Ice Shape

Fluid flow simulation that predicts ice formation on aircraft can help maintain safety, reduce test costs and decrease weight.

By Thomas Michon, Aerodynamics Engineer, and Didier Chartrain, former Aerodynamics and Performance Group Leader, DAHER-SOCATA, Tarbes, France

Buildup of ice during a flight alters aircraft aerodynamics by increasing drag and reducing the airfoil’s ability to provide lift. Ice accumulation also can affect an aircraft engine’s functional efficiency. These are important safety considerations, so protecting aircraft from ice accretion is a major requirement for manufacturers. European Certification Specifications and Federal Aviation Administration Regulations (CS23 and FAR23 for general aviation category) precisely define requirements in terms of ice protection. Requirements for ice shape must be determined for a 45-minute flight in icing conditions for unprotected areas and for a 22.5-minute flight for protected areas to take into account potential failure of the de-icing or anti-icing system[1].

The standard practice by which aircraft manufacturers meet these requirements is to present certifying authorities with flight test data confirming that aircraft performance and flight-handling qualities were not critically affected by ice accretion. To obtain such data, manufacturers traditionally use simulated ice shapes that are affixed to the wing (or other relevant parts of the aircraft).

Although the experimental method provides the most accurate results, it is too expensive to use in conducting design iterations to identify all modifications to the system that might be needed. Since most aircraft manufacturers must reduce weight, cost (both recurring and nonrecurring) and energy consumption to improve aircraft performance, many use numerical simulation to predict ice accretion and determine the most critical ice shapes as early as possible in the aircraft design process. This avoids the considerable time and costs required to carry out both a complete flight test program in known icing conditions and experimental tests in a climatic wind tunnel. Experimental tests can then be reserved mainly for validation.

Dedicated tools, such as 2-D LEWICE from NASA, traditionally have been used in this industry to predict droplet collection and ice accretion. With the continued push to increase aircraft efficiency, there is growing demand for new methods of predicting droplet collection efficiency and ice shapes on complex 3-D geometries. While this is a significant challenge, DAHER-SOCATA uses ANSYS Fluent CFD software to simulate/compute the local droplet collection efficiency and time-dependent ice shapes. When performed as early as possible in the aircraft development process, this practice can avoid costly changes for final certification.

Most aircraft manufacturers use numerical simulation to predict ice accretion and determine the most critical ice shapes as early as possible in the aircraft design process.
PREDICTING ICE ACCRETION

Ice accretion involves complex physics and phenomena, including aerodynamicics, multiphase flow and thermodynamic behavior, which are all time-dependent with geometry deformation. DAHER-SOCATA couples these physical phenomena using the moving deforming mesh (MDM) capability and the unsteady solver in ANSYS Fluent. This process accounts for droplet collection and thermodynamic balance as well as the shape change that occurs with accumulating ice.

FLOW SOLUTION

The flowfield resolution is addressed by the Fluent solver using state-of-the-art aerodynamic methodology. An accurate droplet trajectory prediction relies on a precise description of the boundary layer. In addition to applying the classical external aerodynamic model, the simulation must accurately describe the near-wall area using the appropriate prism layer to precisely reproduce local phenomena. A kω SST turbulence model is used, beginning with first order, then employing second order after hundreds of iterations, and finally employing third-order MUSCL spatial discretization. Appropriate use of the under-relaxation factor allows flow convergence in 600 iterations for a typical 3-D model.

DETERMINING DROPLET COLLECTION

The engineering team computes droplet collection efficiency using a user-defined function (UDF) developed by ANSYS and customized to DAHER-SOCATA’s specific needs. Fluent is a flexible software that has the capability to incorporate complex physics and enable customization. Strictly speaking, the local collection efficiency (β) is defined as the fraction of liquid water droplets that strike the aircraft relative to the number of droplets encountered along the flight path. In areas in which β is positive, ice is likely to appear. The UDF is based on an Eulerian description of the droplet concentration in the atmosphere, given as a volume fraction (α). DAHER-SOCATA engineers specify the droplet diameter as well as the droplet free-stream velocity. Since the droplets are considered a second fluid phase, specific continuity and momentum equations need to be solved to take into account the droplet drag as source term, which is provided by the UDF. Once both the droplet velocity from the above equation and the concentration are determined, the rate of liquid water impinging the surface is computed based on the local face normal.

Engineers validated this customized approach using ANSYS software for a series of 2-D cases by comparing the predicted results with those obtained from NASA’s LEWICE, which has been highly validated to experiment. Because there were no experimental data for this case, engineers considered the LEWICE results to be a suitable, highly respected reference, and, therefore, DAHER-SOCATA was confident to move from simple 2-D to more complex 3-D applications.

