NASA-CR-192234

LEWIS-9N
318155
P9

SIMULATION OF UNSTEADY ROTATIONAL FLOW OVER PROPFAN CONFIGURATION

IN-02-CR
146043
P-9

NASA GRANT No. NAG 3-730

FINAL REPORT

for the period

JUNE 1986 - NOVEMBER 30, 1990

submitted to

NASA LEWIS RESEARCH CENTER

CLEVELAND, OHIO

Attn: Mr. G. L. Stefko

Project Monitor

Prepared by:

Rakesh Srivastava
Post Doctoral Fellow

L. N. Sankar
Associate Professor

School of Aerospace Engineering
Georgia Institute of Technology
Atlanta, Georgia 30332

INTRODUCTION

During the past decade, aircraft engine manufacturers and scientists at NASA have worked on extending the high propulsive efficiency of a classical propeller to higher cruise Mach numbers. The resulting configurations use highly swept twisted and very thin blades to delay the drag divergence Mach number. Unfortunately, these blades are also susceptible to aeroelastic instabilities. This was observed for some advanced propeller configurations in wind tunnel tests at NASA Lewis Research Center, where the blades fluttered at cruise speeds. To address this problem and to understand the flow phenomena and the solid fluid interaction involved, a research effort was initiated at Georgia Institute of Technology in 1986, under the support of the Structural Dynamics Branch of the NASA Lewis Research Center. The objectives of this study are:

a) Development of solution procedures and computer codes capable of predicting the aeroelastic characteristics of modern single and counter-rotation propellers,

b) Use of these solution procedures to understand physical phenomena such as stall flutter, transonic flutter and divergence

Towards this goal a two dimensional compressible Navier-Stokes solver and a three dimensional compressible Euler solver have been developed and documented in open literature.

The two dimensional unsteady, compressible Navier-Stokes solver developed at Georgia Institute of Technology has been used to study a two dimensional propfan like airfoil operating in high speed transonic flow regime. A two degree of freedom structural model was coupled to study classical flutter and flutter dip phenomena observed at transonic Mach numbers. The Euler code solving three dimensional compressible Euler equations was used to predict the flow field around single and counter rotation advanced propellers. This solver has been applied for several flight conditions and both aerodynamic and aeroelastic analyses have been done.

These flow solvers have been validated through comparisons with measured data and documented in several publications (Refs. [1-5]), and have demonstrated the importance of accounting for the elastic characteristics of the structure while evaluating the aerodynamic characteristics of modern high speed propellers. Currently the solver is being modified to solve inviscid flow past ducted propellers.

SUMMARY OF PROGRESS

Two Dimensional Studies:

The two dimensional compressible Navier-Stokes solver has been used for numerical simulation of stall flutter and transonic flutter dip phenomena. For this purpose the aerodynamic solver was coupled to a two degree of freedom structural dynamics solver allowing the airfoil to rigidly pitch and plunge. The resulting solver was used to explore the stall flutter and classical flutter characteristics of several airfoils, through a simultaneous integration in time of the fluid solid dynamics equations. This solver has been shown to predict the onset of stall flutter. The results obtained have been summarized in Ref. [1]. The solver was also used to successfully predict the transonic flutter dip phenomena and to study the effects of various parameters, such as thickness, shape and viscosity on the phenomena. These results have been summarized in Ref. [2] .

Three Dimensional Studies:

A three dimensional Euler solver developed at Georgia Institute of Technology for solving flow past isolated helicopter blades has been modified to solve the flowfield around advanced propeller configurations. The solver may be used to study steady or unsteady compressible flow past single and counter rotating propellers. In order to make the solver efficient, a hybrid scheme is used. The solver is second order accurate in space and first order accurate in time. The accuracy has been extended to fourth order using the compact (Pade) formulation along the streamwise direction.

The advanced propeller blades are somewhat flexible and under cruise conditions are highly loaded. This could lead to large structural deformations. These deformations could be critical to the performance of advanced propellers. In order to study the effect of deformation on the propeller aerodynamic performance characteristics, the solver was modified to be coupled in an open loop fashion with any structural solver.

The solver allows the flowfield to be divided into different relatively moving blocks. This considerably simplifies flow solutions around complex geometries, such as counter rotating propellers or ducted propellers. An effort is currently underway to update the solver for solving flow past ducted propellers. The formulation and results obtained for single and counter rotation propellers are documented in detail in the Ph. D. dissertation [3] of Dr. Rakesh Srivastava, which will be published as a NASA CR in 1991.

Single Rotation Propeller: The modified solver was applied for predicting the aerodynamic performance characteristics and blade loading of advanced geometries such as the SR-3 and the SR-7 propellers. To efficiently solve the flowfield around an advanced propeller, the solution domain was divided into several blocks. In the present study the number of blocks were kept the same as that of the number of blades. The Euler equations are solved in one block at a time, with the flow properties associated with the remaining blocks being stored in the solid state memory. The additional fluid block boundaries were updated by averaging the flow properties across the boundary. The boundary conditions also allow for axisymmetric and unsymmetric (unsteady) flight configurations. The results obtained show good comparison with experimental data [6,7] for spanwise and chordwise blade loading. It was also possible to properly and accurately capture the leading edge suction peaks. These results have been documented in detail in Ref. [3,4]. Because Ref.4 is not yet available in an archival form, a copy of Ref. [4] is included in the appendix.

