ANALYSIS OF COMPLEX AERODYNAMIC CONFIGURATIONS WITH A ZONAL NAVIER-STOKES SOLVER

Turgut Serkan ŞEN¹, Bülent KORKEM²
TAI-TUSAŞ Aerospace Industries Inc., P.O. Box 18, 06692, Ankara, TÜRKİYE

Beytullah TEMEL³
TÜBİTAK-SAGE, P.K. 16, 06261, Ankara, TÜRKİYE

M. Şerif KAVISAOĞLU⁴
Middle East Technical University, Dept. of Aeronautical Engineering
06531, Ankara, TÜRKİYE

Abstract

The flow fields around different aerodynamic bodies have been solved. The steady state solutions were obtained by using two and three dimensional, multi-zone, time marching, Reynolds averaged Thin Layer Navier-Stokes codes with Beam-Warming finite difference implicit algorithm and Baldwin-Lomax, Cebeçi-Smith and Johnson-King turbulence models. The matrix solution is carried out using a diagonally dominant LU-ADI factorization algorithm. The computations around various geometries were performed at well known test cases. The results were compared with experimental and other numerical results.

Introduction

In the last decade, Computational Fluid Dynamics (CFD) has become a part of the aerodynamic design process as a result of the improvements in numerical algorithms, grid generation, computer processing power and post processing graphics. Even though CFD is used for wide variety of applications, there are still many obstacles to overcome. Since the concept of body fitted coordinate transformation has been introduced, structured grid methods became the most popular CFD methods. But increasing demand for more realistic and complex applications forced researchers to seek out alternatives to overcome the inadequacy of these methods. As the flow field becomes more and more complicated, even the sophisticated grid generation codes can not supply reasonable meshes for the simulation. On the other hand, computer speed and memory limitations may restrict the dynamic data storage. Multi block grid generation and implementation to the flow solvers is a good alternative which can overcome the difficulties encountered in a single block solution.

In the present study, a single block thin layer Navier-Stokes code has been modified to a zonal solver. In the grid generation, whole domain is divided into non overlapping, geometrically simpler sub-regions or zones.

Solution Method

The governing equations are written in curvilinear coordinates with strong conservation form for shock capturing purposes,

\[ Q_t + E_x + F_y + G_z = \frac{k}{Re} \left( S_1 + S_2 + S_3 \right) \]  (1)

where \( k = \begin{cases} 0 & \text{for inviscid} \\ 1 & \text{for viscous} \end{cases} \)

Here \( Q \) is the dependent variable vector, \( J \) is the transformation Jacobian,

\[ Q = J^{-1} \begin{bmatrix} \rho, \rho u, \rho v, \rho w, e \end{bmatrix}^T \]  (2)

and \( E, F, G \) are the inviscid flux vectors, \( S_1, S_2, S_3 \) are the thin layer viscous flux vectors in each curvilinear direction, and \( Re \) is the Reynolds number.

The matrix solution is carried out using a diagonally dominant Lower-Upper bi-diagonal ADI (LU-ADI) [1] factorization algorithm. This algorithm requires less memory storage as well as less computing time which has made the practical computations possible. The present LU-ADI factorization simplifies inversion works for the left hand side operators of the most commonly used Beam-Warming [2] scheme. Each ADI operator is decomposed to the product of lower and upper bi-diagonal matrices. The explicit part of the delta form equation is the same as that of Beam-Warming scheme where central difference is used. The method is first order accurate in time and second order accurate in space. Convergence, stability and smoothness of the solution may depend on the implicit and explicit smoothing factors and CFL number;

¹Design Engineer, also Ph.D. student at M.E.T.U.
Aeronautical Eng. Dept., Ankara, Türkiye
²Sr. Design Engineer, also M.S. student at M.E.T.U.
Aeronautical Eng. Dept., Ankara, Türkiye
³Research Engineer, also M.S. student at M.E.T.U.
Aeronautical Eng. Dept., Ankara, Türkiye
⁴Associate Professor, also consultant at TÜBİTAK-SAGE, Ankara, Türkiye, Member AIAA

Copyright © 1996 by the AIAA and ICAS. All rights reserved.
\[ CFL = \frac{\Delta t \cdot \sigma_{\text{max}}}{\min(\Delta x, \Delta y, \Delta z)} \]  

(3)

where \( \sigma_{\text{max}} \) is the maximum eigen value.

Multiblock Technique

The idea of multiple zone technique is to reduce a geometrically complex region into several smaller and more manageable regions. Grids can be independently generated for each zone. In the present zonal scheme, the flow field is divided into conceivable structures provided that cell to cell matching on block boundaries is maintained. The requirement of cell matching is chosen to eliminate complex interpolation processes at the matched faces of two adjacent zones [3]. The grid points were clustered near the surface and near intersected faces of neighbor zones. Also in a direction normal to surface, the grid points have to be clustered at the surface in order to resolve turbulent boundary layer.

