NUMERICAL INVESTIGATION OF THE EFFECTS OF ICING ON FIXED AND ROTARY WING AIRCRAFT

LEWIS GRANT
IN-03-CR
109005
P.22

Progress Report for the Period

January 1, 1992 - June 30, 1992

Attn: Dr. Mark Potapczuk
Mail Stop 77-10
Icing and Cryogenic Technology Branch
NASA Lewis Research Center
Cleveland, OH 44135

Prepared by

L. N. Sankar
Professor, School of Aerospace Engineering
Georgia Institute of Technology
Atlanta, GA 30332

July 1992

(NASA-CR-190542) NUMERICAL INVESTIGATION OF THE EFFECTS OF ICING ON FIXED AND ROTARY WING AIRCRAFT Semiannual Progress Report, 1 Jan. - 30 Jun. 1992 (Georgia Inst. of Tech.) 22 p

N92-34105

Unclas

G3/03 0109005

INTRODUCTION

During the past six years, under the support of NASA Lewis Research Center, a research project dealing with the effects of icing on fixed and rotary wing aircraft performance has been underway at Georgia Tech. The performance predictions are based on the numerical solution of the 2-D and 3-D Navier-Stokes equations, with a suitable model for turbulence.

The first phase of this work dealt with the effect of icing on the lift, drag and stall characteristics of 2-D airfoils. A NACA 0012 airfoil profile modified by a simulated ice shape, used in the experimental work of Prof. M. Bragg, was studied. Reasonable agreement with measured surface pressure data was observed. The numerical solutions predicted that extensive separation and stall occurred at extremely low angles of attack (8 degrees). The solution, and the extent of the separated flow over the upper surface were found to be sensitive to the transition location, and turbulence model. Three turbulence models, one based on the Baldwin-Lomax model, the second based on Johnson-King non-equilibrium model, and the third based on the k-e turbulence model were coded and tested. It was concluded that the higher order models do not significantly improve the correlation with experiments.

The second phase of this work dealt with the extension of the 2-D Navier-Stokes analysis to three-dimensions. A 3-D unsteady compressible Navier-Stokes solver, capable of handling arbitrary tapered, swept wing geometries was developed. The computer code was calibrated using experimental data for a rectangular wing, and a swept wing tested by Prof. Bragg at Ohio State University, and later at the University of Illinois. Both clean wings, and wings with a simulated leading edge ice shape were numerically studied. Again, a reasonably good correlation with experiments was observed. The numerical results revealed several interesting phenomena: the effect of separated flow at the wing root splitter plate on the inboard stall characteristics, helical particle paths inside the leading edge separation bubble etc. In many instances, the numerical calculations and experiments guided one another, leading to an increased understanding of the flow field [Ref. 1,2].

The third phase of this work dealt with the effects of icing on rotary wing aircraft performance. A 3-D Navier-Stokes code developed by the principal investigator under U. S. Army Research Office support was modified to handle rotary wings with simulated leading edge ice shapes. It was numerically demonstrated that the effect of icing causes loss in thrust, and increase in torque for hover and forward flight operating conditions [Ref. 3].

PROGRESS MADE DURING THE REPORTING PERIOD

The proposed research consists of three tasks: Study of High Lift Systems, Continued Correlation of Fixed Iced-Wing studies with Prof. Bragg's experimental data, and modification of the fixed wing analysis to general wing-body configurations. Progress made under these three tasks are discussed in detail below.

Task 1 : Effects of Icing on 2-D High-Lift Systems

A 2-D multi-element airfoil code previously developed at Georgia Tech has been modified to study the effects of icing on the aerodynamic characteristics high-lift systems. This computer code divides the flow field into a number of zones as shown in figure 1. In each of these zones the solver numerically integrates the 2-D compressible Navier-Stokes equations, using a time-marching scheme. Any number of elements (airfoil-flap, airfoil-slat, airfoil-slat-flap etc.) may be present. At block interface boundary, the flow properties are required to be continuous. At solid surfaces, no slip conditions are imposed, and the normal derivatives of temperature and density are set to zero. At the far field boundaries, the flow is either assumed to be undisturbed, or is modified by the addition of a lifting vortex solution.

This solver can account for effects of icing in one of two ways. When the ice build up is small, it is equivalent to an increase in the surface roughness. This influences the transition location, the levels of turbulence, the thickness of the boundary layer and stall. These effects associated with light ice build-up have been modeled in the present work through appropriate changes in the turbulence model routine, and transition criterion routine, using an approach similar to that of Prof. Cebeci and his coworkers, in their interactive boundary layer methods. When the ice shape is large, the airfoil shape is changed, and substantial changes in the flow field result. For these situations, the flow solver and the grid generator will accept the ice shape, and generate the grid and the flow field over the combined airfoil-simulated ice shape. Of course, ice formation over the main element, and slat can both be modeled.

Some sample applications of this 2-D solver are briefly discussed here, and in the abstract of a paper that has been submitted to the forthcoming AIAA Meeting in Reno (1993). The abstract is included in the appendix.

Figure 1 shows the body-fitted grid generated over a GAW 130 airfoil/flap combination, which has been extensively tested at Wichita State by Wentz and Seetharam (NASA CR 2443, 1974). Figure 2 shows the surface pressure distribution over this configuration, for a flap setting of 25 degrees, and an angle of attack equal to 5 degrees, at a freestream Mach number equal to 0.3. Finally, figure 3 shows the C_l vs. α curve for a range of a where the flow over the airfoil and the flap remains fully attached. Calculations at higher angles of attack are now underway.

We have performed a series of calculations to determine the effects of small scale ice build up on the high lift characteristics of this configuration. The abstract in the appendix summarizes the progress made to date.

There is a considerable body of additional experimental data for clean multi-element high lift configurations. there is also a limited amount of experimental data for a four-element iced-airfoil configuration, given in Ref. 4. The computer code will be validated using the data in Ref. 4 during the next six moths..