After 2-D validation, the droplet collection UDF is applied to different 3-D CFD applications. In the preliminary design phase, as soon as a 3-D wing definition is available, the team uses the UDF to identify the impinging area that requires protection by a de-icing system. Therefore, early in the design process, designers can provide an accurate specification of de-icing system requirements.

In another example involving a turbo-prop aircraft, the engine inlet must be protected from ice accretion to ensure sufficient engine performance in all icing conditions. The inlet lip is usually heated with a piccolo system, which uses hot air from the engine. Since this air offtake has a direct impact on engine performance, the system must be carefully sized for the inlet geometry. Because of the complex airflow around the inlet, classical ice accretion codes are not suitable for this type of analysis. Instead, DAHER-SOCATA used the droplet collection UDF to size the heated area of the inlet lip. The team performed analysis for different critical flight cases within the icing conditions envelope. The range of simulations performed allowed the team to assess the effect of aircraft angle of attack and airspeed on the impinged area.

THERMODYNAMIC BALANCE

To determine the thickness of the ice, the team computes the thermodynamic balance using a dedicated UDF in Fluent developed by DAHER-SOCATA. The ice surface is in thermal equilibrium among external convective heat transfer, conduction, mass transfer (from droplets) and phase changes.

Except for convective heat transfer, which is computed by the ANSYS CFD solver, thermal flux (conduction through ice, latent heat from transformation from liquid to solid, sublimation, and sensible heat, which accounts for droplet temperature change) was coded within the UDF. The theoretical definition used for the UDF was established after extensive literature research to find the most suitable model. The thermodynamic balance...
Aircraft icing is computed by the UDF, engineers can determine if the incoming water freezes or runs back, and from this information can compute a new ice thickness.

**Mesh Deformation**
As the thickness of the ice changes, the aerodynamic shape is modified and the aerodynamic flowfield is recomputed. Instead of creating a new mesh at each time step, the team morphs the existing mesh from the previous time step. Boundary nodes are moved normal to the wall, with a distance corresponding to ice thickness as computed using the thermodynamic model described above.

Since the final result strongly depends on the flowfield accuracy, mesh quality is a key parameter. The challenge is to maintain a suitably fine grid close to the surface that accounts for the complex ice accretion shape. The team currently combines both mesh morphing and smoothing as well as local remeshing. Because mesh modification is time-consuming, the remeshing frequency is a trade-off between computation time and accuracy. The mesh modifications were performed using the moving deforming mesh model in Fluent.

**Coupled Simulation Strategy**
Using the different tools described above, DAHER-SOCATA has established a complete unsteady simulation strategy:

- Every five seconds, the droplet collection is updated through a β computation.
- When a new flow solution is available (every second, in the present example), the computation of the thermodynamic balance provides a new ice thickness.
- Using the newly computed ice thickness, the mesh is morphed and smoothed every five seconds.

The above sequence is repeated throughout the full required simulation time. This chronology was developed specifically for DAHER-SOCATA’s TBM850 and its flight envelope. It may need to be adjusted if deployed on another application.
When the team compared an academic test case (NACA0012) using ANSYS Fluent to LEWICE, results showed very good agreement. Some local discrepancy appeared after 200 seconds of simulation, so improvement of the thermodynamic balance model will be performed to remedy this using LEWICE as well as experimental results.

In recent years, certification requirements regarding icing hazards have become more and more demanding, making aircraft icing certification more challenging. Using the droplet-collection UDF combined with the unsteady solver and MDM, ANSYS Fluent is a consistent global tool for ice accretion prediction.

**CONCLUSION**

This methodology to simulate ice accretion using Fluent tools — in final development before deployment at DAHER-SOCATA — is very promising. Comparisons with icing tunnel test data deliver increased confidence that this 3-D method is reliable for use in future aircraft development. DAHER-SOCATA is currently investigating how to optimize the TBM850 de-icing system. The company now has the capability to identify critical ice accretion zones early in the design process, which should lead to a significant development cost reduction by limiting the tests required for certification (both wind tunnel and flight) to a strict minimum. Using CFD in this way should minimize the size and the amount of the de-iced zone, leading directly to weight and cost reduction for this aircraft and eventually to aircraft performance improvement.

**Reference**


![Grid morphing for several time steps](image1)

![Simulation chronology developed specifically for DAHER-SOCATA’s TBM850](image2)

**Time(s)**

<table>
<thead>
<tr>
<th>TIME(s)</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow computation</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Beta computation</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Thermodynamic balance</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Morphing</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Remeshing</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
</tbody>
</table>

**Comparisons with icing tunnel test data deliver increased confidence in this 3-D method for reliable use in future aircraft development.**

![Ice formation prediction comparison using LEWICE and ANSYS Fluent](image3)