JOURNAL OF AIRCRAFT
Vol. 34, No. 6, November–December 1997

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Novel Nacelle Thermal Anti-Icing Exhaust Grill for Enhanced Mixing

Mark A. Gillan*
The Queen's University of Belfast, Belfast BT9 5AG, Northern Ireland, United Kingdom
and
Roy Farren†
Short Brothers plc, Belfast BT3 9DZ, Northern Ireland, United Kingdom

Introduction

TURBO-FAN aircraft flying at subsonic speeds through clouds below an altitude of approximately 22,000 ft would experience ice accretion on the nose cowls of their engine nacelles if the surfaces were not anti-iced. Ice accretion on the leading edge of nacelle nose cowls would considerably restrict the airflow through the engine, thereby incurring a performance loss. Furthermore, damage may result from ice breaking away from the cowl and being ingested into the engine or by ice hitting the acoustic liner.

Anti-icing is commonly performed using a piccolo tube arrangement whereby hot air, which is bled off the high pressure compressor, is ducted through a pressure-regulating valve via a series of holes to the nose cowl. This hot air is subsequently vented overboard through a thermal anti-icing (TAI) exhaust grill (Fig. 1).

Traditionally, a TAI exhaust grill consists of a set of angled vanes that tend to encourage reattachment of the hot anti-icing exhaust air onto the nacelle's fan cowl door (FCD) surface. With the relatively recent introduction of composite FCDs, problems have arisen in-flight, whereby the hot air, which has been vented through the TAI exhaust grill, has reattached on the composite FCD surface, thereby blistering the paint.

The purpose of this investigation is to enhance the mixing characteristics of an in-service TAI exhaust grill to reduce the excessive temperatures occurring on the composite FCD outer skin.

Numerical Method

Numerical calculations are performed using Fluent Inc.'s three-dimensional Navier–Stokes code RAMPANT Parallel.[1] To accurately model the highly complex flowfield, the renormalization group (RNG) k-ε turbulence model with swirl modification is employed. Within this model, the dissipation term in the turbulent kinetic energy transport equation has been modified to account for the effects of compressibility. Moreover, the turbulent dissipation rate transport equation contains a rate of strain term that is important for the treatment of nonequilibrium effects and flows in the rapid distortion limit, such as separated or stagnation flows.

The nacelle skin and TAI geometry are imported from CATIA, in IGES format, to Fluent Inc.'s mesh generating package GeoMesh.[2] The computational domain is discretized by 171 × 81 × 81 grid points. With over 1.1 million grid points, the block structured mesh is split into four separate partitions and the entire problem run in parallel on four SGI Indy workstations. The spatially second-order accurate solution is found to be grid independent for grids finer than 80 × 38 × 36. This is in good agreement with previous numerical jets in crossflow studies.[3,4] A simple two-level sawtooth multigrid scheme is employed with 171 × 81 × 81 representing the finest grid. The first cell spacing normal to the wall is set at $y^+ < 5$. The CFL number is increased to 3.5 with the introduction of implicit residual smoothing. Finally, local time stepping further enhances convergence to a steady-state solution.

The initial boundary-layer profile specified at the pressure inlet boundary, which is located fore of the TAI access panel, is obtained from Queen's University full mass-averaged Navier–Stokes code MGENS.[5] The TAI hot exhaust air may be considered to be originating from an infinite source; therefore, the exhaust boundary is set using RAMPANT's mass flow boundary conditions. Converged solutions (rms residual of each conserved variable $< 1 \times 10^{-3}$) are obtained after approximately 800 multigrid iterations.

Numerical Results

RAMPANT was initially validated for the in-service 60-deg angled-vane TAI exhaust grill design using the worst-case flight-test point data for the specific aircraft under investigation, namely, Mach number $M_\infty = 0.548$, static temperature T_∞ = ISA + 16°C at an altitude of 12,500 ft during maximum power climb.

The numerically generated results along the $y = 0$ plane/nacelle surface intersection (Table 1) are in excellent agreement with the thermocouple flight test data obtained from the specific aircraft under investigation (±4°C), indicating RAMPANT's ability to accurately simulate the highly complex three-dimensional flowfields, including the intricate vortex structures. RAMPANT computes a maximum FCD temperature of 211°C at the FCD/TAI access panel junction, which is 81°C higher than that which can be constantly sustained by the composite FCD skin without blistering the paint. Hence, to alleviate the paint blistering problem, a new TAI exhaust grill, with substantially enhanced mixing characteristics, had to be designed. The optimization procedure (for enhanced mixing) was subject to two major constraints:

1) The open area of the TAI exhaust grill must remain fixed at 15 in.^{2}

2) The maximum temperature allowed on the composite FCD skin was 148°C (for a transient case) and 130°C (for a constant case).

