DROPLET TRAJECTORIES AND COLLECTION ON FAN ROTOR AT OFF-DESIGN CONDITIONS

Kaushik Das  
Department of Aerospace Engineering  
University of Cincinnati  
Cincinnati, OH

Awatif A. Hamed  
Department of Aerospace Engineering  
University of Cincinnati  
Cincinnati, OH

Debashis Basu  
Department of Aerospace Engineering  
University of Cincinnati  
Cincinnati, OH

ABSTRACT

A numerical investigation is conducted to study water droplet collection on a high bypass turbofan engine booster rotor under different rotational speeds. 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. Results are presented for the computed flow field, droplet trajectories, water collection efficiency and droplet exit mass and temperature profiles at 60%, 70%, 80%, 90%, and 100% design rotational speeds. The highest collection is predicted near the tip at the pressure surfaces leading edge. It then drops considerably at 40% - 60% of the booster rotor blade chord and remains low over the aft portion of the pressure side. In general, the rotor blade collection efficiency is strongly influenced by rotor speed and increases as the rotational speed decreases.

INTRODUCTION

There is a renewed interest in numerical simulation of icing phenomena to address some of the certification requirements mandated by regulatory bodies [1-6]. This is motivated both by the high cost of engine testing to meet stringent ice-protection regulations as well as the constraints imposed by the limited time windows during inclement weather. Most computational techniques for ice accretion prediction and ice protection system design have been developed for low speed external flow over airfoils, wings and fuselage [7-10] and a few on helicopter blades [11]. The general approach has been to solve the flowfield and droplet governing equations separately and the Lagrangian approach is used in the droplet trajectory simulations. According to the literature review by Kind et al. [10], the panel method is commonly used in the analysis of the external flow and is sometimes coupled with a boundary layer analysis. The droplet trajectories under the influence of aerodynamic forces are used to predict impingement statistics on the external surface, and a thermodynamic analysis to study the water that freezes to ice. Computed parameters include droplet collection rate, droplet impingement limit and rime ice shape [9-14]. These quasi 2-D methods are inadequate for ice accretion predictions in turbofan engines because of their high speed, 3-D flow fields and complex blade geometries, and the complex shock structures present in modern turbofan and booster blade passages. In addition, the 3-D turbomachinery flowfield exhibit large variation with engine operating points because of the associated changes in rotor speed. This is especially important since ice accretion is of major concern at off design conditions such as idling where the 3D flow field is considerably different from design speed.

Ice accretion in modern aircraft engines raises safety and performance concerns regarding mechanical damage from fan and spinner ice shedding. In addition, ice accretion on turbofan splitters and fan and booster vanes could slow engine acceleration and lead to compressor stall. The simulation of ice accretion in aircraft engines is challenging prospect because of the complex 3D unsteady turbomachinery flow and because droplet trajectories are strongly influenced by the blade rotation. Hamed et al. [16] developed a methodology for the simulation of supercooled droplet trajectories through Aeroengine rotating machinery. The methodology adopts an Eulerian-Lagrangian approach and uses a one-way momentum exchange model for inter-phase interaction. The method was used to study the effect of droplet size on the blade surface collection efficiency in the Rotor-67 research fan [16]. The authors also defined flux-based droplet collection efficiency suitable for internal flow rotating machinery. 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, 18]. 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. Subsequently Das et al. [19] considered the effect of energy exchange between the discrete and continuous phases on the temperature of supercooled droplets. Droplet size was found to affect not only the droplet collection efficiency but also the final droplet temperature at impingement on blade surface. This in turn can influence the type of ice formed on the blade surface.

Experience with engine icing tests indicates that maximum ice accretion takes place at part-load conditions encountered during the idling phase of the engine. Hence the focus of the present investigation is on the behavior of droplet through the rotor passage at part load conditions. Simulations are carried out for the GE-NASA energy efficient engine (E³) [20,21] booster stage rotor to characterize the droplet impingement pattern and water collection rate on the blades at 60%, 70%, 80%, 90% and 100% design speed. The Eulerian Lagrangian approach is adopted for the continuous and discrete phases and the governing equations are solved in the rotating reference frame [16, 19]. Results are presented for the computed turbomachinery flow field, and the circumferential averaged flow variables at rotor exit for 100% speed are compared with design data. Computed results for the rotor blade water collection rate are presented at the fan off design conditions to show the variations in the blade surface collection with rotor speed. The results show that the rotor speed affects not only the collection efficiency but also the droplet temperatures as they travel through the rotor passage.

