# Thin-Layer and Full Navier-Stokes Calculations for Turbulent Supersonic Flow Over a Cone at an Angle of Attack 

Crawford F. Smith and Steve D. Podleski<br>Sverdrup Technology, Inc.<br>Lewis Research Center Group<br>Brook Park, Ohio



Prepared for
Lewis Research Center
Under Contract NAS3-25266

## TABLE OF CONTENTS

Page
Table of Contents ..... ii
List of Figures ..... iii
List of Tables ..... iv
Nomenclature ..... v
Summary ..... vi
1.0 Introduction ..... 1
2.0 Description of Test Data ..... 2
3.0 Numerical Modeling ..... 4
3.1 CFL3D code ..... 4
3.2 PARC3D code ..... 5
3.3 Grid ..... 5
3.4 Boundary Conditions ..... 6
4.0 Results ..... 8
4.1 CFL3D Grid Studies ..... 9
4.2 CFL3D Multigrid Studies ..... 12
4.2.1 Multigrid Results: Single-Block Grid ..... 15
4.2.2 Multigrid Results: Three-Block Grid ..... 15
4.3 CFL3D Comparisons with PARC3D ..... 17
5.0 Conclusions ..... 21
6.0 Acknowledgement ..... 22
7.0 References ..... 23
Appendix A: Test Data for Several Cases ..... 58

## LIST OF FIGURES

Page
1 Schematic diagram of experiment ..... 28
2 Course grid ..... 29
3a Grid block boundaries (Cross-Stream View) ..... 30
3b Boundary Conditions (Streamwise View) ..... 31
4a Convergence History: CFL $=1$ ..... 32
4b Convergence History: CFL $=5,1$ ..... 32
5 Mach Number Contours - Streamwise View ..... 33
6 Mach Number Contours - Cross-Streamwise View ..... 34
7 Mach Number Profiles ..... 36
8 Mach Number Profile at $\phi=170^{\circ}$ after additional iterations ..... 38
9 Flow Angle Definition ..... 39
10 Flow Angle Profiles ..... 40
11 Surface Static Pressure Distribution ..... 42
12 Velocity Profiles ..... 43
13 Convergence Histories ..... 44
14 Mach Number Profiles ..... 45
15 Mach Number Profiles at $\phi=170^{\circ}$ ..... 46
16 Mach Number Profiles ..... 47
17 Flow Angle Profiles ..... 49
18 Surface Static Pressure Distribution ..... 51
19 Mach Number Profiles ..... 52
20 Flow Angle Profiles ..... 54
21 Surface Static Pressure Distributions ..... 56
22 Effects of Length Scale Search ..... 57

## LIST OF TABLES

Page
1 Performance Summary for Single-Block Grid ..... 24
2 Statistics for Multigrid Scheme with Multiblock Grid ..... 25
3 Solution Statistics for Cray Y-MP Computer (Single-Block grid) ..... 26
4 Solution Statistics for Cray Y-MP Computer (3-Block grid) ..... 27

|  | Nomenclature |
| :---: | :---: |
| CFL | $=$ Courand-Friedrichs-Lewy Number |
| $C_{p}$ | $=\left(P_{s}-P_{\infty}\right) / .7 P_{\infty} M_{\infty}^{2}$ |
| $C_{p_{c}}$ | $=\left(C_{p}-C_{p_{0}}\right) /\left(\sin \theta_{c}\right)^{2}$ |
| $C_{p 0}$ | $=C_{p}$ at $\alpha=0^{\circ}$ (inviscid) |
| H | $=$ Height Above Cone Surface, inches |
| $l$ | $=$ Cone Length, inches |
| M | = Local Mach Number |
| $M_{\infty}$ | = Free-Stream Mach Number |
| $P_{s}$ | = Local Static Pressure |
| $P_{s_{\infty}}$ | = Static Pressure at Upstream Reference Station |
| $P_{t_{\infty}}$ | = Total Pressure at Upstream Reference Station |
| $T_{t_{\infty}}$ | = Total Temperature at Upstream Reference Station |
| $u$ | = Local Axial Velocity |
| $u^{+}$ | $=\frac{u}{\sqrt{\tau_{w} / P}}$ |
| $V_{\infty}$ | $=$ Free-Stream Velocity, $\mathrm{ft} / \mathrm{sec}$ |
| $x$ | $=$ Axial Distance, inches |
| $y$ | $=$ Normal Distance from Surface |
| $y^{+}$ | $=y \frac{\sqrt{\tau_{w} / \rho}}{\nu}$ |
| $\alpha$ | $=$ Angle of Attack |
| $\nu$ | = Viscosity |
| $\omega$ | $=$ Flow Angle Relative to Conical Ray |
| $\phi$ | = Circumferential Angular Position |
| $\rho$ | $=$ Density |
| $\tau_{w}$ | = Wall Shear Stress |
| $\theta_{c}$ | $=$ Cone Half Angle |

# THIN-LAYER AND FULL NAVIER-STOKES CALCULATIONS FOR TURBULENT SUPERSONIC FLOW OVER A CONE 

AT AN ANGLE OF ATTACK

Crawford F. Smith and Steve D. Podleski<br>Sverdrup Technology, Inc.<br>Lewis Research Center Group<br>Brook Park, Ohio 44142

## Summary

The proper use of a computational fluid dynamics code requires a good understanding of the particular code being applied. In this report the application of CFL3D, a thin-layer Navier-Stokes code, is compared with the results obtained from PARC3D, a full Navier-Stokes code. In order to gain an understanding of the use of this code, a simple problem was chosen in which several key features of the code could be exercised. The problem chosen is a cone in supersonic flow at an angle of attack. The issues of grid resolution, grid blocking, and multigridding with CFL3D are explored. The use of multigridding resulted in a significant reduction in the computational time required to solve the problem. Solutions obtained are compared with the results using the full Navier-Stokes equations solver PARC3D. The results obtained with the CFL3D code compared well with the PARC3D solutions.