While a diverse range of simple TAI exhaust grill redesigns were analyzed, the introduction of a central blended bar was

Presented as Paper 96-2463 at the AIAA 14th Applied Aerodynamics Conference, New Orleans, LA, June 17–20, 1996; received March 18, 1997; revision received July 28, 1997; accepted for publication July 28, 1997.

*Lecturer, Department of Aeronautical Engineering. Member AIAA.

†Chief Technical and Systems Engineer, Aerospace Engineering; currently Engineer, Propulsion Systems, Boeing Commercial Airplane Group, P.O. Box 3707, M/S 05-60, Seattle, WA.

Table 1 Numerical validation against flight-test thermocouple data

	Maximum FCD temperature (0 in. aft of junction)	Thermocouple no. 1 (2 in. aft of junction)	Thermocouple no. 2 (6 in. aft of junction)	Thermocouple no. 3 (10 in. aft of junction)	Thermocouple no. 4 (14 in. aft of junction)
Flight test	Not recorded	195.4°C	166°C	154°C	138°C
RAMPANT	211°C	194°C	168°C	150°C	135°C

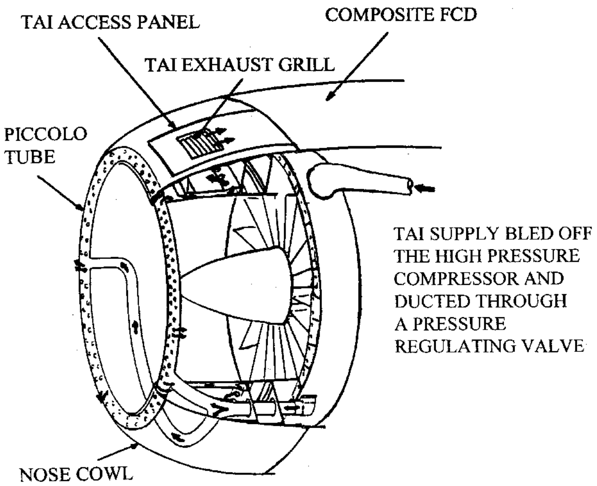

Fig. 1 Schematic of a nacelle TAI system.

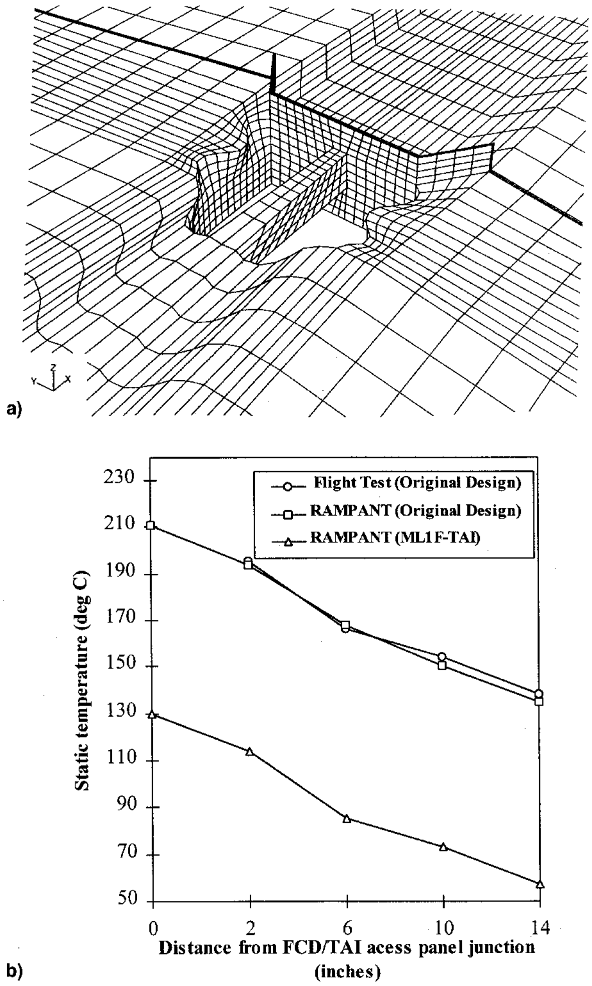

Fig. 2 Mixing lobe TAI with three-quarter-in.-high faired fence (ML1F–TAI design): a) structured surface mesh and b) static temperature plot.

found to be the most effective, in that it splits the hot exhaust flow in half, thereby allowing a cool central flow of freestream air to travel between them, thus enhancing the mixing process. To maintain the open area of 15 in.,^{2} a forward-to-aft taper was introduced into the blended bar. The central blended tapered bar, or church-window TAI exhaust grill (CW–TAI), effectively reduces the maximum FCD temperature by 32°C, from 211 to 179°C.

