Addition of Equilibrium Air to an Upwind Navier-Stokes Code and Other First Steps Toward a More Generalized Flow Solver

Bruce S. Rosen

March 1991

NASA Technical Memorandum 102616
Addition of Equilibrium Air to an Upwind Navier-Stokes Code
and Other First Steps toward a More Generalized Flow Solver*

Bruce S. Rosen†
Grumman Aircraft Systems Division
Bethpage, NY 11714

ABSTRACT

An upwind 3-D finite volume Navier-Stokes code is modified to facilitate modeling of complex geometries and flow fields presented by proposed National Aero-Space Plane concepts. Code enhancements include an equilibrium air model, a generalized equilibrium gas model, and several schemes to simplify treatment of complex geometric configurations. The code is also restructured for inclusion of an arbitrary number of independent and dependent variables. This latter capability is intended for eventual use to incorporate nonequilibrium/chemistry gas models, more sophisticated turbulence and transition models, or other physical phenomena which will require inclusion of additional variables and/or governing equations. Comparisons of computed results with experimental data and with results obtained using other methods are presented for code validation purposes. Good correlation is obtained for all of the test cases considered, indicating the success of the current effort. This work was conducted at the NASA Langley Research Center, during participation in the NASA/Industry Fellowship Program for the National Aero-Space Plane.

*This work was conducted at the NASA Langley Research Center, during participation in the NASA/Industry Fellowship Program for the National Aero-Space Plane
†Engineering Specialist
INTRODUCTION

The National Aero-Space Plane (NASP) program has highlighted the need for development of advanced computational fluid dynamics methodology. The success of the program, unlike that for any previous aircraft, depends upon the availability of the state-of-the-art in flow simulation and prediction. Advances in flow discretization techniques, solution algorithms, equilibrium and nonequilibrium/chemistry gas models, and turbulence and transition models must be incorporated into methodology capable of treating the complex geometries and flow fields presented by proposed NASP concepts.

The NASA/Industry Fellowship Program provided this author with an opportunity to assist in the development of one such method. The basic CFL3D code, an advanced thin-layer Navier-Stokes flow solver which is relatively easy to use and which features the flexibility required to treat complex flows, was modified during this effort to incorporate equilibrium air and generalized equilibrium gas models, and to further enhance its geometric modeling capabilities. At the same time, the code was restructured to facilitate future computations incorporating an arbitrary number of independent and dependent variables. This latter capability is intended for eventual use to incorporate nonequilibrium/chemistry gas models, more sophisticated turbulence and transition models, or other physical phenomena which will require inclusion of additional variables and/or governing equations.

NOMENCLATURE

\begin{align*}
a &= \text{speed of sound} \\
e &= \text{internal energy per unit mass} \\
L &= \text{reference length} \\
M &= \text{Mach number} \\
p &= \text{pressure} \\
Pr &= \text{Prandtl number} \\
Re &= \text{Reynolds number} \\
T &= \text{temperature} \\
u, v, w &= \text{Cartesian velocity components} \\
x, y, z &= \text{Cartesian spatial coordinates} \\
\end{align*}
The computer program to be described is derived from the April 1988 release of CFL3D (Version 1.0), a method which is well documented in the open literature [1,2,3,4]. A brief outline of CFL3D methodology is given below, followed by a discussion of enhancements and features incorporated in the present code.

Overview of Basic CFL3D Methodology

The governing flow equations are the three-dimensional, time-dependent, conservation law form of the compressible Euler or thin-layer Reynolds-averaged Navier-Stokes equations, expressed in generalized coordinates. An upwind-biased approach with up to third order accuracy is used to evaluate the inviscid fluxes at the cell interfaces, as described below. A spatially-split, three-factor approximate factorization algorithm and Euler implicit time integration/linearization is used to advance the solution (cell-averaged flow properties) in time [5].

Inviscid flux interface values are obtained using a MUSCL interpolation scheme [6], coupled to either the flux difference splitting (FDS) scheme of Roe [7,8] or the flux vector splitting (FVS) scheme of Van Leer [5,9]. Flux splittings are based on a one-dimensional Riemann
problem, and are subsequently modified to treat multi-dimensional flows. Overall, these approaches provide an upwind-biasing in the flux interface evaluation. They also introduce an amount of dissipation which is consistent with the discretization of the governing flow equations, and which is required to stabilize the solution procedure. The so-called smooth and min-mod flux gradient limiters are optionally employed, to minimize the adverse effects of large flow gradients and discontinuities (such as shock waves).