### 1.0 Introduction

The analysis of the flow in an aircraft inlet, such as the F-18 inlet, at subsonic speeds and high angles of attack requires the inclusion of the external flow about the forebody, Leading Edge Extension (LEX), and wing in order to account for upstream disturbances such as flow separation and shed vortices which might be entrained by the inlet flow.

The numerical solution of this problem is very difficult and requires large amounts of computational time. For adequate geometry resolution, grid blocking is necessary. The use of multigridding can sometimes significantly decrease the amount of computational time required to obtain a converged solution. In addition, proper grid resolution is needed to capture the details of a very complex flow field.

A computational fluid dynamics (CFD) code that has been used to address the problem of forebodies at high angles of attack is the CFL3D code [Ref. 1]. This code has been developed at the NASA Langley Research Center and solves the thin-layer Navier-Stokes (TLNS) equations. Due to these forebody applications, this code appears to have the capability to address the problem of determining the flow field within an inlet of an aircraft at high angles of attack. This code also has multigrid capabilities.

In order to gain some experience in the use and understanding of the code, a simple problem is chosen, which is the prediction of the flow about a cone in
supersonic flow at an angle of attack. This configuration was chosen because the geometry of a cone is simple and a detailed data base is available which includes off-body measurements of velocity and flow angles. Although the interest is in predicting subsonic, vortical flows, the physics of the vortex development is the same for subsonic and supersonic flows. In addition, the use of multigridding to accelerate the rate of convergence of the numerical solutions is examined using CFL3D (Version 2.1). The solutions obtained are compared with the PARC3D code (NASA Lewis Version, Ref. 2) which solves the full Navier-Stokes (FNS) equations. Solutions were obtained using a Cray Y-MP computer with compiler version 4.0.3.

This report is divided into several sections. A brief description of the experiment and data is presented, followed by a discussion of the CFL3D and PARC3D codes as well as the computational grids and boundary conditions. The results obtained with the CFL3D code are presented for coarse and fine grids (one-block and three-block grids respectively), along with a discussion of the performance of the multigrid algorithms. The results obtained with the PARC3D code using the same grids are compared with the CFL3D results. The report ends by presenting some conclusions.

### 2.0 Description of the Data

For comparison with the numerical results, Rainbird obtained useful data of surface static pressures [Ref. 3] for a cone with an 18.0 inch base and a $12.5^{\circ}$ half angle. The off-body data, which includes the Mach number and flow angle profiles was obtained by personal communications with Rainbird, are compared
with the numerical results. To the best of the knowledge of the authors the full details of this data set have not been published. However, data for the present case and several others are contained in Appendix A. Permission to publish the data in this report was granted by the Director General of the Institute for Aerospace Research, Ottawa, Canada. Rainbird [Ref. 3] indicates that no boundary layer trip was used due to the high free-stream turbulence present in the wind tunnel. He assumes that transition occurs very close to the cone apex (less than $10 \%$ of the cone length).

A 3-hole probe was used to survey the flow field. The probe was kept turned into the local mean flow direction and thus enabled measurements of local flow angle and pitot pressures. Surveys were conducted at an axial position of $85 \%$ cone length. The upstream Mach number was 1.8 , the angle of attack was $15.75^{\circ}$ and the flow was turbulent. A diagram illustrating this test is shown in Figure 1.

### 3.0 Numerical Modeling

### 3.1 CFL3D Code

The CFL3D code [Ref. 1] solves the thin-layer Navier-Stokes equations using upwind differencing with a total variation diminishing (TVD) scheme and employs the Baldwin-Lomax turbulence model [Ref. 4]. The TVD scheme eliminates oscillations due to dispersion errors introduced by the higher order terms in the upwind differences by shutting off these higher order terms in regions of large flow oscillations. Various options are available for TVD schemes, flux vector-differencing, and upwinding accuracy. The options recommended below provided the best results and are used in the solutions presented in this report. These options include the min-mod flux limiter for the TVD scheme, the Roe flux difference splitting scheme and third order accurate upwinding. The threefactor approximate factorization scheme is used to obtain a block tridiagonal system of equations. For the Roe scheme, the equations are diagonalized to obtain a scalar tridiagonal system of equations that yields a more efficient solver. A conservative scheme is employed to transfer information between grid blocks [Ref. 5] and multigridding is also available to accelerate the convergence of the solution [Ref. 6].

### 3.2 PARC3D Code

The PARC3D code [Ref. 2] solves the full three-dimensional Reynoldsaveraged Navier-Stokes equations in strong conservation form using the Beam and Warming approximate factorization scheme, to obtain a block tridiagonal system of equations. Pulliam's scalar pentadiagonal transformation provides for an efficient solver. Like CFL3D, the code uses the Baldwin-Lomax turbulence model [Ref. 4]. Its implicit scheme uses central differencing with artificial dissipation to eliminate oscillations in the solution associated with the use of central differences. Trilinear interpolation [Ref. 7] is used to transfer information at the grid block interfaces when a multiblock grid is used.

### 3.3 Grid

The effects of grid refinement on the numerical solution were explored using two different grids. The grids were algebraically generated with the INGRID3D code [Ref. 8], and clustered near the surface using hyperbolic stretching functions.