**METHODOLOGY**

The flow and particle governing equations are formulated in the rotating blade reference frame based on a Eulerian-Lagrangian approach. One-way interaction models are used to simulate the effects of momentum exchange and energy exchange between the flow and droplets through aerodynamic forces and through convection on the droplet trajectories, and the droplet temperatures. The 3-D flow field is obtained from the numerical solution of the compressible Reynolds Averaged Navier Stokes (RANS) equations using ADPAC [21]. The code utilizes a finite volume Runge-Kutta time marching scheme to solve the time-dependent form of 3-D RANS equations. The Runge-Kutta scheme has been used in many 3-D viscous flow analyses of low-pressure compression system studies because of CPU economy and the ease of implementing convergence acceleration techniques [23, 24]. 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. Several turbulence models, including the algebraic Baldwin-Lomax model, the one equation Spalart-Allmaras model, and the two-equation k-R model are available in the solver. Yuan et al. [25] presented computational results that indicated that the predicted aerodynamic performances as well as the 3-D shock structure within a transonic fan rotor were comparable for both the low Reynolds number k-ε and the Baldwin-Lomax algebraic turbulence models over a wide range of operating conditions. Accordingly, the Baldwin-Lomax algebraic turbulence model is employed in the current investigation of the booster rotor flow field at considerable savings in CPU resources.

**MOMENTUM AND ENERGY EXCHANGE**

Droplet trajectories are determined from the numerical integration of their equations of motion while their temperatures are computed from their energy exchange equation with the airflow in the rotating blade reference frame [16]. The droplet equations of motion are:

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

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

\[
\frac{d^2 z_p}{d\tau^2} = F_z
\]

In the above equations, \(r_p\), \(\theta_p\) and \(z_p\) are the particle location in cylindrical polar coordinates and \(\omega\) is the blade angular velocity. The last term in the RHS of the first two equations represents 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 slip velocity relative to the flow is considered as the primary aerodynamic force on the droplets since the forces due to gravity, buoyancy, pressure gradient, and due to inter-droplet interactions are negligible relative to the aerodynamic and centrifugal forces. The aerodynamic force of interaction per unit mass of droplet is expressed in terms of the drag coefficient \(C_D\) and the droplet slip velocity between the gas velocity \(V\) and the water droplet velocity \(V_p\) as follows [16]:

\[
\bar{F} = \frac{3C_d}{4d_p} \left| \nabla - \nabla_p \right| \left( \nabla - \nabla_p \right)
\]

where \(d_p\) is the droplet 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 (Re) between the droplet and the gas.
the simulations require alternating between rotating and stationary reference frames. Furthermore, since neighboring droplets can continue their trajectories through different blade passages in subsequent blade rows, a different definition of collection efficiency is required in turbomachines. 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 is better suited for internal flow applications and is equally applicable to rotating and stationary blades in multistage turbomachines.

**NUMERICS**

The outlined methodology was used in the numerical simulations of droplet trajectories through the GE-NASA energy efficient engine (E³), for which detailed design data were reported by Sullivan et al. [20]. The off-design operating conditions for the booster rotor were obtained from two performance maps for the core and bypass stream of the low-pressure compression system [21]. Profiles of total temperature, total pressure and absolute flow angles were specified at the booster inlet from the design data. Adiabatic wall and no slip boundary conditions were prescribed relative to stationary and rotating surfaces. The exit static pressure was specified at the hub, and the radial pressure distribution was determined from the integration of the axisymmetric radial momentum equation. Periodic boundary conditions were employed across the solution domain that extended 10% chord upstream and 15% chord downstream of the rotor blade row. The computational grid selection was based on prior experience [19] with the Rotor-67 simulations, in which a grid independent study was conducted using three different grid resolutions. The selected grid consists of 95 × 81 × 65 points in the stream-wise, blade-to-blade and the hub-to-tip directions. Grids were clustered near the walls using a hyperbolic tangent function with a minimum grid spacing in the wall normal direction of 1.0 × 10⁻⁴ times the chord with 15 grid points within the boundary layer.

**RESULTS AND DISCUSSIONS**

The simulations were conducted for the GE-NASA energy efficient engine (E³), a high bypass turbofan that uses a quarter stage boosters to provide the required core supercharging. Results are presented on the booster stage. The booster has 56 blades, a hub-tip ratio of 0.782 and an aspect ratio of 2.12 at inlet. The design total temperature and pressure ratio across the rotor are 1.1776 and 1.683 respectively at the corrected RPM of 3727 and mass flow rate of 143.74 kg/sec. The flowfield and droplet trajectory computations were performed at five operating conditions corresponding to 60%, 70%, 80%, 90% and 100% design speed.

Figure 1 shows the computed relative Mach number contours on a representative blade-to-blade surface at three different conditions corresponding to 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. One can see 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 bubble is seen at the suction surface near the leading edge at intermediate rotor speed but no shock structure is visible. In general, the shock structure in the outer blade region is complex and changes along the blade span as can be seen in figure 2.

