The CFD simulation of the experimental ice shapes shows a stall angle close to 7° and is marked by oscillations in the CFD flow solution convergence history. The stall angle simulation with CANICE2D-NS is close to 8°. Because the CANICE2D could not run multi time-step ice accretion for this case, its aerodynamic performance analyses results have not been included in Figure 17-20. Comparison of Cp for the angle of attack of 8° is presented in Figures 19 and 20. The results show that the pressure coefficient distribution predicted by CANICE2D-NS for numerical and experimental ice shape are in close agreement and, therefore, reinforce the benefits of using CFD for ice accretion simulations.
Figure 13: SMB convergence for 14 time-steps of CANICE2D-NS mode 2 (case 411)

Figure 14: CL convergence for 14 time-steps of CANICE2D-NS mode 2 (case 411)

Figure 15: CD convergence for 14 time-steps of CANICE2D-NS mode 2 (case 411)

Figure 16: ICEM mesh generated experimental ice shape (case 411)

Figure 17: CL comparison (case 411)

Figure 18: CD comparison (case 411)
**Case Study GLC305 run 944**

For this case, a comparison of CANICE2D-NS mode 2 ice shape with LEWICE and experimental ice shape [23] was carried out using larger time steps. The resulting ice shapes are shown in Figure 21. CANICE2D-NS ice shape was obtained after 5 time-steps for a total 22.5 minutes of ice accretion time. The number of time-steps were chosen to obtain the best results for the ice shape predictions. The case study test conditions are listed in Table 3. Mesh generated for the 5th ice-layer predicted by CANICE2D-NS is shown in Figure 22. The SMB3D convergence history for five steps is included in Figure 23. Lift and drag coefficient degradation due to ice growth for a constant angle of attack 3.5° is included in Figures 24 and 25.

The grid generated for the case 944 experimental ice shape is shown in Figure 26. The aerodynamic performance in terms of lift and drag computed by SMB3D for the clean airfoil and predicted ice shapes, as well as comparisons with experimental data, are shown in Figures 27 and 28 [3, 24]. For the clean airfoil analysis at Mach 0.12 and Reynolds number 3.5 million, stall angle is close to 12° and compares well with the experimental data. Both experimental ice shape CFD simulation and CANICE2D-NSiced airfoil results at these conditions, show a stall angle close to 8°, as convergence oscillation is observed, compared to the experimental ice data stall angle of 7°. A comparison of iced airfoil Cp distribution for an angle of attack of 6° is shown in Figures 29 and 30. As in the earlier test cases, the Cp distribution predicted by CANICE2D-NS is found to be close to the pressure coefficient distribution computed for the experimental ice shape.

![Figure 19: C_p comparison (AOA=8°) (case 411)](image)

![Figure 20: C_p comparison, zoom view (AOA=8°) (case 411)](image)

![Figure 21: Ice shape comparison for GLC305 run 944](image)

<table>
<thead>
<tr>
<th>Table 3: GLC305 run 944 test conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord Length</td>
</tr>
<tr>
<td>Angle of Attack</td>
</tr>
<tr>
<td>Flight Speed</td>
</tr>
<tr>
<td>Static Temperature</td>
</tr>
<tr>
<td>LWC</td>
</tr>
<tr>
<td>MVD</td>
</tr>
<tr>
<td>Icing Duration</td>
</tr>
<tr>
<td>CANICE2D-NS</td>
</tr>
</tbody>
</table>
Figure 22: ICEM mesh generated for 5th ice layer of CANICE2D-NS mode 2 (case 944)

Figure 23: SMB convergence for 5 time-steps of CANICE2D-NS mode 2 (case 944)

Figure 24: $C_L$ convergence for 5 time-steps of CANICE2D-NS mode 2 (case 944)

Figure 25: $C_D$ convergence for 5 time-steps of CANICE2D-NS mode 2 (case 944)

Figure 26: ICEM mesh generated experimental ice shape (case 944)

Figure 27: $C_L$ comparison (case 944)
Conclusion

In this paper, results of iced airfoil performance in comparison to experimental data using a newly developed CFD based two-dimensional ice accretion and anti-icing simulation code, CANICE2D-NS are presented. The CANICE2D-NS code is based on the panel method based icing code CANICE2D except that the flow solution is obtained via a Serial Multi-Block (SMB) Navier-Stokes CFD code. To facilitate multiple time step ice accretion simulation, an automated grid generation process using ICEM CFD mesh generator was developed and is used in conjunction with the CANICE2D-NS code to generate structured multi-block grids around ice shapes. To enhance grid quality, elliptic smoothing equations using two background and foreground control functions are applied within the grid generation process. It has been shown that the Navier-Stokes solver, as well as Navier-Stokes grid generation process is very robust for application of complex ice shapes. To further improve agreement between the predicted ice shapes and aerodynamic performance parameters, future direction will focus on analysis of roughness effects, runback water estimate, 3D grid generation techniques for iced geometries leading to a multiple time step 3D CFD based icing simulation code CANICE3D-NS.

Acknowledgements

The work has been performed through a Collaborative R&D Grant No. 341083–06 with Bombardier Aerospace and the Natural Sciences and Engineering Research Council of Canada (NSERC).

References