The first grid shown in Figure 2 had dimensions of $29 \times 37 \times 61$ points in the streamwise, circumferential, and radial directions, respectively. The first axial station is located ahead of the cone due to concerns about locating the inflow boundary on the cone. The typical value for $y^{+}$for the first off-surface grid point is approximately 8. The grid is spaced uniformly in the circumferential direction at $5^{\circ}$ intervals and is packed towards the apex of the cone.

The second grid, not shown due to resolution problems in reproducing the plot, consisted of three blocks with dimensions of $33 \times 73 \times 73,33 \times 41 \times 73$,
and $33 \times 41 \times 33$ in the streamwise, circumferential, and radial directions for Blocks 1, 2 and 3, respectively. The grid block numbering is shown in Figure 3a. For this grid, the first axial station was located at the cone apex and results obtained were not affected by placing the first point at the cone apex. The use of three grid blocks was chosen in order to resolve the leeward side vortex using Block 1, which was much denser than Blocks 2 or 3 . Grid blocks 2 and 3 have a grid distribution similar to that of the single block grid. A value of 1 was obtained for the typical $y^{+}$for the first grid point off the surface in Block one; in Block 2 the value was 8 . The grid is spaced equally at $1^{\circ}$ increments in Block 1 and was packed towards the Block 1 interface for Block 2. Note that for use in CFL3D, the grids are face-to-face while for PARC3D, the grids overlap in order to accommodate the linear interpolation scheme used at the grid block interfaces. These interfaces are non-contiguous for both codes.

### 3.4 Boundary Conditions

The upstream and outer radial boundary conditions are held fixed at supersonic free-stream conditions. The flow properties are extrapolated for supersonic flow at the downstream exit. At the surface, no-slip, isothermal conditions are specified. Slip wall boundary conditions are used along the planes of symmetry. These boundary conditions are illustrated in Figure 3a and 3b.

Isothermal conditions were used because the experiment used a blow-down wind tunnel in which the surface temperature variation was less than $5^{\circ}$ Fahrenheit with a run duration of 20 to 30 seconds. Calculations made using adiabatic conditions produced the same Mach number and flow angle profiles as the cal-
culations made with isothermal wall conditions.

### 4.0 Results

In this section several major results will be discussed. The first will deal with the effects of grid resolution using CFL3D, and the second the use of multigridding with CFL3D will then be discussed. Following this dicussion of multigridding, comparisons of PARC3D solutions with those obtained with CFL3D will be presented.

All of the results reported are derived from PLOT3D format files, which use node-centered data. Since the CFL3D code is a finite volume code, the flow field is determined at the cell centers of the computational grid and not at the grid nodes, as with finite difference codes such as PARC3D. The PLOT3D flow and grid files were obtained from the CFL3D code by averaging the adjacent cell centers of the grid and using these values at the grid nodes.

As a preliminary check on the functionality of the CFL3D code and to ensure proper problem simulation, a laminar case was studied. The free-steam conditions were the same as for the turbulent cases ( $M_{\infty}=1.8, \alpha=15.75^{\circ}$ ). The residuals associated with this solution dropped 6 orders of magnitude in 17,000 iterations and continued to drop. The flow field exhibited a much larger cross-flow separation than the turbulent calculations, which is consistent for laminar flows.

Two criteria were used to evaluate the convergence of the turbulent solutions which are presented in this report. The first criterion was when the residuals reached a constant level, which is typical behavior for turbulent solutions. A plot of the density residuals for a constant CFL number of 1 and then a CFL
number adjusted from 5 to 1 are shown in Figure 4. The residual values reaching a constant level for a CFL number of 1 are shown in Figure 4a. A rapid drop and rise in the residuals for the CFL number of 5 , seen in Figure 4a, is due to the code failing to update local time steps after each iteration until the solution is restarted. This behavior is not apparent for a CFL number of 1, (see Figure $4 a$ ), but may be attributable to the solution nearing stability limits at the higher CFL number. It did not, however, appear to have an adverse effect on the final results. The other criterion was when the change in the boundary layer profiles in the vortex region reached a minimum. However, truly steady solutions within the vortex region were not obtained. Further discussion of convergence issues are presented in the section dealing with multigrid solutions.

### 4.1 CFL3D Grid Studies

In Figure 5, Mach number contours are presented in the plane of symmetry. The single and 3-Block grid solutions are very similar. The shock may be slightly sharper (closer contours) in the 3-Block grid results due to a few more points added in the radial direction. There is a small expansion fan along the leeward side of the cone. Mach number contours in the cross-plane are shown in Figures 6a and 6b. Again, the shock appears slightly sharper in the 3 -Block grid solution. However, there is a dramatic change in resolution of the leeward side vortex as can be seen in the enlarged views of this region shown in Figures 6c and 6d. The single block grid does not indicate the vortex presence with the exception of a rapidly thickening boundary layer. In contrast, 3-Block grid resolved the vortex very well with a distinct region of recirculating flow.

The circumferential positions around the cone are defined as $0^{\circ}$ on the windward side and $180^{\circ}$ along the leeward side. Mach number profiles for several circumferential stations are shown Figure 7. These are taken at $85 \%$ of the cone length. Along the windward side of the cone, $\left(0^{\circ}\right.$ to $\left.90^{\circ}\right)$, the single grid and 3 Block grid solutions agree very well with each other and the data. In this region the boundary layers are very thin and well-behaved. Along the leeward side of the cone the boundary layers begin to thicken and separate in the cross-flow direction at approximately $155^{\circ}$.

