TIP VORTICES OF WINGS IN SUBSONIC AND TRANSONIC FLOW: A NUMERICAL SIMULATION

G R SRINIVASAN ET AL.

Tip Vortices of Wings in Subsonic and Transonic Flow: A Numerical Simulation

G.R. Srinivasan and W.J. McCroskey

July 1986

"Original contains color plates: All DTIC reproductions will be in black and white"
Tip Vortices of Wings in Subsonic and Transonic Flow: A Numerical Simulation

G. R. Srinivasan, JAI Associates, Mountain View, California
W. J. McCroskey, U. S. Aeroflightdynamics Directorate - AVSCOM,
Ames Research Center, Moffett Field, California

July 1986
3ACFM 1986

TIP VORTEXES OF WINGS IN SUBSONIC AND TRANSONIC FLOWS: A NUMERICAL SIMULATION*

G. R. SRINIVASAN AND W. J. McCROSKEY†

JAI Associates Inc, Mountain View, California 94042, U. S. A.
†U.S. Army Aeroflightdynamics Directorate - AVSCOM
NASA Ames Research Center, Moffett Field, California 94035, U. S. A.

ABSTRACT Thin layer Navier-Stokes and Euler equations are numerically solved using a multi-block zonal approach to simulate the formation and roll-up of tip vortices of wings in subsonic and transonic flows. Several wing planforms have been considered to examine the influence of tip-cap shape, planform geometry and free stream Mach number on the formation process. A good definition of the formation and qualitative roll-up of tip vortices has been achieved.

1. Introduction

The phenomenon of vortex formation and its subsequent roll-up behind wings has long attracted attention because of the potential hazard to aircraft that encounter them in the flight and because of the noise and vibration problems that they cause for helicopter rotors. Although the formation of the tip-vortex is a viscous phenomenon, recently Rizzi et al [1] have used Euler equations to numerically simulate the vortex formation. Their reasoning for this success is that the artificial viscosity, used in the numerical scheme to control the nonlinear stability, played the role of the natural viscosity. More recently, Mansour [2] used the thin-layer Navier-stokes equations in conjunction with an elliptic grid generation scheme to simulate the flow field and the tip-vortex on a low-aspect-ratio wing. In the present work an alternate but more efficient method of multi-block zonal approach [3,4] is used for solving the thin-layer Navier-Stokes/Euler equations for isolated wings with a view to simulate the tip-vortex formation in subsonic and transonic flows.

2. Governing Equations and Numerical Scheme

At the Reynolds numbers of interest in the present calculations, the flow on the wing can be considered to be mostly inviscid except in a small region near the body surface where the viscous effects are important. Accordingly, the flow field is solved using a multi-block zonal approach developed at NASA Ames [3,4] wherein the inviscid outer blocks are solved using Euler equations and the inner viscous blocks are solved using thin-layer Navier-Stokes equations. The governing equations are generally nondimensionalized by free-stream quantities and are transformed to the computational domain so as to preserve the strong conservation law form of the equations.

* Research supported under U.S. Army Research Office Contract DAAG29-85-C-0002
Typically each of the computational zone or block has over 40,000 grid points. The grid topology consists of a H-H grid generated by the parabolic grid solver of Edwards [5] and the outer grid had the dimensions of $10 \times 4 \times 10$ for the wing C and $10 \times 10 \times 10$ for the ONERA wing. The above dimensions are based on the root chord of the wing. Each viscous zone typically had $61 \times 27 \times 25$ grid points and was clustered at the leading edge, tip region and in the normal direction.

The numerical scheme used a diagonalized ADI algorithm due to Pulliam and Chaussee [6] with several additional features incorporated to improve the efficiency of the code. The details of this as well as boundary conditions and data management are discussed in detail in Refs. [3-4,7]. With this algorithm, the CPU time for each steady state solution (with over 160,000 grid points in the flow field) is typically under 3 hours on NASA Ames CRAY XMP machine. This is an order of magnitude savings in computational time over Mansour's code [2].

3. Results

Although several wing planforms are considered in this study of tip vortex simulation, only selected numerical results are presented here for the lack of space. The details of these test cases as well as the results are described elsewhere [8].

Typical results in transonic flow cases are described here. Figure 1 shows representative pressure distributions at two spanwise stations for Wing C, which is a small aspect ratio wing in a flow of Mach number $M_{\infty} = 0.9$ and at $\alpha = 5$ degrees angle of attack. The numerical results are compared with the experimental data of Keener [9]. The results show good agreement with the experiments over parts of the wing which do not have massive shock-induced separation. Discrepancies in the separated regions may be due to inadequate spanwise grid resolution, the thin layer assumption, and/or the simple turbulence model used.

Figure 2 shows a farfield view of the tip-vortex formation and roll-up for this wing. Fluid tracer particles were released at various heights from the surface and at several locations along the span to generate this picture. Particles released near the tip region as well as the ones in the separated region contribute to the tip-vortex formation. The fluid particles from the lower surface (high pressure region) seem to go around the tip and get braided into the particles from the upper surface near the tip region and from particles lifting up from the separated region. The trend of these particle paths seem to first cluster and then roll-up and move in-board before leaving the wing surface. It should be pointed out, however, that the outer grids are relatively coarse compared to the viscous grids near the wing surface. Therefore the sharp gradients of the tip-vortex decrease in the downstream wake [10] and the accompanying vorticity contours at different downstream locations shows that the vortex is getting diffused.

Figure 3 shows a farfield view of the tip vortex for the ONERA swept tip blade at $M_{\infty} = 0.85$ and $\alpha = 5$ degrees. Here again, the fluid particles from the lower surface move around the tip to the upper surface and get braided into the particles here and lift-off from the surface and roll in-board in the downstream wake. In this as well as the previous case, the tip vortex stays distinctly above the wake vortex sheet. The vorticity contours shown in this picture at several X-locations in the wake indicate the distorting shapes and
diminishing magnitudes of these due to the grid.

In conclusion, a reasonably good definition of the formation and roll-up of the tip vortex is simulated numerically using a multi-block zonal approach for solving Euler/Navier-Stokes equations.

References


<table>
<thead>
<tr>
<th>UPPER</th>
<th>LOWER</th>
<th>$M_{\infty}$</th>
<th>$\alpha$, deg</th>
</tr>
</thead>
<tbody>
<tr>
<td>△</td>
<td>▽</td>
<td>0.90</td>
<td>5.0</td>
</tr>
<tr>
<td>△</td>
<td>▽</td>
<td>0.902</td>
<td>5.0</td>
</tr>
</tbody>
</table>

Fig. 1. Typical surface pressure distributions for Wing C. $M_{\infty} = 0.9$, $\alpha = 5^\circ$, and $Re = 6.8$ million.
Fig. 2. Farfield view of the tip vortex for Wing C. The vorticity contours at three x-locations in the wake region show the magnitudes and the shapes of the tip vortex.

Fig. 3. Tip vortex for the ONERA swept-tip wing. The vorticity contours show the magnitudes and the shapes of the tip vortex at several x-stations.
**Abstract**

Thin layer Navier-Stokes and Euler equations are numerically solved using a multi-block zonal approach to simulate the formation and roll-up of tip vortices of wings in subsonic and transonic flows. Several wing planforms have been considered to examine the influence of tip-cap shape, planform geometry and free stream Mach number on the formation process. A good definition of the formation and qualitative roll-up of tip vortices has been achieved.