Results

The selected configurations are; NACA0012 airfoil, single flapped airfoil (NLR 7501), ONERA-M6 wing, sharp-edged delta-rectangular wing, a wing-body configuration, and a wrap-around finned missile. The computations for these geometries were performed in a wide range of angles of attack and free stream Mach numbers. The solutions were compared with available experimental and numerical results. The numerical computations were found to agree well with these results.

NACA0012 Airfoil

The NACA0012 airfoil computations were performed at three different flight conditions:

A. \( M=0.550, \alpha=8.34 \, \text{deg} \).
B. \( M=0.700, \alpha=1.49 \, \text{deg} \).
C. \( M=0.799, \alpha=2.26 \, \text{deg} \).

For all cases the Reynolds number is \( 9 \times 10^6 \). A single zone hyperbolic C-type grid was used with the dimensions of 161 \times 61. Three different turbulence models were used (Baldwin-Lomax [4], Cebeci-Smith [6] and Johnson-King [7]) to simulate the turbulence effects. The BL and CS turbulence models are equilibrium models and the JK model is half-equation non-equilibrium model. The aerodynamic coefficients of computations were compared to the experiment [8] in Table 1. In the first two solutions, the three turbulence models agree well with the experiment. But in the third case, the equilibrium turbulence models (BL and CS) predict a shock wave that are too far downstream and very weak separation. On the other hand, the non-equilibrium model (JK) predicts the shock wave location and the shock induced separation well. A sample grid and computed pressure coefficients with BL, CS and JK turbulence models compared to experimental data, were presented in Figure 1a-b.

<table>
<thead>
<tr>
<th>Mach</th>
<th>( \alpha^o )</th>
<th>Computation</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.550</td>
<td>8.34</td>
<td>0.9727</td>
<td>0.0419</td>
</tr>
<tr>
<td>0.700</td>
<td>1.49</td>
<td>0.2450</td>
<td>0.0091</td>
</tr>
<tr>
<td>0.799</td>
<td>2.26</td>
<td>0.3715</td>
<td>0.0317</td>
</tr>
</tbody>
</table>

TABLE 1 - Comparison of Aerodynamic Coefficients

FIGURE 1a - \( C_p \) Distributions for Cases A and B

FIGURE 1b - \( C_p \) Distributions for Case C and C-type Grid
NLR 7301 Single Flapped Airfoil

This case is a low speed multi-element airfoil computation test case. The basic airfoil section is the NLR7301 with trailing edge flap. The flap chord is 32% of the main airfoil and is deflected 20 degrees down. The flap gap is 2.6% of the main airfoil chord and the Reynolds number based on chord length is $2.51 \times 10^6$. The computation was performed at $M=0.185$, $\alpha=6$ degrees with the BL turbulence model only. The computational grid system shown in Figure 2, is composed of 5 blocks with the dimensions given in Table 2 (streamwise and normal directions respectively). The Mach contours, velocity vectors at leading and trailing edges, streamline patterns, and surface pressure coefficient distribution were shown through Figure 3 to Figure 5. The experimental data is taken from Reference [9]. The difference between computation and experiment is due to lack of far field corrections at the outer boundary.

<table>
<thead>
<tr>
<th>Block #</th>
<th>$\zeta$ direction</th>
<th>$\eta$ direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>101</td>
<td>49</td>
</tr>
<tr>
<td>2</td>
<td>115</td>
<td>25</td>
</tr>
<tr>
<td>3</td>
<td>65</td>
<td>25</td>
</tr>
<tr>
<td>4</td>
<td>51</td>
<td>49</td>
</tr>
<tr>
<td>5</td>
<td>51</td>
<td>25</td>
</tr>
</tbody>
</table>

TABLE 2 - Grid Dimensions in Each Zone

Sharp-Edged Wing

The sharp-edged wing geometry was chosen from a NASA wind tunnel test case to verify present zonal code. The details of this geometry is given in Figure 6. The body fitted computational multi zone grid is shown in Figure 7. The calculations were performed at a fixed Mach number of 2.16 and Reynolds number of $1.33 \times 10^6$, and angles of attack $\alpha = 1.3, 6.3, 11.3$ degrees with a total mesh size of 262564 points (see Table 3). A comparison of computed and experimental spanwise surface pressure distributions at $M_a=2.16$ and $\alpha = 6.3\degree$ was presented in Figure 8. Pressure coefficient contours were given in Figure 9. Also the lift coefficients are compared with the available experimental [10], and numerical [11] results in Table 4.