It is anticipated that at the conclusion of this study, a multi-element iced airfoil analysis will have been developed, suitable for analysis and design of high lift systems designed to operate under adverse weather conditions.

Task 2: Continued Correlation of the Iced Wing Code

During the past three years, the 3-D iced wing analysis work reported previously has benefited from cooperation and joint studies with Prof. Bragg at U. of Illinois. These joint studies have brought out flow features that would not have been noticed from numerical or experimental studies alone. During the past reporting period, we worked closely with Prof. Bragg as new measured LDV data in the separated region behind the leading edge ice shape become available.

It was noted by Prof. Bragg and his co-workers that the Georgia Tech 3-D flow solver does a reasonable job of predicting surface pressure distributions and spanwise lift variation, but that the correlation between the computed and measured velocities in the separated region is poor.

This discrepancy may be due to a number of reasons. First of all, a rather coarse grid in the normal direction (with 41 to 45 points) is used in the present work, in order to keep the CPU time requirements to a minimum. Such a coarse grid may not adequately resolve the thin shear layer separating off the ice shape. Secondly, a standard Baldwin-Lomax model is used in the work. Modifications to the model, similar to that done by Potapczuk et al. at NASA Lewis Research center may be needed in the separated region. Finally, the numerical viscosity terms, whose coefficients are chosen to ensure global convergence of the flow field (lift, pressures), may be too large or excessive in the shear layer region, smearing the shear layer.

During the reporting period, we have begun to systematically evaluate these issues. First, a version of the iced wing analysis which uses the well known Roe scheme has been developed. The Roe scheme is spatially third order accurate. It automatically scales the magnitude of numerical dissipation based on local flow speed, and local speed of sound. In particular, shear waves and entropy waves are dissipated very little because their dissipation is proportional to local flow speed. Finally, the Roe scheme can capture shocks, if any, across two or three mesh widths.

We have validated the Roe scheme through calculations of viscous flow over a standard AGARD configuration, an F-5 wing at 0.9 Mach number, and Prof. Bragg's clean wing at 4 degree angle of attack. The surface pressures predicted by the Roe solver are quite comparable to the original central difference based solver.

Additional results from the Roe solver will be presented at the forthcoming NASA Lewis Workshop on Icing, to be held in August 1992.

<u>Task 3: Extension of the Wing-Alone Analysis to Wing-Body Configurations:</u>

Work in this area during the reporting period has just begun. We have already developed a 3-D elliptic grid generator that can handle a number of wing and wing-body shapes. A version of this grid generator was used in the past (under other sponsored support) to generate grids around fighter aircraft, and wing-body-canard configurations.

We have modified the 3-D iced wing analysis to accept such externally generated grids, and to accept multi-block grids. During the next research period, we plan to choose a configurations in consultation with the sponsor. We anticipate that both the iced wing-body and 3-D iced-high-lift versions of the code will be developed and demonstrated.

REFERENCES

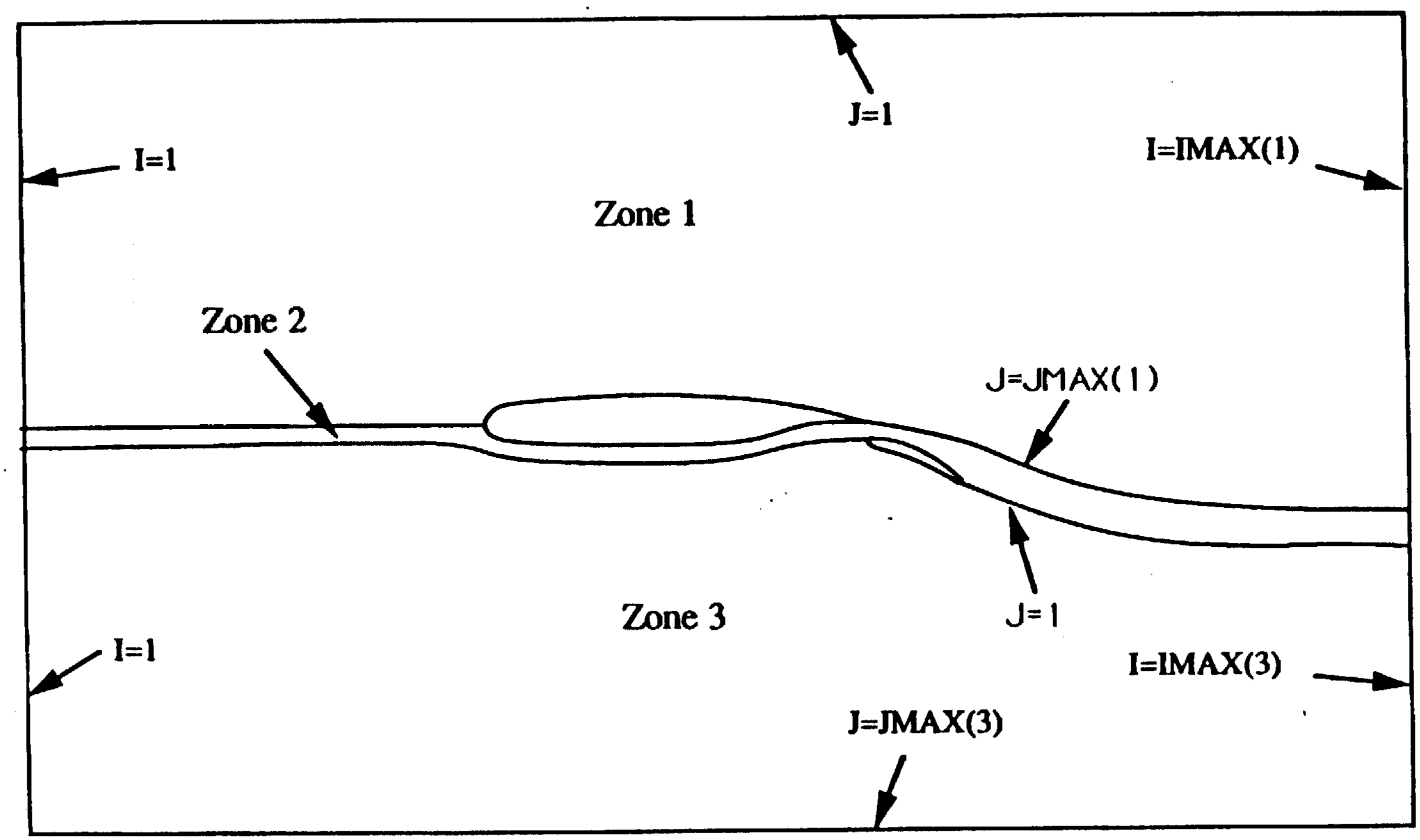
1. Bragg, M. B. and Khodadoust, A., "Effect of Simulated Glaze Ice on a Rectangular Wing," AIAA Paper 89-0750.