In an attempt to further reduce the maximum FCD temperature, the addition of a judiciously placed fence directly aft of the nonducted church-window exhaust was deemed necessary. A three-quarter-in.-high fence, straddling the full width of the access panel (CWFSF–TAI), was analyzed. RAMPANT indicates that the maximum FCD temperature is reduced to 138°C. However, the drag penalty incurred by this design is prohibitively punitive.

A three-quarter-in.-faired fence, with only a 1-in. straddle either side of the exhaust (CW1F–TAI), was then analyzed in an attempt to reduce the drag penalty to a palatable level. The numerical analysis indicates that the maximum FCD temperature is reduced to 150°C, while incurring approximately only half the drag penalty of the CWFSF–TAI configuration.

Finally, employing a novel design commonly utilized by acoustic engineers,[6] a mixing lobe TAI exhaust grill (ML1F–TAI) with a central tapered bar was analyzed in conjunction with a three-quarter-in.-high faired fence with 1-in. straddle (Fig. 2a). The superior mixing characteristics of the TAI mixing lobe design are clearly depicted in Fig. 2b. The ML1F–TAI configuration reduces the maximum FCD temperature to 130°C, thereby satisfying both the transient and constant composite FCD temperature limits of 147 and 130°C, respectively. The incurred drag penalty is comparable to that of the CW1F–TAI design.

Conclusions

A Navier–Stokes investigation has been performed in an attempt to enhance the mixing characteristics of an in-service TAI exhaust grill. This optimization procedure is conducted under the constraint of maintaining a constant TAI exhaust grill open area. Code correlation with flight-test data for the current TAI exhaust grill design, during the maximum power climb phase of the flight regime, is excellent. A novel mixing-lobe TAI exhaust grill, with a blended central tapered bar, is shown to exhibit superior mixing characteristics. However, to reduce the maximum FCD temperature to its 130°C tolerance limit, thus preventing paint blistering, a three-quarter-in.-high faired fence is required in conjunction with the mixing-lobe TAI exhaust grill design.

References

[1]*RAMPANT User's Guide, Version 3.1,* Fluent Inc., Lebanon, NH, 1994.

[2]*GeoMesh User's Guide-Release 3,* Fluent Inc., Lebanon, NH, 1996.

[3]Kim, S.-W., "Calculations of Separated 3-D Flows with a Pressure-Staggered Navier-Stokes Equation Solver," NASA CR-187065, 1991.

[4]Kim, S.-W., and Benson, T. J., "Fluid Flow of a Row of Jets in Crossflow—A Numerical Study," *AIAA Journal,* Vol. 31, No. 5, 1993, pp. 806–811.

[5]Gillan, M. A., "Computational Analysis of Buffet Alleviation in Viscous Transonic Flow over a Porous Airfoil," *AIAA Journal,* Vol. 33, No. 4, 1995, pp. 769–772.

[6]McCormack, D. C., and Bennett, J.C., Jr., "Vortical and Turbulent Structure of a Lobed Mixer Free Shear Layer," *AIAA Journal,* Vol. 32, No. 9, 1994, pp. 1852–1859.

Field Measurements of Helicopter Rotor Wash in Hover and Forward Flight

Milton E. Teske* and Andrew E. Kaufman†
Continuum Dynamics, Inc.,
Princeton, New Jersey 08543-3073
and
Charles W. George‡ and Gregg M. Johnson§
USDA Forest Service, Missoula, Montana 59807-8089

Introduction

THE U.S. Department of Agriculture Forest Service uses helicopters to spray forests with pesticides, spread water or retardants on forest fires, and, along with the U.S. Army, maintains an interest in the propagation of fire along the ground and the movement of contaminants in the air. The dispersing aircraft generally fly low over their target area in an effort to deliver their payloads with as much precision as possible. In this flight configuration, helicopters induce downwash and sidewash velocities that may be significant when weighing the advantages of one helicopter vs another with regard to potential of sideways spread of the released spray material, the injurious possibility of the induced velocities actually enhancing the ability of a fire to spread, or the ability to disperse dusts, aerosols, or other contaminants from a specified location. In the case of forest fire-fighting, this hypothesis suggests that helicopters may create a significant induced sidewash surface velocity, which actually propels the fire sideways, moving it through the fuel and preventing control.