As can be seen, the 3 -Block grid solutions provide much better agreement with the data than the single block grid. In particular, at $170^{\circ}$, the vortex is only resolved with the 3 -Block grid. It should be noted that the increase in the number of grid points solely in this region (block 1) did not improve this result very much and this result is not presented. The reduction of the $y^{+}$value of the first grid point from 8 to 1 in Block 1 was necessary to provide the results shown at $170^{\circ}$. The predicted Mach number profile in the vortex region for $170^{\circ}$ (shown in Figure 8) indicates improved comparison with the data after the solution was iterated an additional 10,000 times. The remaining discrepancies between the predictions and data may be due to the turbulence model not accounting for the vortical flow adequately. This may also be a contributing factor to the discrepancies between the predicted and measured Mach number profiles at $180^{\circ}$.

The flow angles are defined in Figure 9, and those predicted with the single and 3 -Block grids are compared with data in Figure 10. From the wind-
ward plane of symmetry $\left(0^{\circ}\right)$ to $145^{\circ}$, both solutions provided similar results. However, in the vortical region from $155^{\circ}$, the 3-Block grid provides improved comparisons with the data.

Surface static pressures at $85 \%$ of cone length calculated with both grids are compared with data in Figure 11. Very little improvement is shown with the 3-Block grid along the windward side of the cone although both grids provided good results along the leeward side of the cone. Increasing the number of grid points and density of the grid in the radial direction provided for some improved shock resolution but offered little improvement in the surface static pressure calculations. The good agreement along the leeward side of the cone may indicate that inadequate shock resolution on the windward side of the cone is the contributing factor in the discrepancies. Some of the discrepancies are due to using the difference of two static pressure coefficients in obtaining the coefficients presented. Decreasing the $y^{+}$value for the first off-body grid point in Block 2 may improve these results. Although, since the boundary layer profiles are in very good agreement with the data, there may be no further improvement.

Rainbird noted that there was an error in the surface static pressure measurements due to the windward boundary layer thickness being only twice the diameter of the static pressure holes (personal communications). This error diminishes as the boundary layer thickens along the leeward side of the cone. He indicated that.the correction to the surface static pressure data was never implemented because the error was a function of the constantly varying boundary layer thickness. This error would account for a small amount of the discrepancy
between the calculated and measured surface static pressures along the windward side of the cone. Rainbird also indicated that the model alignment error was within $.1^{\circ}$, therefore, misalignment of the model is probably not an issue.

One check on the grid dependency of a solution is to compare the velocity profiles in unseparated regions with the Law of the Wall. The single and 3Block grid results are shown in Figure 12. Significant improvements were made in the comparisons with the fine grid. The discrepancies in the $0^{\circ}$ station can be attributed to the value of 11 for the $y^{+}$of the first off-body point, which places it out of the viscous sublayer (linear region), making accurate wall shear stress calculations impossible. The grid clustering near the wall was not changed in Block 2 from the clustering used with the single block grid.

### 4.2 CFL3D Multigrid Studies

Multigridding is a process in which solutions obtained on successively coarser grids are used to accelerate the convergence rate for the highest level or finest grid. Each successive lower grid has one-half the number of points as the next higher level grid. Large scale flow features are developed very rapidly with the coarse or lower level grids, while small scale or finer details are resolved with the highest level or finest grid because the effectiveness of multigridding is problem-dependent; the results reported may not be directly applicable to another problem. The results reported in this section are for a single block grid.

The convergence histories for several multigrid and single grid (nonmultigrid) schemes are presented in Figure 13, which shows the density residual. All of these curves terminate at iterations or cycles where the solution was judged
to be converged. Further iterations or cycles, not shown on the plots, did not reduce the residual levels further. One criteria used for convergence was when the residual histories reached the same constant levels. Another criteria used to determine convergence was when the Mach number profiles about the cone exhibited minimal or no change with additional iterations. This convergence criteria is illustrated in the selected Mach number profiles shown in Figure 14. As can be seen, solutions obtained with all of the schemes used are virtually identical with the exception at $\phi=170^{\circ}$. At this location, the possible unsteadiness of the vortex may not allow for a truly steady-state solution. Therefore, the point where minimum changes in this profile occurred was used as the convergence criteria in this region. All solutions were run for 100 iterations in the laminar mode prior to running with turbulence.

The convergence histories of two single grid (non-multigrid schemes) are shown in Figures 13a and 13b. One of these grid schemes used a constant CFL number of 1 (Figure 13a). The other one used a CFL number of 5 for 2700 iterations and then a CFL number of 1 for an additional 1300 iterations (Figure 13b). As can be seen, the use of a high CFL number for the initial calculations reduced the number of iterations required for a converged solution from 8400 iterations to 4000 iterations. The lower CFL number allows the residuals to drop approximately 1.5 orders of magnitude from the level obtained with a CFL number of 5 . The solution obtained when the residual history became constant for a CFL number of 5 is identical to the solution obtained when the CFL number was reduced to 1, as shown in Figure 15a. However, the solution obtained after

4600 iterations with a constant CFL number of 1 is different for the converged solution obtained after 8400 iterations, as shown in Figure 15b.

The residual histories for two three-level multigrid cycles are shown in Figures 13 c and 13d. A single three-level V-cycle consists of obtaining solutions on two successively coarser grids and then using the corrections obtained from the coarse grids to update the solutions on successively finer grids up to the highest level. Each three-level W-cycle consists of obtaining solutions on two successively coarser grids, using the corrections obtained on the coarse grids to update the solution one grid level up, and then return down one grid level. Following these coarse grid solutions, the solutions are updated on successively finer grids up to the highest level.