2. Kwon, O. J. and Sankar, L. N., "Numerical Studies of the Effects of Icing on Finite Wing Aerodynamics," AIAA Paper 90-0757, To appear in the Journal of Aircraft, 1991.

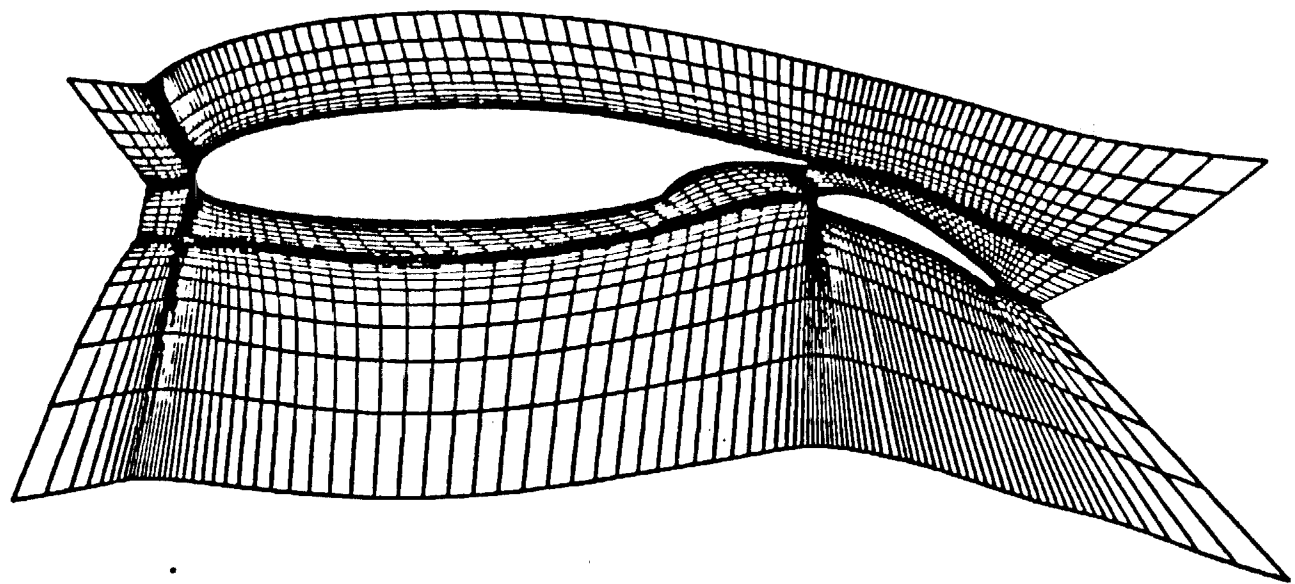
3. Sankar, L. N. and Kwon, O. J., "Numerical Simulation of the Effects of Icing on Fixed and Rotary Wing Aircraft Aerodynamics," NASA Lewis Aircraft Performance-in-Icing Workshop, August 1991.

4. Lynch, F. T., Valarezo, W. O. and McGhee, R. J., "The Adverse Aerodynamic Impact of Very Small Leading-Edge Ice (Roughness) Build-Up on Wings and Tails," Proceedings of the 68th AGARD Fluid Dynamics Panel Specialists Meeting on Effects of Adverse Weather on Aerodynamics, May 1991.

