Icing Analysis of Fan Rotor at Part Load Conditions

K. Das, A. Hamed, D. Basu
Department of Aerospace Engineering and Engineering Mechanics
University of Cincinnati
Cincinnati, OH 45221-0070

ABSTRACT
Numerical simulations have been performed to investigate the supercooled water droplet trajectories and ice accretion in a fan rotor blade under part load conditions. An Eulerian-Lagrangian approach is used in formulating the flow and droplet governing equations in the rotating reference frame. A one-way interaction model is used to model the effects of momentum and energy exchange effects with the flow on the droplets as they travel through the rotor. Simulation results are presented for the flow field at 60%, 70%, 80%, 90%, and 100% design rotational speeds. Results showing the droplet trajectory paths and collection efficiency contours at part load conditions highlight the influence of engine speed and blade geometry. A heat transfer analysis shows the effect of speed on droplet temperature rise. Based on the impingement statistics and computed flow characteristics, a quasi-3D analysis of the ice accretion over the rotor blade is also conducted using the code LEWICE. Results are presented for the ice shape variation along the span and rotor speeds. It was found that the accreted ice mass and shape is highly dependent on rotor speed and local flow velocity.

INTRODUCTION
Techniques for predicting water impingement and ice accretion are needed to allow engine manufacturers and ice protection designers to continue to improve aircraft safety in icing conditions, to reduce design costs by minimizing the need for expensive icing ground and flight testing [1-3]. Therefore there has been a renewed interest in numerical simulations to address some of the certification requirements mandated by the regulatory bodies [4-9]. The icing of external stationary surfaces like airfoils, wings and fuselage has been an area of intense research and has led to the development and validation of computational methods for ice accretion prediction [10-13]. Some research have been conducted for rotating external surfaces like helicopter rotors [14]. Kind et al. [13] conducted a literature review of computational techniques in icing simulations. The most common approach is to solve the flowfield and droplet governing equations separately. The panel method is widely used to simulate the flowfield, sometimes coupled with a boundary layer analysis. The Lagrangian approach is used in the droplet trajectory simulations. Simulated results usually include the droplet collection efficiencies, impingement limits and ice shape. Da Silveria et al. [15] compared the predictions of various collection efficiency methods with available experimental data over a wing and a fuselage.

The supercooled water droplets suspended in clouds and moist atmosphere in the icy conditions are ingested by the engine intake that subsequently freezes to form ice. Ice accretion in aircraft engines raises safety and performance concerns such as mechanical damage from fan and spinner ice shedding and slow acceleration leading to compressor stall from ice accretion on turbofan splitters and fan and booster vanes. The compressible 3D unsteady turbomachinery flow and the effects of rotation on droplet trajectories present additional challenges in the simulation of ice accretion in aircraft engines. The quasi 3D or 2D methods traditionally employed in external surfaces are inadequate for ice accretion predictions in turbofan engines because of their high speed, 3D flow fields and complex blade geometries, and the complex shock structures present in modern turbofan and booster blade passages. In addition, the 3D turbomachinery flowfield exhibit large variation with engine operating points. This is especially important.
since ice accretion is of major concern at off design conditions such as idling where the flow field is considerably different from design speed.

Hamed et al. [16] developed a methodology for the simulation of supercooled droplet trajectories through aeroengine rotating machinery. It is based on an Eulerian-Lagrangian approach and one way modeling of the inter-phase interactions. The methodology was validated on rotor-67, which is a high-speed research fan with highly twisted blades. The results indicated that, in general, a higher proportion of the large droplets impinged the rotating blade pressure surface and that the impingement locations were within a smaller portion of the blade pressure surface compared to the smaller droplets. This resulted in higher local collection efficiency near the fan blade leading edge for the larger droplets. This is consistent with solid particle erosion predictions at the leading edge and over the rotor blade pressure surface, which is manifested in the blunting of the leading edge and pressure surface roughness [17-20]. One difference between solid particles trajectories and the supercooled water droplet trajectories is that the solid particles rebound after their surface impacts and cause additional damage but droplet trajectories terminate their path on contact with blade surface. Simulated results in rotor-67 at design speed indicated that the overall blade collection efficiency was higher near the leading edge and a greater proportion of large droplets impacted the rotating blade surface.

