AIAA-83-0236
Three-Dimensional Euler Equation
Simulation of Propeller-Wing Interaction in Transonic Flow
D.L. Whitfield, Mississippi State Univ., Mississippi State, MS; and A. Jameson, Princeton Univ., Princeton, NJ

AIAA 21st Aerospace Sciences Meeting

January 10-13, 1983/Reno, Nevada

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics
1290 Avenue of the Americas, New York, NY 10104
THREE-DIMENSIONAL EULER EQUATION SIMULATION OF PROPELLER-WING INTERACTION IN TRANSONIC FLOW

D. L. Whitfield
Department of Aerospace Engineering
Mississippi State University
Mississippi State, MS 39762

and

A. Jameson
Department of Mechanical and Aerospace Engineering
Princeton University
Princeton, N. J. 08544

Abstract

A method is presented for the computation of propeller-wing interaction in transonic rotational flow. The approach is to use the three-dimensional time-dependent Euler equations with forcing terms included to simulate the propeller. Viscous-inviscid interaction on the wing surface is included by coupling the three-dimensional Euler equations with the two-dimensional compressible turbulent inverse integral boundary-layer equations. Numerical solutions are compared with experimental data for a 32° swept supercritical wing without a propeller simulator, wing with a propeller simulator producing thrust only, and wing with a propeller simulator producing thrust and swirl in each direction.

I. Introduction

Studies indicate that a significant reduction in fuel consumption of transport aircraft can be achieved by advanced technology turboprop or propfan propulsion systems. It is intended that such propulsion systems be used on transonic aircraft. Small changes in transonic flow about a wing can cause appreciable change in shock wave strength and location, and consequently influence lift, drag, boundary-layer growth, separation, etc. Because propellers can produce significant changes in the transonic flow about a wing, it is necessary to understand the influence of a propeller slipstream on a supercritical type wing. The purpose of this paper is to present a computational fluid dynamic method of simulating three-dimensional transonic propeller-wing interaction, including swirl.

Most investigations of propeller-wing flowfields have been limited to subsonic and high subsonic Mach numbers. Rice(7) and Harain(8) have investigated the transonic problem using the potential flow equations. A transonic propeller-wing interaction flowfield, however, is rotational due to variations in flow properties induced by the propeller, embedded shocks, and viscous effects. The approach taken in this work is to use the three-dimensional time-dependent Euler equations, including the energy equation, in order to allow for rotational flow. In addition, viscous effects are taken into account by coupling the three-dimensional Euler equations with the two-dimensional compressible turbulent inverse integral boundary-layer equations.

*This research was sponsored by the Lockheed-Georgia Company, Marietta, GA.

†Professor, Member AIAA.

‡Ph.D. Professor, Member AIAA.

The method used to incorporate the influence of a propeller in the Euler equations is described in the following section. The Euler equation solver is then briefly discussed followed by an explanation of how viscous-inviscid interaction is included in the computational method. Finally numerical results are presented and compared with results from a transonic flow experiment designed to simulate the influence of a propeller on a supercritical wing.

II. Propeller Simulation

The integral form of the Euler equations is used in the present work. From these conservation laws, certain jump conditions can be derived for discontinuity surfaces. One of the discontinuity surfaces permitted is a shock, across which there is mass flow. Shocks can be captured in the solution. Another discontinuity surface permitted is one whereby there is no mass flow across the surface. For this latter discontinuity surface, the jump in tangential velocity and density across the surface are arbitrary but the jump in pressure is zero. A jump in energy is permitted although the jump must satisfy an equation of state. Such jump conditions include those necessary to resolve a propeller slipstream. Assuming the method of incorporating the influence of the propeller in the Euler equations is adequate (to be explained below) the approach is to capture the slipstream in the solution as opposed to modeling it explicitly.