The V-cycle was first run with a CFL number of 5 for 700 iterations and then with a CFL number of 1 for an additional 600 iterations. As can be seen from Figure 15 c , the solution obtained when the residual history became constant for a CFL number of 5 is the same as that obtained after reducing the CFL number to 1 and iterating until the residuals become constant again. The number of multigrid cycles required to obtain a converged answer was 1300 , as compared with the much larger number of iterations required using the single grids. The W-cycle was run with a CFL number of 3 and a converged solution was obtained with 700 multigrid cycles. This W -cycle result represents approximately half the number of cycles required by the $V$-cycle to obtain the same level of residual drop.

### 4.2.1 Multigrid Results: Single Block Grid

When examining code performance, several factors which are shown in Table 1 must be examined. One important factor is the computational speed in terms of CPU time per cycle per point. As can be seen in Table 1, the single grid (non-multigrid) scheme provides approximately twice the computational speed of either multigrid scheme. This difference is due to the additional solutions required in each multigrid cycle. However, the actual computational time required by these various schemes differed widely. The W-cycle multigrid scheme required the least amount of computational time to reach the same level of convergence as all of the other schemes. In general the multigrid schemes proved to be very effective at reducing computational time for this particular problem. Part of this effectiveness may be attributable to the fact that the flow had only small regions containing three-dimensional effects, specifically the leeward side vortex which occupies a very small portion of the flow field.

### 4.2.2 Multigrid Results: 3-Block Grid

The use of multigridding with a three-block grid was also investigated and the results are discussed in this section. In order to reduce CPU time the number of grid points in the three-block grid was reduced to approximately one-half of the original number of points. The computational speed is summarized in Table 2. As can be seen, the three block grid required significantly more CPU time per cycle per point when used in the multigrid mode than the one block grid results (shown in Table 1). This increase is attributed to the need to transfer information from one block to another in the three-block grid. The convergence
criteria used for this study was the same as that used for the one-block grid multigrid study. Although the multigrid scheme was more costly per cycle, the overall time required to obtain a converged solution was reduced significantly from the time required for the non-multigrid solution. This result is similar to the results obtained using a one-block grid.

### 4.3 CFL3D Comparisons with PARC3D

One concern in using CFL3D is that the code solves the thin-layer NavierStokes equations. In this approximation, the derivatives parallel to a surface are ignored and therefore regions where there are significant streamwise gradients, such as flow reversal, may not be modelled adequately. In order to study the flow in this region, the PARC3D code, which solves the full Navier-Stokes equations was used to obtain solutions for this cone with the same grids that CFL3D used.

The computed Mach number profiles obtained with CFL3D and PARC3D using the coarse grid (single block grid) are shown with the data in Figure 16. The two solutions are identical with the exception of $\phi=180^{\circ}$. It is not clear what is causing these discrepancies at this location. In addition, the flow angles predicted by the two codes agree well with each other and are shown in Figure 17. The surface static pressure distributions predicted by CFL3D and PARC3D also agree well with each other, as can be seen in Figure 18. For this particular case, the thin-layer approximations used in CFL3D do not appear to influence the computed results.

The performance of the two codes for this problem is shown in Table 3. The CFL3D code requires approximately $64 \%$ more memory than PARC3D. The values of the PARC3D storage requirements for thin-layer and full NavierStokes solutions are instantaneous values displayed during program execution. Ideally, the storage requirements are the same. The PARC3D code carries all arrays, regardless of solution mode. The CFL3D code was about $50 \%$ faster than PARC3D in terms of CPU time per iteration per point. However, the
actual CPU time required to obtain a converged answer is difficult to state since the time required depends on the manner in which the problem is solved. For example, running with a large initial CFL number can increase the rate of convergence. Another factor affecting the convergence time required, is running in the laminar mode for a few hundred iterations, which can reduce the amount of computational time significantly. Therefore, several calculations would have to be made with each code in order to determine the optimum approach to solving this particular problem. However, preliminary comparisons indicate that the CFL3D code, when run in the single grid mode, required approximately $42 \%$ less computational time than PARC3D.

The performance comparisons for the PARC3D and CFL3D codes using the three-block grid are shown in Table 4. The PARC3D memory requirements remained about the same, while the CFL3D memory requirement reduced $27 \%$ as compared to the one-block grid. Overall, the CFL3D code required $15 \%$ more memory per point than PARC3D using the three-block grid. One advantage of PARC3D is that it uses only one grid block in core memory at a time, whereas CFL3D keeps all grid blocks in core memory. Therefore, by using additional grid blocks, the core memory required by PARC3D can remain constant as the number of grid points increases which is not the case with CFL3D. In addition, the speed of the codes remained approximately the same as the single block grid case and the ratio of total CPU time required for a converged solution using CFL3D to PARC3D was similar to the single block grid case. The extra grid points in Block 1 of the PARC3D grid are for the required one grid cell overlap
which is not needed with the CFL3D code.

It should be noted that since these comparisons were made, the Cray Y-MP compiler was updated to version 5.0 .2 . . For reasons unknown, the speed of the PARC3D computations increased approximately $20 \%$. No significant changes in the speed of CFL3D were noted.

Solutions were also obtained with the PARC3D code using the same fine grid (3- block) that was used with the CFL3D code. The Mach number and flow angle profiles are shown in Figures 19 and 20, respectively. The results obtained with the two codes are in excellent agreement with each other. The only significant discrepancies occur in the vortex region $\left(\phi=155^{\circ}, 170^{\circ}\right)$. Because the length scales used for the Baldwin-Lomax turbulence models in each code are almost identical throughout this region, differences in the turbulence models are not likely an issue. Differences in the solutions may be attributable to the varying amounts of numerical dissipation present in the solutions. The surface static pressure distributions obtained with the two codes (shown in Figure 21) are in excellent agreement with each other. The discrepancies between the predictions and data have been discussed in a previous section.