At each time step, the FDS/FVS approaches lead to a series of 5-by-5 block tridiagonal matrix inversions, for each of the spatial directions. Additional approximations may also be made in the FDS scheme so as to diagonalize the solution matrices [3]. This leads to a series of scalar tridiagonal matrix inversions, and an attendant reduction in execution time.

Viscous and heat flux interface values are obtained using central finite-difference formulae. The laminar thin-layer Navier-Stokes terms may be included in all three directions. A Baldwin-Lomax algebraic turbulence model [10] is also employed. The effects of turbulence may be included in one direction, or in two directions via a distance-weighted two-wall corner model for the turbulent eddy viscosity.

A zonal grid structure facilitates modeling of complex geometries and/or flow fields. Explicit treatment of grid boundaries further simplifies this task, since boundary condition subroutines are easily modified for specific or unusual cases. Provisions are included for treatment of blocked grids, longitudinally-patched grids [11], and dynamic moving grids. A variety of cell-center or cell-interface type boundary conditions may also be specified at grid boundaries ... freestream flow, extrapolation from the interior (supersonic outflow), subsonic characteristic inflow/outflow (based on one-dimensional Riemann invariants [12]), inviscid wall (flow tangency), viscous wall (adiabatic or fixed wall temperature), and an assortment of symmetry/periodicity/singular-axis/wake-continuation type boundary conditions.

Several schemes are available to reduce overall execution time, particularly for computing steady flows. Local-time-stepping and multigrid [12,13,14] techniques accelerate code convergence. Mesh sequencing is a technique whereby solutions obtained on coarser grids are
used to initialize flow field data on successively finer grids, until finally a solution is obtained on
the desired input grid. Mesh embedding is a technique whereby enhanced solution accuracy is
obtained by locating even finer grids in particular regions of interest. Both mesh sequencing and
mesh embedding reduce the computational effort expended to achieve a given level of solution
accuracy, and their use is facilitated by automated grid generation and flow field interpolation
routines.

In February 1989, while the present code was still under development, an updated version
of CFL3D (Version 1.1) became available. The enhanced capabilities of the updated version of
CFL3D were subsequently incorporated in the present code, including an improved treatment for
longitudinally-patched grids [15], more generalized boundary conditions, and mesh sequencing for
two-dimensional flows. The only CFL3D enhancement not found in the present code is an
alternate two-factor approximate factorization algorithm [12].

Equilibrium Air and Generalized Equilibrium Gas Models

The methodology described above assumes a perfect gas model for the thermodynamic and
transport properties of the fluid. Versions 1.0 and 1.1 of CFL3D further assume air to be the
working fluid. These restrictions do not apply to the present code, which features more general
equilibrium gas capabilities.

The flux-splitting schemes of Roe and Van Leer are extended in the present code to treat
real gases, using techniques developed by Grossman and Walters [16]. The perfect gas
relationships are replaced by equilibrium gas relationships, usually in the form of curve fits. The
specific heat ratio \( \gamma \) employed in the flux-splitting schemes is then replaced by the "equivalent"
values \( \tilde{\gamma} = 1 + p / \rho e \) and \( \tilde{\Gamma} = a^2 \rho / p \).

Two equilibrium gas models for thermodynamic properties (\( \rho, p, e, a \) and \( T \)) are
incorporated into the present code. The first, due to Srinivasan and Tannehill [17], consists of
curve fits for equilibrium air, and executes in scalar mode. The second, due to Liu and Vinokur
[18], is a generalized equilibrium gas model, uses bicubic spline interpolation (based on an
auxiliary interpolation coefficient data file), and executes in vector mode. An interpolation coefficient file for equilibrium air obtained from Liu was augmented by this author in order to use the approach at lower temperatures normally considered to be in the perfect gas regime.

The equilibrium gas model for transport properties ($\mu$, $\kappa$, and $Pr$) is the equilibrium air curve fits due to Srinivasan and Tannehill [19]. Versions which execute in vector mode were developed by this author, after discovering that more execution time was used for computing transport properties than for computing thermodynamic properties.

The Liu and Vinokur thermodynamic property model is not restricted to equilibrium air, since auxiliary interpolation coefficient data files for other equilibrium gases could be constructed. A similar approach for the transport properties is hopefully under development, and when available can be incorporated into the present code as well.

Relative to perfect gas computations, the original (scalar) Srinivasan and Tannehill thermodynamic and transport property gas models result in roughly a 125% increase in execution time. Using the vectorized Srinivasan and Tannehill transport property model results in only about a 50% increase in execution time, while using the vectorized Srinivasan and Tannehill transport property model and the vectorized Liu and Vinokur thermodynamic property model results in only about a 20% increase in execution time. Of course, these numbers are approximate, and reflect average values obtained for a variety of test cases.