The method used to incorporate the influence of the propeller in the Euler equations is as follows. Forces operate on the fluid as a consequence of the lift and drag of the blades as illustrated in Fig. 1. Because the numerical formulation of the Euler equations used here is finite volume, the forces of the blades on the fluid become force vectors associated with the finite volumes located in the propeller region (Fig. 1). These finite volume force vectors are variable due to propeller rotation, blade twist, etc.

If these finite volume force vectors had resulted from the usual concept of a body force per unit mass, $f$, such as gravitational or electromagnetic forces, then the three-dimensional time-dependent Euler equations could be written in Cartesian coordinates for a stationary grid as

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \tag{1a} \]
of divergence theorem written in terms of a stress system using

Therefore, because numerical discretization of the
regard to Maxwell stresses
larities to that used in magnetohydrodynamics with
To account far the effect of
results in forces
on
applied to the cell faces, and numerical treatment
each component of force multiplied by the
stress system composed of forces per unit
exerted on the cell faces. This concept has simi-
larities to that used in magnetohydrodynamics with
regard to Maxwell stresses (see e.g. Shercliff(11)).
Therefore, because numerical discretization of the
last term in Eq. (1a) at each cell results in a
force in each of the three momentum equations, and
each component of force multiplied by the corre-
ponding component of velocity in the energy equa-
tion, the influence of the propeller on the flow
is regarded as a force per unit area (stress)
applied to the cell faces, and numerical treatment
of a surface integral rather than a volume integral
results in forces occurring in the Euler equations
in the same manner as body forces. The last term
on the left hand side of Eq. (1a), therefore, is
written in terms of a stress system using Gauss' 
divergence theorem as

\[ \vec{b} = \frac{\partial}{\partial t} \hat{n} - \left\{ 0, \vec{f}_x, \vec{f}_y, \vec{f}_z, \vec{f}_u + \vec{f}_v + \vec{f}_w \right\}^T \]  

(3)

and \( \vec{f} \) is a force per unit area.

Information necessary to specify the stress
vector \( \vec{f} \) might be obtained from the propeller char-
acteristics. The available experimental transonic
flow data of Wedge and Crowder(12) however, were
obtained using a simulator to investigate the
effects of a propeller on a supercritical wing.
Total pressure jumps were introduced by the simu-
lator and vanes were used to induce swirl.
The thrust was, therefore, imposed in the present
computations by introducing the same jump to total
pressure as used in the experiments. The force
components in the plane perpendicular to the thrust
were adjusted to provide the swirl velocities in-
duced by the simulator vanes. This was necessary
because swirl velocities are not imposed in the
computations, rather they are obtained as part of
the solution. A variation in the stress vector,
\( \vec{f} \), proportional to the square of the distance from
the center of the propeller (or simulator in this
case) was used to more appropriately simulate spa-
tial variations.

III. Euler Equations Solver

The method used to solve the three-dimensional
time-dependent Euler equations with force terms is an exten-
sion of that described in Ref. 13. Finite
volume spatial discretization is applied to
the integral form of the Euler equations and the
resulting equations are solved using a four-stage
two-level numerical scheme. The four-stage scheme
is second-order accurate whereas the four-
stage scheme used in Ref. 13 was fourth-order
accurate. The second-order scheme is used because it
requires slightly less storage than the fourth-
order scheme and, also, steady state solutions are
of interest as opposed to time accurate solutions.

Dissipative terms are introduced in this central
difference scheme in a separate filter stage
at the end of each time step. Convergence to a steady
state is accelerated by the addition of a forcing
term that depends on the difference between the
local total enthalpy and the freestream value of
enthalpy. Convergence is also accelerated by using
a local time step determined by the maximum Courant
number of \( 2/\sqrt{3} \). Far field boundary conditions are
based on a characteristic combination of vari-
able. (13) Pressure at the wall is determined by(14)
a three-dimensional version of the work of Rizzi
involving the normal momentum relation.