In the process of matching of the turbulence model length scales in the two codes, it was found that the search for a length scale was critical to predicting the proper location of the vortex. This effect is illustrated in Figure 22 using the PARC3D results. The Mach number profile in Figure 22(a) is the result of restricting the search for a length scale to the edge of the undisturbed boundary layer. This distance happens to correspond to the center of the vortex since the
vortex is not much larger than the boundary layer. The profile indicates that the predicted vortex position is not the same as the actual position since the predicted Mach number profile is different from the experimental profile. The Mach number profile shown in Figure 22b is the result of restricting the length scale search to the lower edge of the vortex region. This last comparison of the predicted profile to the experimental profile improved with this additional restriction. The restriction to the lower edge of the vortex eliminated the contribution of streamwise vorticity to the turbulent viscosity calculations which are due to the vortex. Only the transverse component attributable to the attached boundary layer was included in the calculation. This result is consistent with the original formulation of the Baldwin-Lomax turbulence model.

The search distance of the turbulent length scale used by the CFL3D code is obtained by using the first $64 \%$ of the grid points from a surface. The percentage of the number of grid points is fixed within the turbulence model subroutines and cannot be adjusted by user inputs, as is available in the PARC3D code.

### 5.0 Conclusions

A major accomplishment of this study was to gain some experience in the use of the CFL3D code. The use of block grids and multigridding in analyzing the flow about the Rainbird cone has been explored successfully and the application of these techniques to a complicated configuration such as the $\mathrm{F}-18$ inlet, forebody, LEX, and wing should be reasonable.

The grid studies indicate that significant improvements in the prediction of details in the flow field can be made by proper selection of grid density and proximity to the surface. A major gain in the agreement of the boundary layer in the vortical flow region was obtained by placing the first grid point off the surface in this region to a distance within a $y^{+}$value of 1 . Outside of this crossflow separation region, improvements were made by increasing the number of grid points without decreasing the distance for the first off-body grid point.

Despite improvements in the boundary layer profiles with increased grid resolution, the predictions of the windward surface static pressure distribution did not improve. A small portion of the discrepancies may be due to experimental errors attributed to the similarity of the windward boundary layer thickness to the diameters of the static pressure holes.

The use of multigridding indicated a significant reduction in the required computational time for this problem. However, the effectiveness of multigridding is problem-dependent. The use of multigridding with multiple grid blocks also showed a significant reduction in CPU time.

The CFL3D results compared well with PARC3D, indicating that for this
problem, the thin-layer approximation is adequate. Further studies with larger recirculating flow regions may be necessary to evaluate properly the range in which the thin-layer approximation is valid. This study indicates that with proper grid resolution, flow field details may be resolved with an algebraic turbulence model and may not require the use of higher order turbulence models. In addition, the use of proper length scales in the algebraic turbulence model is critical to obtaining a good prediction of the vortical flow region.

### 6.0 Acknowledgement

This work was supported by NASA Lewis Research Center under Contract NAS3-25266. The authors would like to recognize the excellent job done by Mrs. Tammy Langhals in the compilation and presentation of the results contained in this report. Also we express our thanks to Kristine Dugas for her editorial review of this document.

### 7.0 References

1. Thomas, J.L., Taylor, S.L., and Anderson, W.K., "Navier-Stokes Computations of Vortical Flows Over Low Aspect Ratio Wings," AIAA Paper 87-0207, Jan. 1987.
2. Cooper, G., and Sirbaugh, J., "The PARC Distinction: A Practical Flow Simulator," AIAA Paper 90-2002, July 1990.
3. Rainbird, W.J. "Turbulent Boundary-Layer Growth and Separation on a Yawed Cone," AIAA Journal, December 1968, pp. 2410-2416.
4. Baldwin, B.S., and Lomax, H., "Thin Layer Approximation and Algebraic Turbulence Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.
5. Thomas, J.L., et al, "A Patched-Grid Algorithm for Complex Configurations Directed Towards the F/A-18 Aircraft," AIAA Paper 89-0121, Jan. 1989.
6. Anderson, W.K., "Implicit Multigrid Algorithms for the Three-Dimensional Flux Split Euler Equations," Ph.D. Dissertation, Mississippi State University, August 1986.
7. Stokes, M.L., and Kneile, K.L., "A Search/Interpolation Algorithm CFD Analysis," Presented at the World Congress on Computational Mechanics, University of Texas, Austin, TX, Sept. 1986.
8. Soni, B.K., "Two- and Three-Dimensional Grid Generation for Internal Flow Applications for Computational Fluid Dynamics," AIAA Paper 851526, July 1985.