Unlike the external flow over wings, the flow temperature rise through the fan rotor can affect droplet temperature as they travel through the blade passage. The effect of energy exchange between the discrete and continuous phases was subsequently taken into consideration by Das et al. [21]. It did not affect the droplet trajectory path and blade collection efficiencies but caused the droplet temperature to rise through the rotor and the effect was greater for smaller droplets.

Experience with engine icing tests indicates that maximum ice accretion takes place at part-load conditions during engine idling. Das et al [22] conducted a preliminary investigation on supercooled droplet trajectories through the rotor passage at part load conditions on the GE-NASA energy efficient engine (E3) booster rotor [23,24]. Subsequently Das et al. [25] calculated the ice shape at part load conditions at different radial locations on the same blade geometry. The results indicated that the rotational speed has a significant effect on droplet impingement locations, and the peak collection efficiencies at the blade pressure surface leading edge increased with rotor speed reduction. The results also predicted thicker ice shapes at the leading edge of the blade and more pressure surface coverage towards the hub. Ice formation increases with engine speed reduction and is very sensitive to inlet temperature changes.

The present study deals with the supercooled liquid droplet trajectories and ice accretion on NASA Rotor-67 [26,27] blade under off-design operating conditions. Simulations are carried out at 60%, 70%, 80%, 90% and 100% design speed. The present investigation will highlight the effect of engine speed on droplet trajectory path thorough fan rotor passage and blade ice accretion. Results are shown for the computed flow field data, droplet trajectory path, blade water collection efficiency and accreted ice shape at different radial locations and for engine speeds. Predicted ice shape using this method [16,20] is also compared with full LEWICE simulation using panel flow solver.

**METHODOLOGY**

Both flow and particle governing equations are formulated in the rotating blade frame. A Eulerian-Lagrangian approach is employed for the flow field and droplet trajectories and one-way interaction model was used to simulate the effects of momentum and energy exchange on the droplet trajectories.

The droplet trajectories are determined from the numerical integration of the equations of motion in the blade rotating reference frame [22,25]:

\[
\frac{d^2 r_p}{d\tau^2} = F_r + r_p \left( \frac{d \theta_p}{dt} + \omega \right)^2 \tag{1}
\]

\[
r_p \frac{d^2 \theta_p}{dt^2} = F_\theta - 2 r_p \frac{dr_p}{dt} \left( \frac{d \theta_p}{dt} + \omega \right) \tag{2}
\]

\[
\frac{d^2 z_p}{dt^2} = F_z \tag{3}
\]

In the above equations, \( r_p, \theta_p \) and \( z_p \) define the particle location in cylindrical coordinates, and \( \omega \) the blade angular velocity. The last term in the RHS of the first two equations represent the centrifugal force and Coriolis acceleration. The first term on the RHS of equations 1-3 represents the components of the aerodynamic force of interaction between the two phases. The drag due to the relative velocity is considered as the primary aerodynamic force on the droplets since the forces due to gravity and buoyancy are negligible compared to the aerodynamic and centrifugal forces. Forces due to inter-droplet interactions and pressure gradient are also negligible for the small supercooled water droplets. The aerodynamic force of interaction is expressed in terms of the drag coefficient and the droplet slip velocity as follows [22]:

Copyright © 2006 by ASME
\[ F = \frac{3}{4} \frac{C_D}{d_p} \left| \nabla \cdot \left( \nabla - \nabla \right) \right| \]  

(4)

Where \( d_p \) is the particle diameter and \( \nabla \) and \( \nabla_p \) are the gas and the droplet velocity vectors. The drag coefficient \( C_D \) is computed from empirical correlations involving the Reynolds number based on the relative velocity between the droplet and the gas [17-20].