IV. Viscous-Inviscid Interaction

Viscous effects were taken into account in an
approximate fashion by coupling the three-dimen-
sional Euler equations with the two-dimensional
compressible turbulent inverse integral boundary-
layer equations. The inverse boundary-layer method
is described in Ref. 15. An inverse method allows
for regions of separated flow and the one described
in Ref. 15 can be used everywhere, whether or not the
flow separates.

Viscous-Inviscid interaction solutions were
obtained using the surface source method as ex-
plained in Refs. 16 and 17. This method accounts for interaction by imposing a surface source term in each of the Euler equations as described in Ref. 18. The iteration procedure used in Ref. 17 was followed except that the boundary-layer equations were solved after every 20 time cycles of the Euler equations rather than after every 50 cycles. The reason for this is that the time step resulting from the present three-dimensional grid (96 x 16 x 16) was larger than that resulting from the finer two-dimensional grid (128 x 30) used in Ref. 17. Both the inviscid and viscous-inviscid interaction solutions involved 400 Euler equation cycles. Carter’s method(19) of iterating on the displacement thickness was used, with a relaxation of unity.

Boundary-layer solutions were obtained over the wing in the freestream direction at each spanwise location. Ten cells in the spanwise direction were taken on the wing. The boundary-layer solutions were started at each span station at the 20 percent local chord location on the lower surface and at the 15 percent local chord location on the upper surface, as this corresponded to the boundary-layer trip locations used in the experiment. (12)

V. Results and Comparisons

Available experimental transonic data appear to be those of Wedge and Crowder. (12) This experiment was stated (12) to be of an exploratory nature to identify order-of-magnitude slipstream-wing interaction effects and not to establish highly accurate results. A simulator consisting of a high pressure air driven ejector system was used to simulate a modern prop-fan slipstream. A schematic of the simulator and wing is shown in Fig. 2. The simulator exit diameter was 11.7 percent of the model span. The airfoil sections defining the wing are tabulated in Ref. 12. This wing had 32° of quarter chord sweep and a taper ratio of 0.30.

All numerical solutions were obtained on a 96 x 16 x 16 C-type grid with 60 points on the wing at each of 11 span locations. The experiment used a wing-body combination; however, the grid was for a wing alone. Moreover, for these initial computations no special gridding of the propeller was used. The propeller (or simulator) region was, therefore, essentially a rectangle in appearance when viewed from directly upstream or downstream. The C-type grid did permit a reasonably vertical simulation of the propeller region.

An example of viscous-inviscid interaction results compared to purely inviscid results is given in Fig. 3 for $M_a$ (freestream Mach number) of 0.80, Re (Reynolds number based on mean aerodynamic chord) of $2.3 \times 10^{6}$, $\alpha$ (angle of attack) of 1.83°, $\eta$ (semispan location) or 0.65, and no propeller simulator. As illustrated in Fig. 3, viscous-inviscid interaction improves the agreement between the numerical and experimental results by moving the shock location forward and decreasing the lift in general. However, this grid is not sufficiently fine, particularly over the aft portion of the wing where the grid spacing approaches 5 percent of the local chord, to resolve the shock adequately. The discrepancy on the aft portion of the lower surface exists in all comparisons. This may be an indication that three-dimensional boundary-layer effects, not included in these computations, are important.

Comparisons of numerical and experimental wing surface pressures with and without the simulator, and with and without the swirl are given in Fig. 4 for $M_a = 0.70$ and $\alpha = 3°$. The term $\Delta P_T$ is the jump in total pressure at the simulator. Results are presented at two span locations, $\eta = 0.35$ and $\eta = 0.50$. These span locations were selected because the propeller simulator has the greatest influence on the flow about the wing at these locations (see Fig. 2). Figures 4a and 4b indicate that the propeller simulator changed the wing
Fig. 4  Numerical and Experimental Wing Surface Pressures With and Without Simulator for $M_a = 0.70$, $Re = 2.3 \times 10^6$, and $\alpha = 3^\circ$
pressure distribution somewhat compared to no simulator. However, Figs. 4c and 4d indicate that swirl, both up inboard and down inboard, has a much larger effect. Swirl influences the pressure distribution more on the inboard station ($n = 0.35$) than the outboard station ($n = 0.50$). The computations in Fig. 4 include viscous-inviscid interaction, and are considered to be in reasonable agreement with the experimental data.