Single Grid (Non-multigrid): 16.9
Multi-grid (V-cycle, 3 levels): 29.7
Multi-grid (W-cycle, 3 levels): 33.6

| MULTIGRID CONVERGENCE |  |  | STATISTICS |
| :--- | :---: | :---: | :---: |
| Scheme | CFL | Iterations/Cycles | CPU (hours) |
| Single Grid | 1 | 8400 | 5.8 |
| Single Grid | 5,1 | 4000 | 2.8 |
| Multi-grid, V-cycle | 5,1 | 1300 | 1.8 |
| Multi-grid, W-cycle | 3 | 700 | 1.1 |
| Grid: 149,650 points |  |  |  |

Table 1. Performance Summary for Single-Block Grid
COMPUTATIONAL SPEED
$\left(10^{-6}\right.$ second/cycle/point $)$

Table 3. Solution Statistics for Cray Y-MP Computer (Single-Block Grid)


CFL3D PERFORMANCE
33 X 73 X 73, 33 X 41 X 73, 33 X 41 X 33


GLて'6IE
Table 4. Solution Statistics for Cray Y-MP Computer (3-Block Grid)
$P_{t \infty}=25 \mathrm{psia}$
$T_{t \infty}=530^{\circ} \mathrm{R}$



Note: Single Grid for Coarse Grid Solution
Figure 3a. Grid Block Boundaries (Cross-Stream View)

Figure 3b. Boundary Conditions (Streamwise View)


Figure 5. Mach Number Contours - Streamwise View

Note: Contour Increment $\mathbf{= 0 . 0 5}$
6b. 3 Block Grid
Figure 6. Mach Number Contours - Cross-Streamwise View


Figure 6. Mach Number Contours - Cross-Streamwise View Close-Up
$M=1.8$
$\alpha=15.750^{\circ}$
$x / I=0.85$

(a) $\phi=0^{\circ}$

Figure 7. Mach Number Profiles $(x / I=0.85)$


$\begin{array}{llllllll} & 0.0 & 0.2 & 0.4 & 0.6 & 0.8 & 1.0 & 1.2 \\ \text { Local Mach number, M } & & 1.4 & 1.6 \\ \text { (b) } \phi=45^{\circ} & & & & & \\ \text { (b) } & & & & & & \end{array}$


$\begin{array}{lllllllllll}0.0 & 0.2 & 0.4 & 0.6 & 0.8 & 1.0 & 1.2 & 1.4 & 1.6 & 1.8 & 2.0 \\ \text { Local Mach number, M }\end{array}$

 Local Mach number, $M$
(h) $\phi=180^{\circ}$
Concluded


Figure 7.




(c) $\phi=90^{\circ}$
Figure 10. Flow Angle



$$
\text { (d) } \phi=135^{\circ}
$$

$$
\text { Profiles }(x / l=0.85)
$$




| $\circ$ | $\mathrm{U}+180^{\circ}$ |
| :--- | :--- |
| $\square$ | $\mathrm{U}+90^{\circ}$ |
| $\Delta$ | $\mathrm{U}+\mathrm{o}^{\circ}$ |
| - | VISCOUS SUBLAYER |
| $\cdots$ | CO..... |
| COLES |  |







 (c) $\phi=170^{\circ}$



[^0]

Figure 15. Mach Number Profiles at $\phi=170^{\circ},(x / l=0.85)$










Figure 19. Mach Number Profiles
3-Block Grid $(\mathrm{x} / \mathrm{I}=0.85)$







Appendix $A$

## Test Data for Several Cases

Appendix A: Test Data for Several Cases

| H | $=$ height above surfaces in inches |
| :---: | :---: |
| $\omega$ | $=$ flow angle in degrees relative to cone generator |
| M/ME | $=$ Mach number / edge Mach number |
| T/TE | $=$ static temperature / edge static temperature |
| $U B / U E$ | $=$ velocity $/$ edge velocity |
| U/UE | $=$ velocity component parallel to edge velocity $/$ edge velocity |
| V/UE | $=$ velocity component normal to edge velocity / edge velocity |
| $M * \sin (O M-O M E)$ | $=$ Mach number * sin (flow angle - edge flow angle) |
| $M * \cos (O M-O M E)$ | $=$ Mach number * cos (flow angle - edge flow angle) |
| U1/UE | $=$ velocity component parallel to cone generator / edge velocity |
| $V 1 / U E$ | $=$ velocity component normal to cone generator / edge velocity |
| MSOM | $=$ Mach number * sin (flow angle) |
| MCOM | $=$ Mach number * cos (flow angle) |
| PE/POD | $=$ edge static pressure /pitot static pressure outside |
|  | boundary layer on windward generator |
| DEL1 | $=\text { streamwise displacement thickness }=\int_{0}^{h_{c}}\left(1-\frac{\rho_{u}}{\rho_{c} u_{c}}\right) d h$ |
| DEL2 | $=$ crossflow displacement thickness $=-\int_{0}^{h_{e}} \frac{\rho v}{\rho_{e} u_{e}} d h$ |
| TH 11 | $=\int_{0}^{h_{e}} \frac{\rho u}{\rho_{e} u_{e}}\left(1-\frac{u}{u_{e}}\right) d h$ |
| TH12 | $=\int_{0}^{h_{e}} \frac{\rho_{c} v}{\rho_{e} u_{e}}\left(1-\frac{u}{u_{e}}\right) d h$ |
| TH21 | $=-\int_{0}^{h_{c}} \frac{v}{u_{c}} \frac{\rho u}{\rho_{e} u_{e}} d h$ |
| TH22 | $=-\int_{0}^{h_{e}} \frac{\rho v^{2}}{\rho_{c} u_{e}^{2}} d h$ |
| AL/THC | $=$ angle of attack / cone semi angle |
| phipp | $=$ circumferential angle at which data was taken |









 $c$
$c$
$c$
$c$
$c$
 0.110 $n$
$x$
$c$
$c$
$c$ $0.0 n 91$ ก． 11095 ．0100 $m$
$c$
$c$ ．018 5 .0086
0043 とてO
をっO .002
.0
.0