The change in droplet temperatures along their trajectory is determined from the energy exchange rate between the droplets and the airflow. Only the heat transfer by convection is taken into consideration since no phase change takes place at supercooled droplet temperatures.

\[ \frac{dT}{dt} = \frac{3}{16} \frac{h}{\rho_p C_p} \frac{1}{d_p} (T_g - T_p) \]  

(6)

where, \( C_p \) is the specific heat of the supercooled droplet and \( \rho_p \) is the droplet density. The heat transfer coefficient \( h \) is evaluated using Ranz and Marshall’s [28] correlation for spherical droplets:

\[ h = \frac{k}{d_p} \left( 2.0 + 0.6 \times 10^{12} \rho \frac{1}{Pr^{1/3}} \right) \]  

(7)

Where \( k \) is the thermal conductivity of the gas, \( Pr \) is the Prandtl number.

**COMPUTATIONAL DETAILS**

The numerical solution for the three-dimensional compressible Reynolds Averaged Navier Stokes (RANS) equations in the rotating frame is obtained using ADPAC [29]. The code utilizes finite volume, Runge-Kutta time marching scheme to solve the time dependent form of 3-D RANS equations. Convective fluxes are approximated using a second order central difference scheme stabilized with scalar artificial dissipation. Local time stepping, implicit residual smoothing, scaled dissipation function, and a multigrid technique are employed to accelerate convergence [30-32]. Several turbulence models including the algebraic Baldwin-Lomax model, one equation Spalart-Allmaras model and two-equation k-R model are available in the solver. Rotor-67 flowfield simulations by Yuan et al. [33] indicated that the predicted aerodynamic performance as well as the 3-D shock structure within the rotor passage were comparable for both the low Reynolds number k-ε and the Baldwin-Lomax algebraic turbulence models over a wide range of operating conditions. Therefore, the Baldwin-Lomax algebraic turbulence model was used in the present investigation.

The solution domain for the both 3D flow and droplet trajectory simulations extended 80\% chord upstream and 45\% chord downstream of the rotor blade row. Figure 1 shows the computational grid consisting of 145 × 65 × 65 grid points in the streamwise, blade-to-blade and the hub-to-tip directions. The computational grid selection was based on prior experience [21] with the Rotor-67 simulations, in which three different grid resolutions were used in a grid independence study. The grid was stretched in the near wall regions using a hyperbolic tangent function. The minimum grid spacing in the wall normal direction was 1.0 × 10^{-4} times the chord with 15 grid points within the boundary layer. The total temperature, total pressure and absolute flow angles were specified at the booster inlet. Adiabatic wall and no slip boundary conditions were prescribed on the stationery and rotating surfaces. The exit static pressure was specified at the hub, and the radial pressure distribution from the integration of the axisymmetric radial momentum equations.

Equations 1-3 and 6 are solved using a four-stage Runge-Kutta time integration starting from the droplet inlet conditions upstream of the blades. The droplet distance from the blade passage is monitored throughout the droplet trajectory until impingement on a surfaces or exit from the blade passage. The outlined procedure was implemented in the code TURBODROP, a parallel solver based on the MPI message passing technique. In the parallelization strategy, a number of droplets are assigned to each computer node and the integration proceeds independent of other droplets. At the end of the solutions the computer nodes report the results to the master processor. Distributed computing was found to reduce the computational time by two orders of magnitude for the 50,000 droplet trajectories considered at each operating point in the current investigation.

The trajectory simulations were carried out for 30 \( \mu \)m droplets that were assumed to be spherical with uniform loading across the inlet. The relative humidity was considered to be 100 \% with liquid water content of 0.01.

**BLADE SURFACE COLLECTION EFFICIENCY**

The limiting droplet trajectories have been commonly used for the definition of collection efficiency for external flows [11-15] by initiating a number of droplet trajectories that are released over incremental distance across the free stream direction until they reach the surface outermost boundaries. The slopes of the limiting trajectories relative to the free stream direction are used to define collection efficiency for ice-growth calculations. The collection efficiency is typically less than one, which is the...
upper limit had the droplets not been deflected by the flow around the surface. The concept of limiting droplet trajectories in the particle frame of reference is not suitable for turbomachinery flow due to blade rotation and because neighboring droplets could continue their trajectories in different blade passages of subsequent blade rows. Therefore, a flux based collection efficiency was defined by Hamed et al. [16] as the ratio of the local droplet mass flux at the blade surface to its value at the turbomachinery inlet station. This definition, which is equally applicable to stationary and rotating blades, is utilized in the present investigation.