Counter Rotation Propeller: As mentioned earlier the flow solver has been written in a fashion to treat a complex flowfield by dividing it into different possibly relatively moving blocks. This scheme was utilized here to obtain flow solutions around counter rotating propellers by dividing the solution domain into blocks associated with each of the blade rows. As the blades are rotating in opposite direction the grid blocks will also be rotating in opposite direction with one common interfacing surface. The Euler equations are solved in each of the blocks independently. The communication between the blocks is handled by averaging the flow properties across the interface boundary. Using the averaging procedure used in the present solver allows arbitrary time steps without requiring complex grid distortion techniques. The solver was used to predict performance characteristics of a GE F7/A7 counter rotating propeller. The results obtained showed good comparison with experimental data [8]. The comparisons are not as good for lower advance ratio where the blades are loaded more heavily than higher advance ratios. Higher loading will lead to larger deformations under operating conditions. As the solver is purely aerodynamic, it was not possible to account for these deformations. The formulation of the solver and the results are documented in detail in Ref. [3].

Aeroelastic Effects: The solver has been also modified to allow the study of blade deflections and deformations under loading. However, the solver does not have the capability of solving structural equations. The effect of deformations on performance was studied by coupling the solver with NASTRAN in a loose open loop fashion. This requires interpolating the loads obtained from the aerodynamic solver grid onto the NASTRAN grid, and interpolating the deflections from the NASTRAN grid back onto the aerodynamic solver grid. First, a centrifugally deformed geometry was used to calculate aero loads on a SR-7 advanced propeller. These air loads were the used in NASTRAN to calculate a new deformed shape due to combined centrifugal and steady air loads. This deformed shape was then again used to obtain the updated air loads using the Euler solver. This study showed that as much as 40% change in performance can be expected once the blade deformations are taken into account. The formulation and the iteration process has been discussed in detail in Ref. [3] and has been documented in Ref. [5] and may be found in the appendix.

Ducted Propeller: Work is currently under way to modify the solver for solving the flowfield around ducted advanced propellers. Again the ability of the solver to divide the flow domain into different possibly relatively moving blocks is being used. In order to be able to resolve the suction peak of the cowl a wrap around C-O grid topology is being used. The grid topology around the blades is being retained as H-O. The grid block around the cowl will be stationary whereas the grid block around the blades will be rotating with the blades. The communication between the blocks would again be handled by averaging the flow properties across the block interface boundary. The coding modifications have been completed however, the code has not been applied and verified at the present time.

CONCLUDING REMARKS

A versatile two dimensional Navier-Stokes solver capable of carrying out stall flutter and classical flutter has been developed and verified. It has successfully predicted the onset of stall flutter and flutter dip phenomena for several airfoil cross sections operating in different speed regimes.

An efficient three dimensional Euler solver capable of obtaining flow solutions around advanced propeller geometries, such as single rotation counter rotation and ducted propellers, has been developed. The predicted spanwise and chordwise blade load distributions compare very well with experimental measurements. It has also been successfully coupled with the NASTRAN structural analysis program to study the effect of blade loading on the performance of advanced propellers.

REFERENCES

1. Wu, J. C., Kaza, K. R. V., and Sankar, L. N., "A Technique for Prediction of Airfoil Characteristics in Separated Flow," *Journal of Aircraft.* Vol. 26, No. 2, 1989, pp. 168-176.
2. Reddy, T. S. R., Srivastava R., Kaza, K. R. V., "The Effects of Rotational Flow, Viscosity, Thickness, and Shape on Transonic Flutter Dip Phenomena," AIAA Paper 88-2348, April 1988 (to appear in *Journal of Aircraft*).
3. Srivastava, R., "An Efficient Hybrid Scheme for the Solution of Rotational Flow Around Advanced Propellers," Ph. D. Dissertation, Georgia Institute of Technology, August 1990.
4. Srivastava, R., and Sankar, L. N., "An Efficient Hybrid Scheme for the Analysis of Counter Rotating Propellers," AIAA Paper 91-0703.
5. Srivastava, R., Sankar, L. N., Reddy, T. S. R., and Huff, D. L., "Application of an Efficient Hybrid Scheme for Aeroelastic Analysis of Advanced Propellers," AIAA Paper 90-0028, January 1990 (to appear in *Journal of Propulsion and Power*).
6. Roharbach, C., Metzger, F. B., Black, D. M., and Ladden, R. M., "Evaluation of Wind Tunnel Performance Testings of an Advanced 45^o Swept Eight-Bladed Propeller at Mach Numbers From 0.45 to 0.85," NASA CR-3505, 1982.

7. Bushnell, P., " Measurement of the Steady Pressure Distribution on a Single Rotation Large Scale Advanced Prop-Fan Blade at Mach Numbers From 0.03 to 0.78," NASA CR 182124, July 1988.

8. Jeraki, R. J., and Rose, G. E. "Effects of Mach Number, Loading, and Blade Angle on High-Speed Performance of the F7/A7 Highly Loaded Counterrotation Propeller," NASA TP 2927, 1991.