.0 | 8 |
| :--- |
| 8 |
| 8 |
| 0 |
| 1 |
| 1 |
| $n$ |













#### Abstract

      











 | -029 |
| :--- |
| 627 |




$\qquad$ ocoorvさotounvonommsomartomommNnnm इ

 ज





10
GM-GMEI
$0.99 ?$
0.993







돈








$$
\begin{aligned}
& \text { SIOM-OMFI } \\
& 0.669 \\
& 1.177 \\
& 1.574 \\
& 1.550 \\
& 1.770 \\
& 1.911 \\
& 2.031 \\
& 2.071 \\
& 2.099 \\
& 2.172 \\
& 2.236 \\
& 2.329 \\
& 2.353 \\
& 2.412 \\
& 7.499 \\
& 2.567 \\
& 2.654 \\
& 2.653 \\
& 2.718 \\
& 2.902 \\
& 3.134 \\
& 3.337 \\
& 3.536 \\
& 3.750 \\
& 3.949 \\
& 4.118 \\
& 4.233 \\
& 4.328 \\
& 4.386 \\
& 4.424 \\
& 4.484
\end{aligned}
$$













R
늘





$\overline{\mathrm{TH}} \mathrm{H} 2=0.0005$






ERROR - FLOATING-POINI DIVISION BY ZERO HAS OCCURREO IN SUB-PROGRAM DUEG
ERROR - FLGATING-POINT DIVISION GY 2ERO HAS OCCURRED IN SUB-PROGRAM DWEG

- AT ADORESS ODOQ52 RELATIVE TO THE ENTRY POINT OF DWEG






 11 11 11


[^1]














|  |
| :---: |
|  |  |























DEL $2=-0.0081$









$\ddot{0} 9$
$10 M-$
 HII




$\qquad$







 U












$1+21=0 . \overline{0} 434$

5
$o$
0
0
0
$\vdots$
$\vdots$
$\vdots$
0
0
0
0





 0
1
1
0
0
0
0
$c$
$c$
$c$
$c$
0
$i \neq 1$

 - 0.0 one7-5. -0.noli-0.
 $=0$
$\vdots$
0
0
0
0
0
0
$c$
1 $-0.0097-0.0$
$-0.0097-0$. i
$i$
$a$
$\vdots$
$c$
$i$
$i$ $c$
$=$
$i$
$i$ 0
0
0
$c$
$i$





 NNMのはNNよ 50
$N 0$
$N$




$\because+/ P n \cap=0.2021$




를

方．

 ミ No













$\qquad$
5
$m$
$\infty$
$\infty$
$\cdots$
0






mí -


 ₹

 ioipil $\qquad$










 $\underset{2}{5}$



 u'in min ind $\sigma-\operatorname{coc} \pi \times n=N m-1+40$ mic









 $i$





 VIfUE
















1H22 $=$



| REPORT DOCUMENTATION PAGE | Form Approved OMB No. 0704-0188 |
| :---: | :---: |
|  <br>  <br>  |  |
| 1. AGENCY USE ONLY (Leave blank) 2. REPORT DATE <br> November 1993 3. REPORT TYPE | DATES COVERED nal Contractor Report |
| 4. TITLE AND SUBTTILE <br> Thin-Layer and Full Navier-Stokes Calculations for Turbulent Supersonic Flow Over a Cone at an Angle of Attack | 5. FUNDING NUMBERS |
| 6. AUTHOR(S) <br> Crawford F. Smith and Steve D. Podleski | C-NAS3-25266 |
| 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) <br> Sverdrup Technology, Inc. <br> Lewis Research Center Group <br> 2001 Aerospace Parkway <br> Brook Park, Ohio 44142 | 8. PERFORMNG ORGANIZATION REPORT NUMBER E-8197 |
| 9. SPONSORINGMONTOAING AGENCY NAME(S) AND ADDRESS(ES) <br> National Aeronautics and Space Administration <br> Lewis Research Center <br> Cleveland, Ohio 44135-3191 | 10. SPONSORINGMONITORING AGENCY REPORT NUMBER NASA CR-189103 |

11. SUPPLEMENTARY NOTES

Project Manager, Robert E. Coltrin, Propulsion Systems Division, (216) 433-2181.

12a. DISTRIBUTIONAVAILABILITY STATEMENT
12b. DISTRIBUTION CODE
Unclassified -Unlimited
Subject Category 02
13. ABSTRACT (Maximum 200 words)

The proper use of a computational fluid dynamics code requires a good understanding of the particular code being applied. In this report the application of CFL3D, a thin-layer Navier-Stokes code, is compared with the results obtained from PARC3D, a full Navier-Stokes code. In order to gain an understanding of the use of this code, a simple problem was chosen in which several key features of the code could be exercised. The problem chosen is a cone in supersonic flow at an angle of attack. The issues of grid resolution, grid blocking, and multigridding with CFL3D are explored. The use of multigridding resulted in a significant reduction in the computational time required to solve the problem. Solutions obtained are compared with the results using the full Navier-Stokes equations solver PARC3D. The results obtained with the CFIL3D code compared well with the PARC3D solutions.


| 14. SUBJECT TERMS <br> Navier Stokes; Conical flow; Computational fluid dynamics |  |  | 15. NUMBER OF PAGES 97 |
| :---: | :---: | :---: | :---: |
|  |  |  | 16. PRICE CODE A05 |
| 17. SECURTY CLASSIFICATION OF REPORT <br> Unclassified | 18. SECURTY CLASSIFICATION OF THIS PAGE Unclassified | 19. SECURITY CLASSIFICATION OF ABSTRACT <br> Unclassified | 20. LIMITATION OF ABSTRACT |

National Aeronautics and
Space Administration
Lewis Research Center
Cleveland, OH 44135-3191
Officlal Business
Penaly lor Privale Use $\$ 300$


[^0]:    (a) Single Grid, $C F L=5,1$

[^1]:    