FIGURE 4 - Mach Contours Around NLR 7301

FIGURE 5 - Streamline Pattern and $C_p$ Distribution over NLR 7301

FIGURE 2 - Grid Around NLR 7301

FIGURE 3 - Velocity Vectors over NLR 7301

FIGURE 6 - Sharp-Edged Delta-Rectangular Wing

2530
FIGURE 7 - Upper Half of Computational Grid

FIGURE 8a - Three Chordwise Locations

FIGURE 8b - $C_p$ Distributions at $x/L=0.375$ and $\alpha=6.3$

FIGURE 8c - $C_p$ Distributions at $x/L=0.625$ and $\alpha=6.3$

FIGURE 8d - $C_p$ Distributions at $x/L=0.844$ and $\alpha=6.3$

FIGURE 9 - $C_p$ Contours at $M=2.16$ and $\alpha=6.3$

<table>
<thead>
<tr>
<th>Block #</th>
<th>$\xi$ direction</th>
<th>$\eta$ direction</th>
<th>$\zeta$ direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 and 5</td>
<td>25</td>
<td>23</td>
<td>41</td>
</tr>
<tr>
<td>2, 4, 6 and 8</td>
<td>37</td>
<td>23</td>
<td>41</td>
</tr>
<tr>
<td>3 and 7</td>
<td>37</td>
<td>25</td>
<td>41</td>
</tr>
</tbody>
</table>

TABLE 3 - Grid Dimensions in Each Zone

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.3</td>
<td>0.033</td>
<td>0.030</td>
<td>0.034</td>
</tr>
<tr>
<td>6.3</td>
<td>0.168</td>
<td>0.164</td>
<td>0.168</td>
</tr>
<tr>
<td>11.3</td>
<td>0.305</td>
<td>0.308</td>
<td>0.306</td>
</tr>
</tbody>
</table>

TABLE 4 - $C_L$ Comparison for Sharp-Edged Wing
ONERA-M6 Wing

The ONERA-M6 wing computations were performed at three different test case conditions:

A. \( M=0.84, \alpha=3.06 \) deg.
B. \( M=0.84, \alpha=5.06 \) deg.
C. \( M=0.84, \alpha=6.06 \) deg.

For all cases the Reynolds number is \( 11.7 \times 10^6 \). The first case is known as the attached flow case and was computed with BL and JK turbulence models, the \( C_p \) distributions along span were presented for both models, but the particle traces and the surface pressure distributions were only presented for the BL turbulence model. The other two cases are separated flow cases and were computed using only the BL turbulence model. The computational grid system shown in Figure 10 is a single block mesh which is hyperbolic C-type in streamwise and normal directions and algebraic O-type in spanwise direction, and dimensioned as \( 181 \times 51 \times 41 \) in streamwise, spanwise and normal directions respectively. The distance to the outer boundary is 6 root chords and the first point off the body is \( 4 \times 10^{-3} \). The experimental data are taken from references [12] and [13].

![FIGURE 10 - Grid Generated Around ONERA-M6 Wing](image)

In Figures 12 and 13, the experimental and computational particle traces and the surface pressure distributions were presented. The particle traces show that there separation for this case. The surface pressure distribution locate the shock wave on the wing.

![FIGURE 11 - \( C_p \) Distributions at Different Spanwise Locations on ONERA-M6 Wing for Case A](image)

![FIGURE 12 - Experimental Particle Traces and Surface Pressure Distribution on ONERA-M6 Wing for Case A](image)

![FIGURE 13 - Computational Particle Traces and Surface Pressure Distribution on ONERA-M6 Wing for Case A](image)
The mild separation case was presented as only particle traces, Figure 14, and this figure also has the tendency as is on the experimental data that there exists shock induced separation towards the wing tip.

![Figure 14: Experimental and Computational Particle Traces on ONERA-M6 Wing for Case B](image)

The separated flow case was also computed with BL turbulence model only and the Figure 15 shows the lambda shock over the wing and Figure 16 demonstrates the shock induced separation zone. These are also in agreement with other computational and experimental data in literature.