Numerical and experimental span load distributions are presented in Fig. 5 for $M_w = 0.80$ and $\alpha = 3^\circ$. The span load parameter used in Fig. 5, $cc_p/c_{mac}$, is the local lift coefficient $c_p$ times the ratio of the local chord $c$ to the mean aerodynamic chord $c_{mac}$. The most interesting results are those which include swirl. Quantitative agreement between theory and experiment is worse on the outboard portion of the wing although the numerical results are in good qualitative agreement with the experimental data in Fig. 5. The computations include viscous-inviscid interaction.

The influence of the propeller simulator on the upper wing surface boundary layer for $M_w = 0.80$ and $\alpha = 3^\circ$ is illustrated in Fig. 6. Of particular interest is whether or not flow separation occurs as a consequence of the propeller slipstream. Although the boundary-layer treatment is two-dimensional, the inverse method used can predict separated flow. The local skin friction coefficient, $c_f$, is plotted in Fig. 6 at $n = 0.43$ which is the wing span station located downstream of the center of the propeller simulator. The influence of $7^\circ$ down inboard swirl as compared to $7^\circ$ up inboard swirl is to strengthen the upper surface shock and move it aft somewhat. Although the flow does not separate in the computations with swirl in either direction, the upper surface boundary layer corresponding to the case with $7^\circ$ down inboard swirl is close to separation just downstream of the shock and near the trailing edge due to the stronger shock.

Experimental incremental lift coefficients are presented in Ref. 12 for various simulator to freestream total pressure ratios ($P_t / P_w$), freestream Mach numbers and up inboard swirl angles. The lift increments are referenced to $P_t / P_w = 1.000$ and zero swirl. Computations for simulator to freestream Mach numbers of 0.70 and 0.80, and swirl angles of zero and $7^\circ$ up inboard are compared with experimental data in Fig. 7. The computed incremental lift results with viscous-inviscid interaction are shown in Fig. 7 to be in good agreement with the experimental data.

### VII. Conclusions

A computational method was presented for solving the three-dimensional time-dependent Euler equations with force terms which are similar to body forces. These particular force terms were introduced to simulate a propeller. In addition, viscous effects were accounted for in an approximate fashion by coupling the three-dimensional Euler equations with the two-dimensional compressible...
turbulent inverse integral boundary-layer equations.

- **7° UP INBOARD SWIRL**
- **M∞ = 0.70**
- **M∞ = 0.80**

OPEN SYMBOLS ARE EXPERIMENT
CLOSED SYMBOLS ARE COMPUTATIONS

Viscous-inviscid interaction computations were carried out for transonic flow about a 32° swept supercritical wing with: (1) no influence of a propeller, (2) the influence of a propeller producing thrust only, and (3) the influence of a propeller producing both thrust and swirl (in either direction). Although a wing with only one propeller was investigated, there is no restriction on the number of propellers that can be included, nor the direction of swirl produced by each propeller. The 96 x 16 x 16 grid used was rather coarse; however, comparisons with experiments indicate the computational method is a useful tool for investigating this practical three-dimensional flow problem with vorticity.

All computations were performed on a CRAY-1S computer that had about 900,000 words of available memory. A viscous-inviscid interaction solution on a 96 x 16 x 16 grid with 400 Euler equation cycles and 20 boundary-layer equation cycles required 341 seconds and the 900,000 words of memory.

References

16. Whitfield, D. L., Swafford, T. W., and...