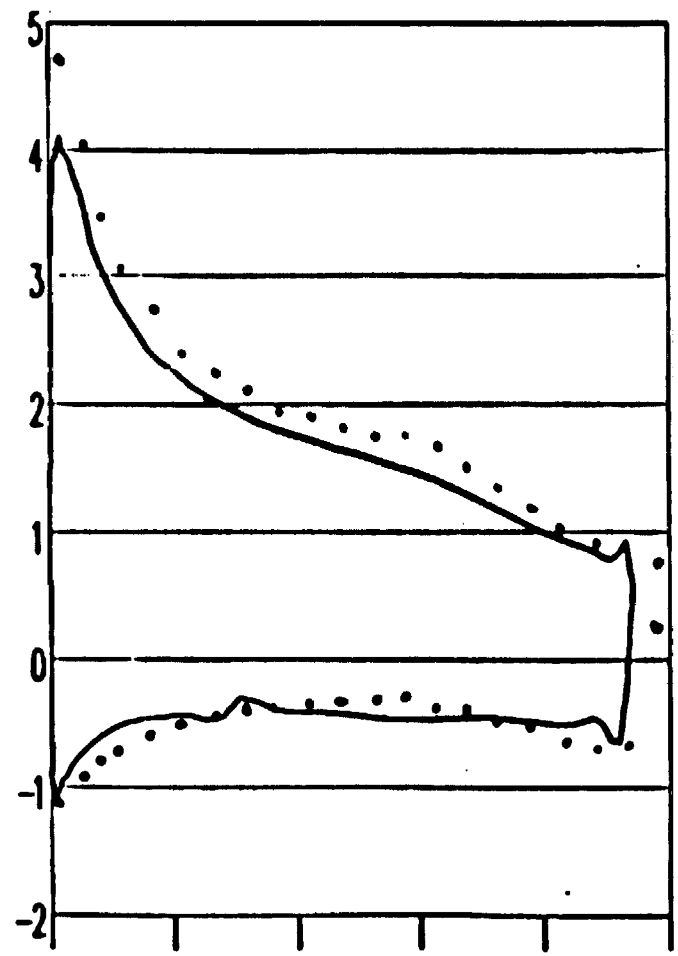
Grid Topology for a Two-Element Airfoil



Close-up view of Computational Grid for GA(W)-1 Airfoil/Flap Combination

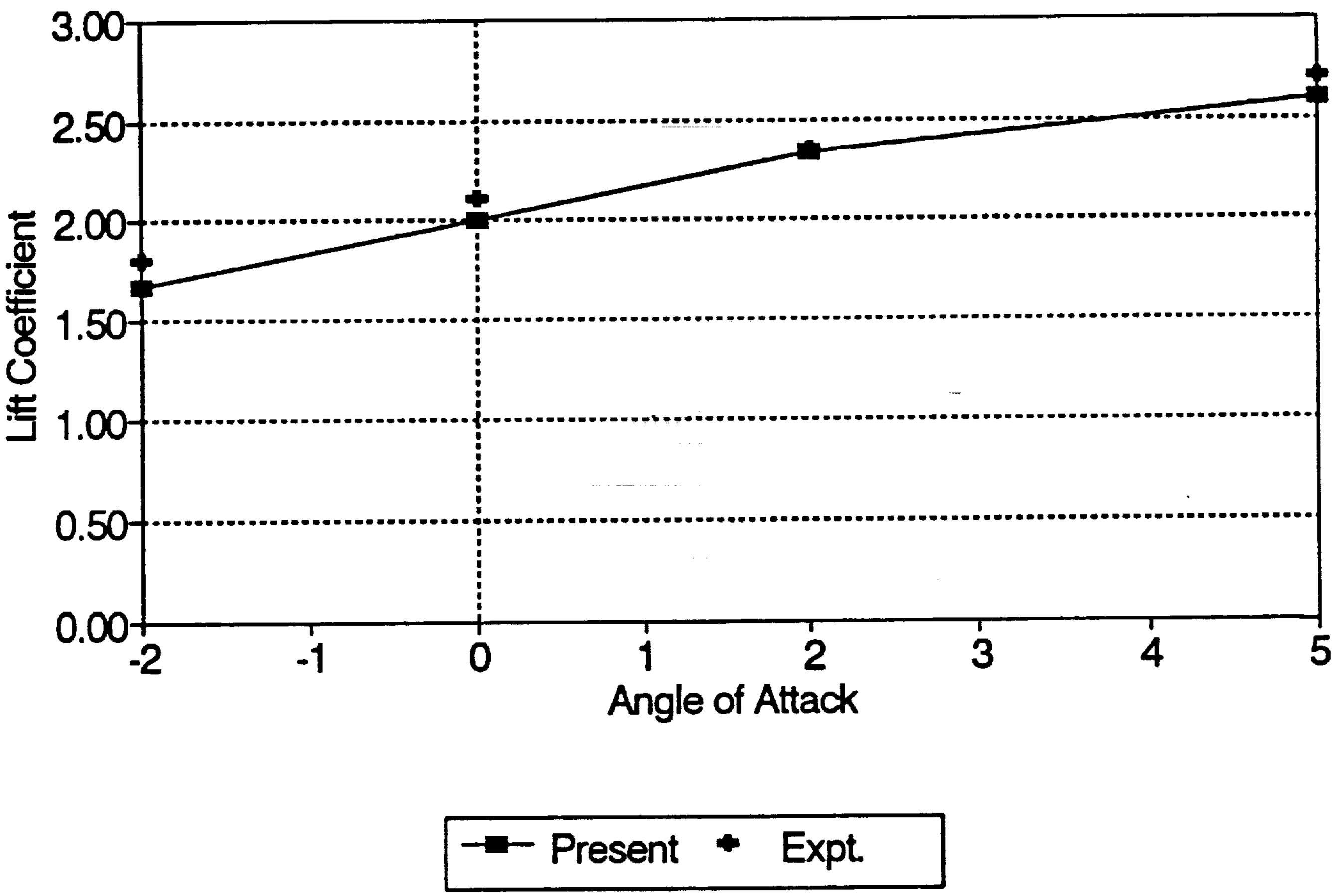


Pressure Coefficient on GA(W)-1 Airfoil, Flap Angle = 20 Degree, Re = 2.2 Million



Lift vs. Alpha for a GAW Airfoil-Flap

Mach Number = 0.3, Flap Angle = 25 Deg.



APPENDIX

Effects of Icing and Surface Roughness on Aerodynamic Performance of High-Lift Airfoils.

V.M. Kaladi, Post Doctoral Fellow
L. N. Sankar, Professor
School of Aerospace Engineering
Georgia Institute of Technology, Atlanta, GA 30332

Introduction

The problem of icing in aircraft operations is currently receiving considerable attention in the aerospace community owing to the number of weather-related accidents blamed on the formation of ice on aerodynamic surfaces. This increased interest in the effects of icing and such other meteorological phenomena worldwide is reflected in papers presented at a recent AGARD Conference (Ref:1) by researchers in the industry, government and academia. Brumby (Ref:2) provides a sample list of accidents where icing on wings has been found to be a contributing factor. It can be seen that icing adversely affects aircraft of all sizes and wing configurations. In the past, the efforts have concentrated on ice protection systems including anti-icing and de-icing systems. The most common approach for protection of transport aircraft has been to use the hot compressor bleed air flowing inside the wing. With the recent advances in engine technology, the use of hot bleed air for ice protection is increasingly becoming an unacceptable solution. Today's high-by-pass ratio engines have much less hot bleed air available for de-icing. Also, designers of high-performance aircraft are seeking ways of avoiding the extra burden of ice protection systems. Systems that tolerate a certain amount of ice build-up without significant performance penalties are being sought. There is, therefore, a need for a more basic understanding of the physical processes involved and also for a more accurate prediction of the potential performance penalties associated with icing.