![Figure 15: Computational Shock Location on ONERA-M6 Wing for Case C](image)

Wing-Body Configuration

As another geometry, a wing-body configuration was selected. In order to compute the flow field around this geometry, the multi-block code was run both in inviscid and viscous mode. The comparisons were made with wind tunnel tests which contain selected data of surface pressure distributions. The experiments were done in RAE (Royal Aircraft Establishment in United Kingdom) 8 ft x 6 ft transonic wind tunnel on RAE research wing "A" in combination with an axisymmetric body [5]. Wing "A" is a wing of simple planform with no dihedral or twist and is based on a uncambered "RAE 101" airfoil. In the experiment, the test Reynolds number has been kept at \(1 \times 10^6\) based on the geometric mean chord. The complete geometry definition of wing-body configuration is shown in Figure 17. In the experiment, the tunnel wall interference was relatively small. Although the tests have been made on a complete model of a size, the computations were performed by using only one-fourth of the configuration at zero angle of attack and one half of it at a non zero angle of attack. These were done due to mirror symmetry. The wake was modeled by extending the boattail up to \(X = 34.95\) inch, turning the grid line parallel to the model axis and extending it to the downstream boundary. The computational grid was combined of five blocks as shown in Figure 18. For each block, the size of grid is presented in Table 5.

![Figure 16: Computational Particle Traces on ONERA-M6 Wing for Case C](image)

![Figure 17: Geometry of Wing-Body Configuration](image)

The computations were done at Mach numbers 0.4 and 0.8 with angles of attack, \(\alpha = 0.0^\circ\) and \(\alpha = 2.0^\circ\). For zero angle of attack cases, the computational time for one converged solution is approximately sixty hours in DEC-Alpha 3000/800 workstation with 110 Mflops speed. If there is an angle of attack in the computation then the CPU time is doubled due to increase in the size of the grid. The comparison of computed and experimental surface pressure coefficients are presented from Figure 19 to 23.
### TABLE 5 - Grid Dimensions in Each Zone

<table>
<thead>
<tr>
<th>Block #</th>
<th>(\xi) direction</th>
<th>(\eta) direction</th>
<th>(\zeta) direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25</td>
<td>21</td>
<td>65</td>
</tr>
<tr>
<td>2</td>
<td>17</td>
<td>21</td>
<td>65</td>
</tr>
<tr>
<td>3</td>
<td>41</td>
<td>21</td>
<td>65</td>
</tr>
<tr>
<td>4</td>
<td>25</td>
<td>21</td>
<td>65</td>
</tr>
<tr>
<td>5</td>
<td>41</td>
<td>21</td>
<td>65</td>
</tr>
</tbody>
</table>

**FIGURE 21a - Pressure Coefficient Distribution on the Wing at Spanwise Location Y/S=0.25 for \(M_\infty = 0.4\)**

**FIGURE 18 - Overall View of Computational Domain**

**FIGURE 19 - Surface Pressure Coefficient Contours at \(M_\infty = 0.4, \alpha = 0.0^\circ\)**

**FIGURE 20 - Pressure Coefficient Distribution on Wing-Body Junction at \(M_\infty = 0.4\)**

**FIGURE 21b - Pressure Coefficient Distribution on the Wing at Spanwise Location Y/S=0.4 for \(M_\infty = 0.4\)**

**FIGURE 22 - Pressure Coefficient Distribution on Wing-Body Junction at \(M_\infty = 0.8\)**
A sample computational grid around one fourth of the missile is shown in Figure 25. As the flow symmetry occurs at zero angle of attack, the computations were done by using a quarter of the full geometry. For this domain, the number of total grid size is 107×41×21. The Reynolds number varied from 4.6×10⁹ to 9.0×10⁹ based on the diameter in the Mach number range of 0.8 to 2.5. The elapsed time for a converged solution is approximately 24 hours in DEC-Alpha 3000/800 work station. The aerodynamic coefficients were compared with available experimental and computational results. The preliminary results for forebody axial force coefficient, $C_x$, and rolling moment coefficient, $C_M$, given in Table 6 are in good agreement with the experimental data [14]. The Mach contours for two different free stream Mach numbers were given in Figures 26 and 27.
Validating the code by comparing the numerical solutions with experimental data is necessary in order to make the CFD method a truly complementary engineering tool to wind tunnel testing. In this study the code validation was also done. For this purpose, various test cases were examined and were compared with different experimental and computational results from literature. The computations were found to agree well with the experimental and computational data.

The major disadvantage of the present method is that the grid generation must be done providing continuous block interfaces. This problem will be solved in order to get a more flexible design tool.

References


[5] Tradegold, D. A., Jones, A. F., Wilson, K. H., "Pressure Distribution in the RAE 8ftx6ft Transonic Wind Tunnel on RAE Wing 'A' in Combination with an Axisymmetric Body at Mach Numbers 0.4, 0.8 and 0.9," Royal Aircraft Establishment, Farnbrough, Hants, United Kingdom.


Conclusion

From this study, one can conclude that the multi zone capability of the code is essential in solving flow problems involving complex and realistic geometries. Multi block approach also overcomes the difficulties encountered by the hardware limitations such as data storage and memory requirement.