Up to now, the effect of a passing helicopter on the local meteorology, particularly near a forest fire, has been understood only from a qualitative standpoint. Because the quantitative data are considered important in understanding fire propagation and the dispersal of pesticides and other contaminants, a cooperative field study was recently conducted to collect such data, with the hope that these data may be used to validate models of helicopter wakes, suggest operational methods for fighting forest fires, and provide insight into the dispersal of dusts, aerosols, and other contaminants in the atmosphere.

Discussion

During July 26 to July 29 and Sept. 27 to Oct. 1, 1994, 181 passes were made by seven helicopters (Bell 205H, Bell 206B, Blackhawk, Boeing Vertol BV-107, Chinook CH-47, Sikorsky S-61, and Skycrane) over an instrumented tower grid at Yuba City, California. Propeller anemometers measured the induced downwash and sidewash velocities generated by the helicopters in hover or forward flight above the tower grid, in a fashion similar to the technique used to collect data to infer the decay behavior of aircraft vortices near the ground.[1] Resulting time histories from the anemometers were examined to recover the magnitude and behavior of the induced velocities. The complete data set, including descriptive data about the helicopters, is summarized in Table 1.

Collected data were reduced and presented previously in Teske and Kaufman[2] and Teske et al.[3] These data show an easily identifiable sidewash velocity obeying a simple exponential decay law. The four parameters developed from these data were 1) the maximum induced surface velocity, 2) its exponential decay time constant, 3) the apparent depth of the layer containing the sidewash motion, and 4) its frontal speed outward from the helicopter. Data average results suggest that a significant induced surface sidewash velocity may result from the passage of a helicopter (depending on the size, weight, and flight speed of the helicopter, and its height above the ground), that the peak-induced sidewash velocity moves slowly along the ground in a tall gust front, and that the induced sidewash velocity remains important for a long period of time. While these results are not surprising to anyone who has been in the neighborhood of a helicopter or observed its effects in a theatrical movie, they are quantified by the field study.

The most important result is the estimate of maximum induced surface velocity, as this result may be used to infer preferential flight conditions for a helicopter in a forest fire-fighting situation. To generalize this result from all field tests, we have applied a regression algorithm to the collected data that develops the least-squares best fit to an equation of the following form

$$V = aS^bH^c \tag{1}$$

where V is the maximum induced surface velocity, S is the helicopter ground speed, and H is the helicopter drop height. In this equation, H is normalized by the helicopter rotor radius R, and V and S are normalized by the induced downwash velocity w, taken from actuator disk theory

$$w = \frac{1}{R}\left[\frac{W}{2\pi\rho}\right]^{1/2} \tag{2}$$

where W is the helicopter weight and ρ is air density. For all helicopters examined in this study, the regression algorithm recovers

$$a = 1.635, \qquad b = -0.442, \qquad c = -0.792$$

with a correlation of $R^2 = 0.677$. Figure 1 displays the results and indicates the level of anticipated induced surface velocity, given a value for helicopter ground speed and drop height. For example, for the Blackhawk, if a maximum induced surface velocity of 4.32 m/s is allowed in a specific situation (V = 0.4), with a drop height of 32.0 m (H = 4.0), the helicopter should be traveling at a minimum ground speed of 21.6 m/s (S = 2.0).

Alternatively, Fig. 1 may be used to estimate induced surface velocity. For example, a Blackhawk traveling at a ground

Table 1 Field study summary

Helicopter	R, m	w, m/s	Flybys
Bell 205H	7.4	8.85	30
Bell 206B	5.5	6.12	19
Blackhawk	8.0	10.80	36
Boeing Vertol BV-107	7.6	14.79	29
Chinook CH-47	9.1	12.88	24
Sikorsky S-61	9.4	11.17	16
Skycrane	11.0	13.14	27
Total	——	——	181

Received March 16, 1997; revision received June 20, 1997; accepted for publication June 20, 1997.

*Senior Associate, P.O. Box 3073. Member AIAA.
†Associate, P.O. Box 3073.
‡Project Leader and Senior Scientist, Intermountain Fire Sciences Laboratory, P.O. Box 8089.
§Physical Scientist, Intermountain Fire Sciences Laboratory, P.O. Box 8089.