ICE SHAPE CALCULATION

Utilizing LEWICE icing physics, a quasi-3D approach was adopted using the computed collection efficiency at a number of radial blade sections. In this approach the 2D potential flow, droplet trajectory modules, and heat transfer calculation procedures of LEWICE version 3.0 [34]. The convective heat transfer coefficients, pressure coefficients and the droplet collection efficiency distributions at a number of rotating blade sections are extracted to use as input for the ice accretion prediction. A pitchwise averaged flow meridional inlet velocity and droplet inlet temperatures were extracted from the 3D flowfield and droplet trajectory data at each radial location for use in the 2D LEWICE code.

RESULTS AND DISCUSSIONS

Numerical simulations were conducted for the flow field and droplet trajectories through the NASA Rotor-67 fan. The detail design data for the engine, whose cross section is shown in Fig. 1, were reported by Strazisar et al. [26-27]. Rotor-67 is the first stage rotor of a two-stage fan, with a design pressure ratio of 1.63 and a mass flow rate of 33.25 kg/sec. The tip radius varies from 25.7 cm at the leading edge to 24.25 cm at the trailing edge, and the hub/tip ratio varies from 0.375 to 0.478. It has 22 blades with 1.56 aspect ratio. Its design 16043 RPM corresponds to a rotor tip speed of 429 m/sec and a relative inflow Mach number of 1.38 at the tip. The off-design operating conditions for the fan rotor were obtained from the experimental data of Strazisar et al. [27].

The computed static pressure ratio behind the rotor blade is shown in figure 3. One can see that the computed profile is in close agreement with the experimental results.

Figure 3 shows the computed relative Mach number contours on a representative blade-to-blade surface at three different conditions corresponding to 60%, 70%, 80%, 90% rotor design speed. The contours indicate that the rotational speed have a significant effect on the flow in the rotor reference frame. It can be seen that the flow is mainly subsonic at the low rotor speed. On the other hand, the flow has significant supersonic region, a shock wave that originates from the suction side at the high rotor speed. A small transonic zone is visible on the suction surface near the leading edge at intermediate rotor speed with formation of shock structure at 90% rotor speed.

Figure 4 shows the Mach number contours at 80% design speed at three radial locations. It can be observed from the figure that the flow is subsonic near the hub and transonic near the tip. Hence there could significant radial variation in the flow field at part load conditions similar to that observed in design condition [21,22].

The trajectory simulations were carried out for 50,000 droplets injected across the inlet for a uniform loading. The inlet temperature for the 30-micron droplets was 233 K, compared to flow temperature of 270 K. All the droplets are assumed to stick to the blade after they impinge on its surface. Droplets are injected with zero absolute velocity, which in the rotor frame of reference is equal to rotor speed towards the blade pressure surface.

Figure 5 and 6 shows the droplet trajectories associated velocity vectors at 30% and 70% span from tip for 60%, 70%, 80% and 90% design speeds respectively. One can see that the droplets enter the blade passage with a positive incidence angle in the rotor reference frame. This is a result of their lower absolute velocities compared to the flow. However for the velocity vector diagrams in Figs 5 and 6 it is clear that the angle of incidence does not change significantly with rotor speed. With increase in rotor speed, both gas and droplet axial velocity increases proportionally effectively maintaining a constant angle of incidence. Consequently, the impingement statistics and exit mass flow rate will exhibit little sensitivity to rotor speed. This is confirmed by the droplet exit mass flow rate variation in Figure 7, which shows little variation with rotor speed.

Figure 8 shows the blade pressure surface collection efficiency at 60%, 70%, 80%, 90% and 100% design speed. It is clear from the figure that the overall impingement pattern and water collection rate do not change with rotor speed. However at 90% and 100% design speed, relatively higher accumulation can be noticed in the pressure surface leading edge. The insensitivity of predicted droplet trajectory path and impingement location to rotor speed does not agree with corresponding results obtained for GE-NASA E^3 booster rotor in a similar study [22]. The E^3 rotor exhibited significant variation in water collection rate due to changing rotor speed. This can
be attributed to the fact that the flow conditions are different for R-67 and E3 booster rotors. The E3 booster has much lower design speed ($\omega=390$ rad/sec) compared to that of rotor-67 ($\omega=1634$ rad/sec). This significantly impacts the flow field and aerodynamic drag on the droplet resulting in different response to rotor speed variation.