First Steps towards a More Generalized Flow Solver

In conjunction with the equilibrium gas flux-splitting capability, the present code was enhanced so as to permit an arbitrary number of independent and dependent variables to be stored in the $q$-vector.

The basic CFL3D code stores only the five independent variables $\rho$, $u$, $v$, $w$, and $p$ in the $q$-vector. In the present code, an arbitrary number of independent variables (i.e., the number of conserved variables or governing equations), $lqcv$, and an arbitrary number of dependent variables (e.g., $\tilde{T}$ and $\Gamma$), $lqdv$, may be stored in the $q$-vector. The values of $lqcv$ and $lqdv$ need be set only
once, in a parameter statement in the main program. These values, and the total number of variables in the $q$-vector, $lqt = lqcv + lqdv$, are subsequently passed to the required subroutines as arguments and/or through common blocks, for appropriate dimensioning of arrays and indexing of do loops.

This coding structure is a first step towards a more generalized flow solver which might incorporate nonequilibrium/chemistry gas effects, more sophisticated turbulence and transition modeling, or other physical phenomena which will require inclusion of additional variables and/or governing equations. Additional work will be required before this goal is achieved. For example, matrix inversion logic is currently fixed to treat 5 governing equations, and increasing the value of $lqt$ may result in overlap or overflow of flux routine scratch arrays. Nevertheless, the majority of the present code should not require further modification in order to incorporate more generalized flow models.

Other Code Enhancements

Two important features of the present code were developed to enhance user friendliness. First, the path and name of all auxiliary data files (currently as many as 11) are specified via the standard unit 5 input data, rather than in the FORTRAN coding itself, to avoid code modification and recompilation. Second, user specified scale factors for length, mass, and temperature permit the use of arbitrary dimensions (e.g., metric or English) for the input and output data.

Other features of the present code offer enhanced capabilities. Most significant of these is a very generalized grid blocking boundary condition capability (developed by George Switzer, Analytical Services and Materials, Inc.). Also noteworthy is a "jagged" boundary condition algorithm (developed by Mark Eppard, Analytical Services and Materials, Inc.) which permits treatment of surface edges that are skewed with respect to, or cut across, grid lines. A new flux interface averaging procedure, developed at NASA Langley, may enhance convergence for cold wall cases. Since the jagged boundary condition and flux interface averaging capabilities are not
yet fully generalized, they have been commented out in the FORTRAN coding (lines start with the characters "cbse"), and should be activated only by knowledgeable users.

RESULTS

Results computed for several test cases are presented in order to evaluate the present code's capabilities. For each test case, calculations are compared to results obtained using other methods, or to experimental data, and previous comparisons by other investigators are cited.

Computations were obtained using the perfect gas model, the Srinivasan and Tannehill equilibrium air model, and the Liu and Vinokur generalized equilibrium gas model with the augmented auxiliary interpolation coefficient data file for equilibrium air. Since the two equilibrium air models gave essentially identical results for all of the test cases, only those obtained with the Liu and Vinokur model are presented herein.

Unless otherwise noted, all results were computed using FDS, third order upwind-biased spatial accuracy, min-mod flux limiter, and the 5-by-5 block tridiagonal matrix inversion algorithm. The majority of the computations were made for laminar flow, and included thin-layer terms in the $k$-direction (normal to the body surface) only. A fixed wall temperature was specified for use in all viscous wall boundary conditions. Local time stepping was used to accelerate convergence to steady state.

Supersonic Laminar Flat Plate Boundary Layer

The first test case consists of supersonic laminar flow over a flat plate (this is also one of the test cases studied in [20]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 2.0 \\
Re_\infty / L &= 1.65 \cdot 10^6 / m \\
T_\infty &= 221.6 \text{ }^\circ K \\
T_w &= 221.6 \text{ }^\circ K
\end{align*}
\]
A grid consisting of 51 grid points in the streamwise direction and 100 grid points normal to the surface was employed. Average grid spacing normal to the surface was $0.43 \cdot 10^{-4}$ m, producing an average $y^+$ of 1.33. The residual was reduced approximately 4.5 orders of magnitude over 4000 time steps. NASA Cray-YMP (Reynolds) execution times required for the perfect gas, Srinivasan and Tannehill, and Liu and Vinokur gas models were $4.5 \cdot 10^{-5}$, $7.2 \cdot 10^{-5}$, and $5.4 \cdot 10^{-5}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed supersonic laminar flat plate boundary layer results are compared to predictions made using a conventional boundary layer calculation [21] (boundary layer calculations supplied by Douglas Dilley, Analytical Services and Materials, Inc.). Velocity and temperature profiles at an axial location $x = 1m$ are presented in Fig. 1. Axial distributions of heat transfer and skin friction are presented in Fig. 2. All of the present results show excellent correlation with the boundary layer predictions. As expected, equilibrium gas effects are not significant for this relatively low temperature flow.