It is well known that the most critical flight regimes affected by icing are take-off and landing. The primary effect of the presence of ice on wings is a dramatic change in the lift available at an angle of attack. Large losses in the lift coefficient as well as the occurrence of premature stall are noticed even with small amounts of ice accretion. Associated problems of increased aircraft drag and degradation in stability are also observed. It is imperative to obtain a better understanding of these phenomena.

Historically, analytical attempts to study the effect of icing on clean and iced multi-element airfoils were made using panel methods. However, these methods fail when separation occurs over the flap or at the leading edge due to appreciable ice accretion. In such cases, it is desirable that these methods be complemented by Navier Stokes methods for detailed analysis.

The present authors have, in the past, studied the effects of icing on fixed and rotary wing aircraft using Navier Stokes methods (Ref:3,4,5). These studies have produced results that correlate well with experimental data for clean and iced conditions and have thus demonstrated the capability for predicting accurately the effects of icing. The present study builds on this earlier work by extending the analysis to include high-lift devices such as trailing edge flaps, leading edge slats, etc. Only small-scale ice accretion is modeled and the analysis is restricted to the two-dimensional case.

Mathematical and Numerical Formulation

A 2-D multi-zone Navier Stokes solver forms the basis for the computations carried out in this study. Thus, the flow field surrounding the multi-element airfoil is divided into zones depending on the number of component elements to be modeled. The code has the capability to handle an arbitrary number of flow zones. In each of the flow zones, the 2-D, compressible, Navier Stokes solver is employed. The governing equations are discretized using finite difference schemes which are first order accurate in time and second order accurate in the spatial derivatives. The solution proceeds using an implicit, ADI scheme (Ref: 6).

Figure 1 shows a sample flow domain partition for a 2-element airfoil. The flow zones are numbered from the top to the bottom. For the 2-element airfoil shown in figure 1, the flow field is divided into three zones separated by the forward and aft wake lines and the airfoil solid surface lines.

A typical body-fitted grid for the problem is shown in figure 2 for an airfoil/flap combination and in figure 3 for an airfoil/slat combination. Figure 4 shows the details of the grid in the vicinity of the airfoil. The grid is generated using an elliptic solver which can generate computational grids around arbitrary airfoil sections. The grid generator provides the user with some control over the location and spacing of the grid points both in the streamwise and normal directions. The grid generation code has been structured to handle an arbitrary number of airfoil elements.

In the present numerical procedure, all the boundary conditions are applied explicitly at each time step. At all the solid surfaces the no-slip boundary condition is imposed. Since only low subsonic Mach number flows are dealt with in these cases, the far-field flow quantities are assumed equal to the undisturbed freestream values. At the interfaces between the different flow zones continuity of flow properties is ensured by taking averages for the flow quantities at the interface from the values in the neighboring zones. In some versions of the present solver, the blocks overlap by one cell so that the governing equations can be solved for, directly at the interface boundaries.

Results and Discussion

Preliminary computations have been carried out using the present method for the flow over a GAW-130 airfoil/flap combination. Figure 5 shows the chordwise pressure distribution over the main airfoil as well as the flap. The results shown are for a flap angle of 20° and an angle of attack of 5°. It is seen that the C_p distributions over the main airfoil as well as the flap compare well with the experimental results for the same case (Ref: 7, 8).

The above result was obtained for a clean airfoil-flap combination with no ice accretion. Next the effects of small scale ice accretion on the high-lift system aerodynamics were looked at. The small-scale icing considered here may be viewed as a roughness effect. In the Baldwin-Lomax eddy viscosity model that is used in the present work, the roughness effect is incorporated as a modification, Δy, that must be added to the mixing length. The values of Δy depend on the roughness height ratios. The empirical equation used in this case is taken from Cebeci (Ref: 9):

$$\Delta y = 0.9 (\nu / u_\tau) [\sqrt{(k_s^+)} - k_s^+ \exp(-k_s^+/6)]$$

where ν is the kinematic viscosity, u_τ is the wall shear stress and k_s^+ is the equivalent sand roughness factor.

Preliminary results for an iced GAW-130 airfoil/flap case are presented here. Table 1 shows the results obtained for the GAW-130 airfoil/flap under the same flow conditions with different levels of surface roughness due to icing modeled. The roughness factor, k_s^+, is varied from 20 to 2000 and the computed C_x and C_y values are tabulated. As can be seen from this table, roughness has the effect of reducing C_y, though the effect is not appreciable at this low angle of attack condition. The drag is found to be dominated by the pressure drag on the flap and the effect of roughness on drag is not very noticeable. However, in general the trends for the effect of icing roughness on lift and drag are seen to be in the expected direction.

In the full paper, additional computational results will be presented for the GAW-130 airfoil-flap combination for flow conditions near the maximum lift where the effect of icing roughness is expected to be more significant. There is also some experimental data available for a Boeing 737-200 ADV wing section (Ref: 9). If detailed geometry information is available for this case, correlation study results for this case will also be included in the full paper.

Acknowledgments

This work was sponsored by NASA Lewis Research Center under grant NAG3-768.

References

1. "Effects of Adverse Weather on Aerodynamics," AGARD Conference Proceedings 496, December 1991.