Figure 3 compares the radial profiles of the design flow variables to the circumferential average of the rotor exit computational results at 100% rotor speed. It can be observed that the computed total temperature, exit relative and absolute Mach number and absolute flow exit angle are in close agreement with the design data. On the other hand, the computed absolute Mach number is higher than the design value near the tip region. No attempt was made to change the specified static pressure at the hub in order to get closer to the design exit Mach number.

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. Figure 4 presents representative droplet trajectories through the rotor at different rotational speeds. 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. One can see that at lower speeds the droplets traverse across several blade passages before they enter one and impinge the rotor blade pressure side near the leading edge. As the rotor speed increases, they enter the passage sooner and their impingement locations are spread over the pressure surface beyond the leading edge. At higher speeds, some of the droplets manage to cross the passage without impinging the blades.

The computed change in droplet temperature across the rotor is presented in figure 5 for the different rotational speeds. The results follow the specific droplets that eventually pass trough the rotor passage and exit the stage, although majority of the droplets impinge on the blade. One can see that at any axial location, the droplets temperature rise is greater at lower rotor speeds. This can be explained in terms of the longer time needed to reach the location at lower rotor speeds. The effect of rotor speed on the residence time of the droplets that go across the rotor as shown in figure 6. One can see that the characteristic time for the droplet that exit the rotor ranges between 6.6 milliseconds 100% rotor speed to 6.6 milliseconds at 70% rotor speed, compared to 0.3 milliseconds and 1.0 milliseconds for the airflow. The increased residence time with rotor speed reduction can also be deduced from the trajectories in figure 4.

The corresponding radial variations in the circumferential averaged droplet mass flux and droplet temperature at the rotor exit are presented in figures 7 and 8. According to figure 7, none of the droplets leave the rotor blade at 60% design speed. On the other hand, the portion of droplets that leave the rotor without impinging the blade passage increases with rotor speed. Figure 8 shows that the exit droplet temperatures increase as the rotor speed decreases because of the reduced residence time.

Figure 9 presents results for the computed water collection efficiency over the rotor blade pressure surface at various rotor speeds. The results indicate that in general, the collection efficiency is higher at the leading edge near the tip and that the maximum collection efficiency is predicted at the lowest rotor speed. Only very few droplets impinge the leading edge of the rotor blade suction surface. This is consistent with prior predictions in Rotor - 67 by Hamed et al [16] that indicated higher collection efficiency at the pressure surface leading edge for larger droplets. It is important to observe however that the rotor-67 research compressor highly loaded blades [16] had considerable twist and higher chord and consequently the pressure surface collection efficiency pattern exhibited larger radial variations than in the current investigation.

Figures 10 and 11 demonstrate the radial variation in droplet collection efficiency with rotor speed both for the local maximum at the pressure surface leading edge and for the chord averaged value. These figures clearly demonstrate that overall, the rotor collection efficiency increases with reduced rotor speed. The difference is greater for the maximum local value at the leading edge. It is clear from the predicted collection efficiency patterns that ice will mostly be accumulated at low speed near the leading edge. This is consistent with the observations of leading edge glaze ice horns formation in engine tests at idling conditions [27].

**CONCLUSION**

A numerical investigation was conducted to investigate the effect of rotor speed on droplet trajectories through a turbofan booster rotor. The simulations are carried out in the GE-NASA energy efficient engine (E³) booster rotor 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. The computed relative Mach number contour are presented for different rotor speeds and at different radial locations. The computed exit angles, total pressure ratio, relative and absolute Mach numbers at 100% rotational speed are presented and compared with design values. The presented results for the droplet trajectories indicate that rotational speed has a significant effect on droplet trajectories impingement locations, and blade surface collection efficiency. High
collection efficiencies are predicted at the blade pressure surface leading edge near the tip and the peak value increase with rotor speed reduction. Also the droplet temperatures increase with the reduction in rotor speed due to the increase in their residence time. The presented results lead to the conclusion that rotor speed has significant influence on droplet impingement, ice accretion location and the associated ice accretion type.

ACKNOWLEDGMENTS
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 useful discussions and Mr. Robert Ogden of UC for his technical support.

REFERENCES
Fig. 1 Relative Mach number contours at different rotor speeds

70% Design Speed

Mid-Span

80% Design Speed

10% From Hub

Fig. 2 Relative Mach number contours at 80% design speed

100% Design Speed

10% span from tip
Figure 3. Comparison of circumferentially averaged computational results with design values at rotor exit.
Figure 4. Droplet Trajectories at different rotor speeds
Figure 5. Effect of rotor speed on droplet temperature

Figure 6. Effect of rotor speed on droplet residence time

Figure 7. Circumferentially averaged droplet mass flux

Figure 8. Circumferential averaged droplet temperature at rotor exit
70% of design speed

60% of design speed

80% of design speed

90% of design speed

Figure 9. Collection Efficiency on blade suction surface.

Figure 10. Leading edge collection efficiency

Figure 11. Chord averaged collection efficiency