### Hypersonic Laminar Flat Plate Boundary Layer

The second test case consists of hypersonic laminar flow over a flat plate (this is also one of the test cases studied in [22]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 20.0 \\
Re_\infty/L &= 2.0 \cdot 10^5 / m \\
T_\infty &= 100.0 \, ^\circ K \\
T_w &= 1000.0 \, ^\circ K
\end{align*}
\]

A grid consisting of 64 grid points in the streamwise direction and 64 grid points normal to the surface was employed. Average grid spacing normal to the surface was $0.1 \cdot 10^{-3}$ m, producing an average $y^+$ of 1.08. The residual was reduced approximately 5 orders of magnitude over 4500 time steps. Reynolds execution times required for the three gas models were $4.4 \cdot 10^{-5}$, $8.8 \cdot 10^{-5}$, and $5.5 \cdot 10^{-5}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively.
Computed hypersonic laminar flat plate boundary layer results are compared to predictions made using CFL3DE, an extension of the CFL3D method by other investigators [23] which also incorporates equilibrium air effects (CFL3DE calculations supplied by Douglas Dilley, Analytical Services and Materials, Inc.). Velocity and temperature profiles at an axial location \( x = 1m \) are presented in Fig. 3. Axial distributions of heat transfer, skin friction, and pressure are presented in Fig. 4. The present results show excellent correlation with the CFL3DE calculations. Equilibrium gas effects are significant, particularly for the temperature profile predictions.

High Speed Inlet

The third test case is the high speed flow through an inlet (this is also one of the test cases studied in [16]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 5.0 \\
Re_\infty / L &= 4.94 \cdot 10^6 / m \\
T_\infty &= 3573.0 \ ^\circ K
\end{align*}
\]

The inlet features a 10° compression, followed downstream by a 10° expansion. Inviscid computations were obtained, to permit comparison with the exact perfect gas and equilibrium air solutions. A grid consisting of 201 grid points in the streamwise direction and 51 grid points normal to the surface was employed. The residual was reduced approximately 3 orders of magnitude over 3000 times steps. NASA Cray-2 (Navier) execution times required for the three gas models were \( 1.3 \cdot 10^{-4}, 1.6 \cdot 10^{-4}, \) and \( 1.3 \cdot 10^{-4} \) cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed high speed inlet results are compared to the exact inviscid solutions. Inlet-wall density, pressure, and temperature distributions appear in Fig. 5. The agreement is good, except for the temperature level aft of the expansion, which is overpredicted. The same effect is seen in [16]. No attempt was made to try to eliminate the post-shock oscillation evident in the present predictions, which nonetheless indicate the proper perfect gas/equilibrium air trends.
Calculations were also made using FVS. The residual was reduced 3.5 orders of magnitude over 3000 time steps. Navier execution times required for the three gas models were $8.1\cdot10^{-5}$, $1.2\cdot10^{-4}$, and $8.3\cdot10^{-5}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively. The results are shown in Fig. 6, and are similar to those obtained using FDS.

**Bent Nose Biconic**

The fourth test case is high speed laminar flow past a bent nose biconic (one of the test cases studied in [24]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 9.86 \\
Re_\infty/L &= 1.842\cdot10^6/m \\
T_\infty &= 49.75\, ^\circ K \\
T_w &= 300.0\, ^\circ K
\end{align*}
\]

As shown schematically in Fig. 7, a total of 85 grid points in the streamwise direction, 45 grid points normal to the surface, and 23 grid points circumferentially was used to model one-half of the configuration, with symmetry imposed across the $x$-$z$ plane. Average grid spacing normal to the surface was $0.5\cdot10^{-5} m$, producing an average $y^+$ of 0.23. To avoid difficulties sometimes encountered using FDS to compute blunt nose flow fields, FVS was employed. The mesh sequencing capability was also used, to minimize overall execution time. The residual was reduced approximately 4.5 orders of magnitude over 4300 time steps. NASA Cray-2 (Voyager) execution times required for the three gas models were $7.0\cdot10^{-5}$, $9.9\cdot10^{-5}$, and $8.3\cdot10^{-5}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed bent nose biconic surface heat transfer rates are compared to experimental data [25] in Fig. 8. The present results show good correlation with the data. Equilibrium gas effects are less significant than expected for this high speed flow.
Flared Cone (Laminar)