2. Brumby, R. E., "The Effect of Wing Ice Contamination on Essential Flight Characteristics," Presented at the Fluid Dynamics Panel Specialists' Meeting, Toulouse, France, April 29 - May 1, 1991, AGARD-CP-496.

3. Kwon, O. and Sankar, L., "Numerical Study of the Effect of Icing on Finite Wing Aerodynamics," AIAA Paper 90-0757, Jan. 1990.

4. Kwon, O. J. and Sankar, L. N., "Numerical Study of the Effect of Icing on Fixed and Rotary Wing Performance," AIAA Paper 91-0662, Jan. 1991.

5. Potapczuk, M. G., Bragg, M. B., Kwon, O. J., and Sankar, L. N., "Simulation of Iced Wing Aerodynamics," Presented at the Fluid Dynamics Panel Specialists' Meeting, Toulouse, France, April 29 - May 1, 1991, AGARD-CP-496.

6. Beam, R. M., and Warming, R. F., "An Implicit Factored Scheme for the Compressible Navier-Stokes equations," AIAA Journal, Vol. 16, No. 7, July 1978, pp. 393-402.

7. Narramore and Sankar, L. N., Proceedings of the Third International Conference on Rotorcraft Aerodynamics, Atlanta, GA, 1990.

8. Wentz, W. H., Seetharam, H. C., "Development of a Fowler Flap System for a High Performance General Aviation Airfoil," NASA CR-2443, December 1974.

9. Cebeci, T. and Chang, K. C., "Calculation of Incompressible Rough-Wall Boundary-Layer Flows," AIAA Journal, Vol. 16, No. 7, July 1978, pp. 730-735.

10. Potapczuk, M. G. and Berkowitz, B. M., "An experimental Inversitgation of Multi-Element Airfoil Ice Accretion and Resulting Performance Degradation," NASA TM-101441, Jan 1989.