Figures 9 and 10 show the predicted ice shapes at different rotor speeds on the blade. It can be seen that ice is predicted over the pressure surface and the leading edge of the suction surface. It is clear that ice formation is very sensitive to rotor speed. Ice accretion on the blade surface could be observed at lower rotor speed of 60% but gradually decreases as the rotor speed increases and then it is not predicted beyond 90% speed. At the 60% speed ice accretion is highest near the hub and decreases radially over 75% of the rotor span. No accretion is predicted at the tip due to very high gas velocity.

Figure 11 and 12 show the computed ice shapes at four blade sections located at 10%, 40%, 70% and 90% span from the hub. As can be seen from the figures that ice formation is maximum for the blade pressure surface and comparatively low on the suction surface. Thicker ice shapes are predicted at the lower rotor speeds, and the mass of accreted ice tends to decrease as one moves from hub to tip. To illustrate this point further, the total iced area and maximum thickness along the span is shown in Figures 13 and 14 for 60% and 70% design speed. It is clear that the iced area and maximum thickness exhibits almost linear variation in the spanwise direction with maximum value near the hub. The previous ice accretion studies [22,25] on E3 booster rotor showed similar trend. This can be attributed to the fact that flow velocity is lowest near the hub and increases towards the tip resulting in higher level of aerodynamic heating.

CONCLUSIONS

A numerical investigation was conducted to investigate the effect of rotor speed on droplet trajectories and ice accretion on a fan rotor blade. The simulations are carried out in the NASA rotor-67 at 60%, 70%, 80%, 90% and 100% of the design speed. The inter-phase coupling between the air and droplet are modeled through both momentum and energy exchange equations in relative frame of reference.

Results are presented for the computed relative Mach number contours for different rotor speeds and at different radial locations. The static pressure ratio at the blade exit is compared with experimental data at design condition. The results presented for droplet trajectory path at different speeds highlight the effect of rotation on the droplet impingement angle and location. Computed pressure surface collection efficiency contours indicate that rotational speed has no significant effect on droplet trajectories impingement locations. The predicted ice accretion on the blades indicates thicker ice shapes at the leading edge of the blade and more pressure surface coverage towards the hub. It also shows that ice accumulation decreases with increasing rotational speed.

ACKNOWLEDGEMENT

This research was sponsored by Ohio Aerospace Institute grant CCRP 2003-05. The authors would like to thank Dr. K. Venkataramani of GE Aircraft Engines, Cincinnati for many helpful discussions and Dr A. Strazisar of NASA Glenn Research Center for providing the experimental data of R-67 at off-design conditions.

REFERENCE

27. Straziser, A., Private Communications.
34. Lewice 3.0 Users Manual; NASA Glenn research center.
Figure 1 Computational Grid

Figure 2 Static Pressure Profiles at exit:
Comparison with experimental data

60% design speed

70% design speed

80% design speed

90% design speed

Fig. 3 Effect of rotor speed on Mach number at mid-span
Fig. 4 Mach contours at different radial location at 80% of design speed.

Fig. 5 Droplet trajectories and velocity vectors through blade passage at different speed near 70% span from tip. Vector scale 1:70.
Fig. 6 Droplet trajectories and velocity vectors through blade passage at different speed near 30% span from tip. Vector scale 1:140

Fig. 7. Effect of rotor speed on droplet mass flow at rotor exit
Figure 9. Effect of rotor speed on ice accretion at 90% span from tip.
Figure 10. Effect of rotor speed on ice accretion at 70% span from tip.

Figure 11. Spanwise variation of ice accretion at 60% design speed

Figure 12. Spanwise variation of ice accretion at 70% design speed
Figure 13 Iced area distributions along the span for 60% and 70% design speed

Figure 14 Maximum ice thicknesses along the span for 60% and 70% design speed