FIGURE 1

Grid Topology for a Two-Element Airfoil

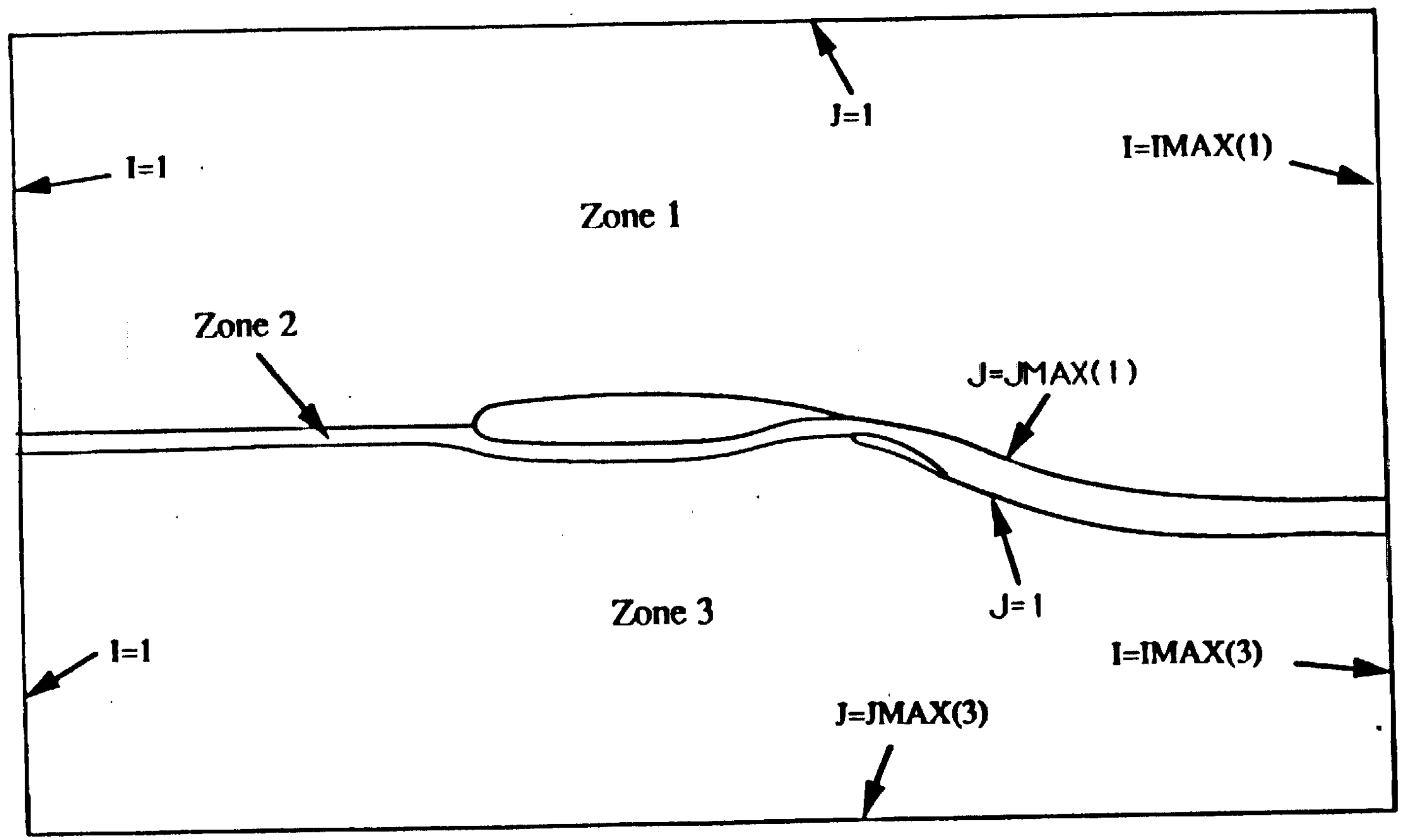
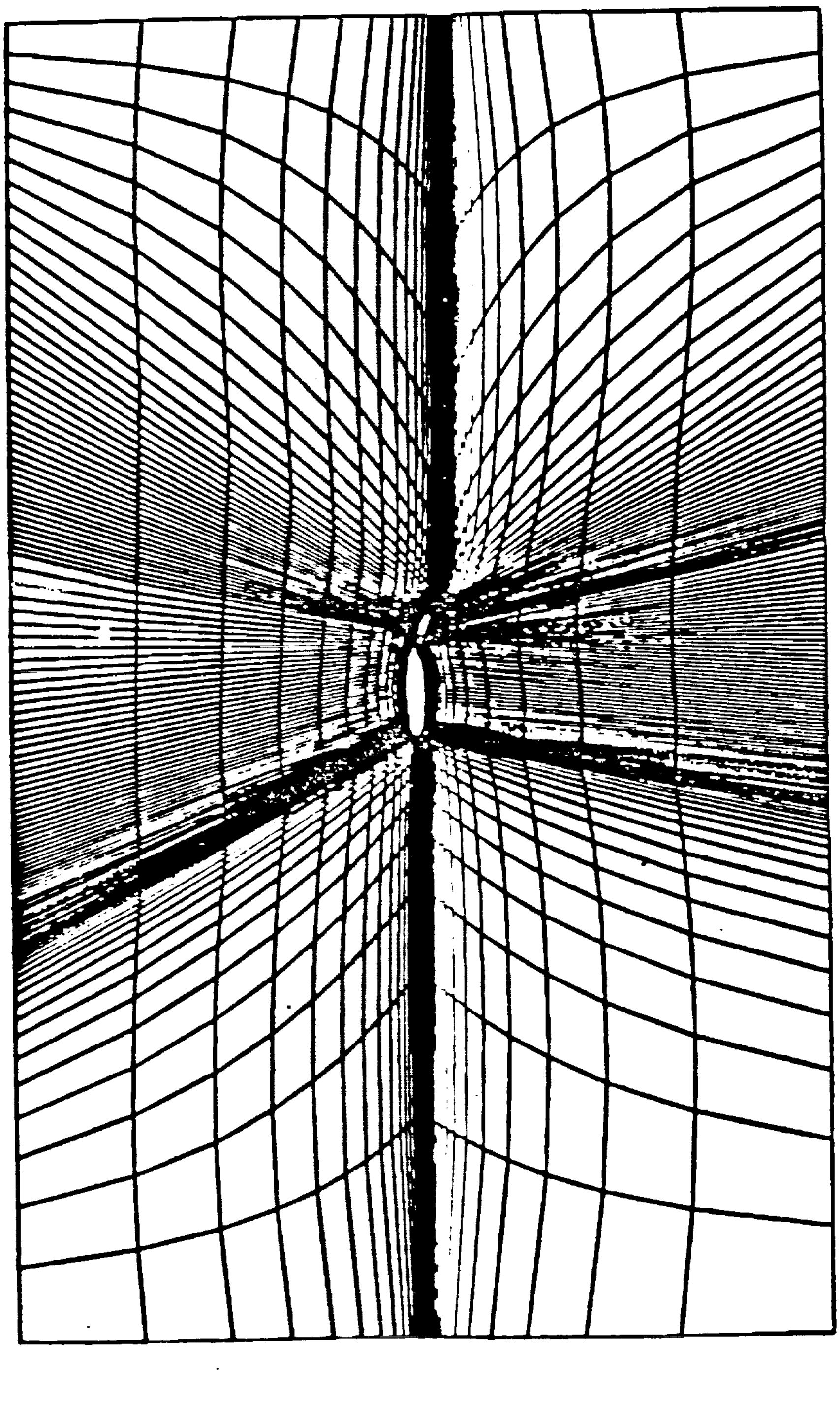
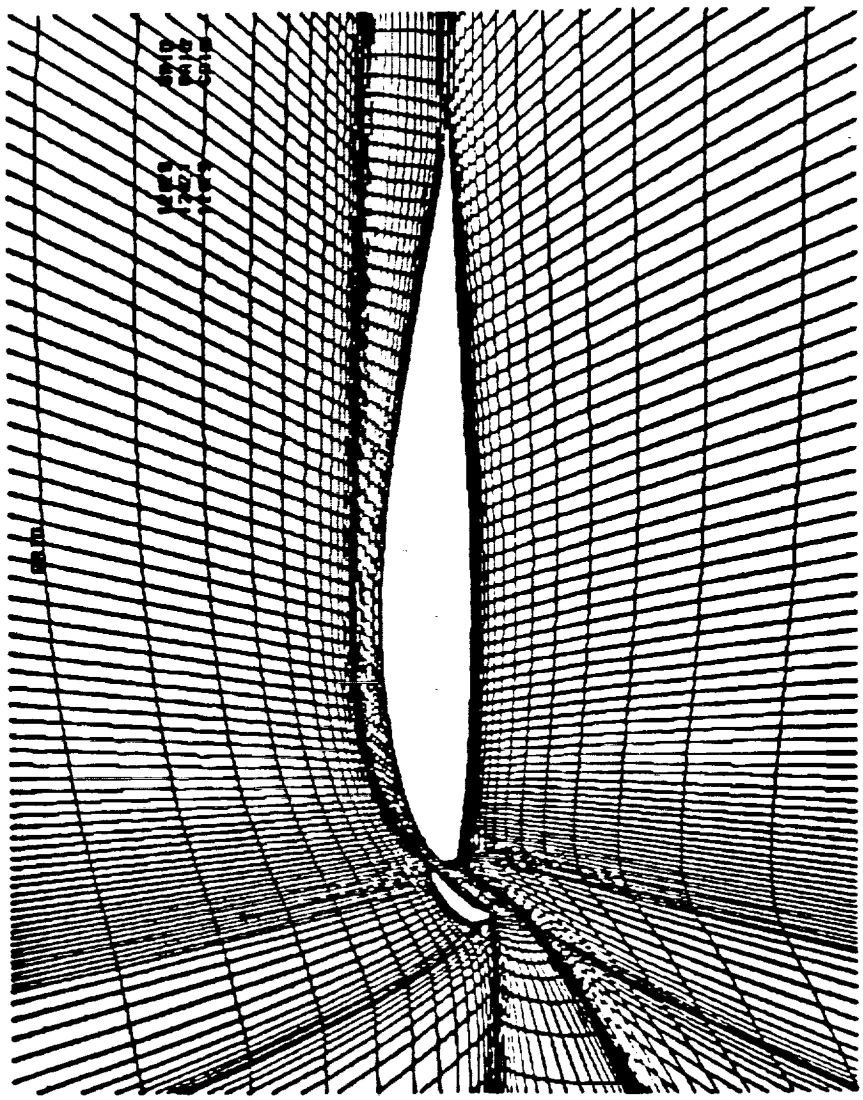


FIGURE 2



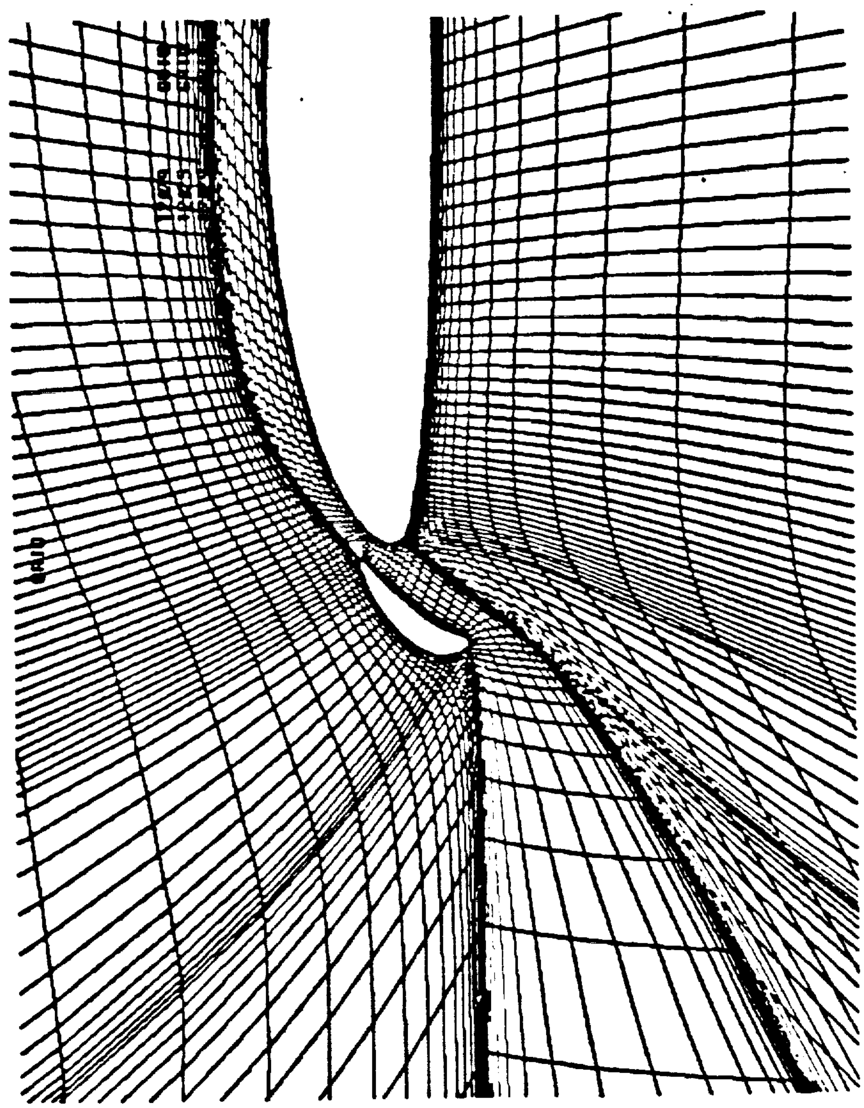
Computational Grid for GA(W)-1 Airfoil/Flap Combination

FIGURE 3



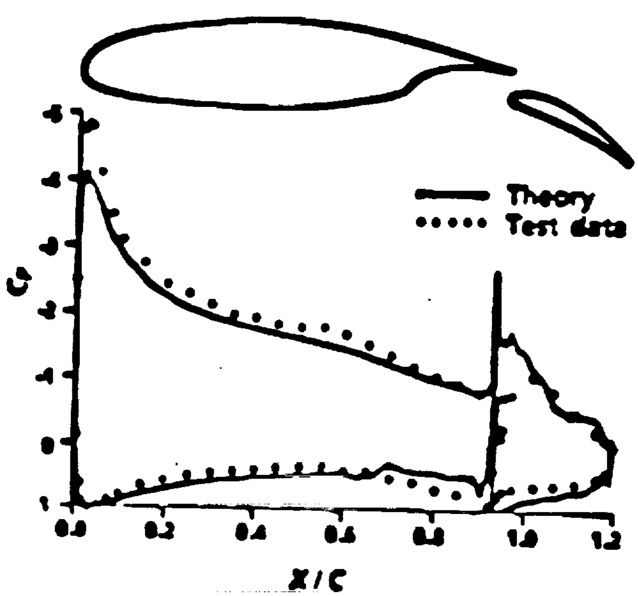
Computational Grid for VR-7 Airfoil/Slat Combination

FIGURE 4



Computational Grid in the vicinity of slat for VR-7 Airfoil/Slat Combination

FIGURE 5



Pressure distribution - GAW-130 airfoil, flap angle 20°, $M = 0.3$, $Re = 2.2x10^6$.

Table 1

k_s^+	C_τ	C_v
20	1.9031	.2136
200	1.9029	.2139
2000	1.9014	.2242