The fifth test case is that of high speed laminar flow past a flared cone (one of the test cases studied in [24]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 16.93 \\
Re_\infty / L &= 1.976 \times 10^5 / ft \\
T_\infty &= 83.73 \degree R \\
T_w &= 530.0 \degree R
\end{align*}
\]

As shown schematically in Fig. 9, a total of 97 grid points in the streamwise direction, 45 grid points normal to the surface, and 19 grid points circumferentially was used to model one-half of the configuration, with symmetry imposed across the x-z plane. Average grid spacing normal to the surface was 0.24-10^{-4} ft, producing an average y+ of 0.09. Employing the mesh sequencing capability, the residual was reduced approximately 3.5 orders of magnitude over 3100 time steps. Navier execution times required for the three gas models were 1.2-10^{-4}, 1.6-10^{-4}, and 1.5-10^{-4} cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed flared cone surface heat transfer, skin friction, and pressure distributions for laminar flow are compared to experimental data [26] in Fig. 10. The present results show good correlation with the data. Equilibrium gas effects are less significant than expected for this high speed flow.

Flared Cone (Turbulent)

The sixth test case considered is high speed turbulent flow past a flared cone. The flow conditions are:

\[
\begin{align*}
M_\infty &= 7.85 \\
Re_\infty / L &= 4.697 \times 10^6 / ft \\
T_\infty &= 130.2 \degree R \\
T_w &= 530.0 \degree R
\end{align*}
\]

The grid, shown schematically in Fig. 11, is similar to that for the laminar case. Average grid spacing normal to the surface was 0.83-10^{-5} ft, producing an average y+ of 0.51. The turbulence
model was employed in the $k$-direction (normal to the body surface) only. As for the laminar case, mesh sequencing was employed, and the residual was reduced approximately 3.5 orders of magnitude over 3100 time steps. Voyager execution times required for the three gas models were $6.4 \cdot 10^{-5}$, $8.7 \cdot 10^{-5}$, and $7.2 \cdot 10^{-5}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed flared cone surface heat transfer, skin friction, and pressure distributions for turbulent flow are compared to experimental data [26] in Fig. 12. The present results show good correlation with the data. The figure clearly indicates that appropriate use of the algebraic turbulence model can enhance code predictions.

### Laminar Corner Flow

The seventh test case consists of laminar flow in a corner formed by two intersecting wedges (this flow is also studied in [27]). The flow conditions are:

\[
\begin{align*}
M_\infty &= 3.0 \\
Re_\infty &= 2.22 \cdot 10^5 \\
T_\infty &= 105.0 \, ^\circ\text{K} \\
T_w &= 294.0 \, ^\circ\text{K}
\end{align*}
\]

The flow was computed on a 120 by 120 crossflow plane grid, assuming conical flow in the streamwise direction. Average grid spacing normal to the surface was $0.14 \cdot 10^{-3}$ times $x$, producing an average $\gamma^+$ of 6.05. Laminar viscous thin layer terms normal to both walls were included, in the $j$- and $k$-directions. The residual was reduced approximately 3 orders of magnitude over 3600 time steps. Navier execution times required for the three gas models were $7.8 \cdot 10^{-5}$, $1.2 \cdot 10^{-4}$, and $1.1 \cdot 10^{-4}$ cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed wall pressure distributions for laminar corner flow are compared to experimental data [28] in Fig. 13. The present results show good correlation with the data.
Turbulent Corner Flow

The eighth test case consists of turbulent corner flow (also studied in [27]). The flow conditions are:

\[ M_\infty = 3.0 \]
\[ Re_\infty = 3.0 \cdot 10^6 \]
\[ T_\infty = 105.0 \, ^\circ K \]
\[ T_w = 294.0 \, ^\circ K \]

The flow was again computed on a 120 by 120 crossflow grid, and was assumed to be conical. Average grid spacing normal to the surface was \(0.10 \cdot 10^{-3}\) times \(x\), producing an average \(y^+\) of 5.51. The two-wall corner model was used to simultaneously include turbulence effects normal to both walls, in the \(j\) and \(k\)-directions. The residual was reduced approximately 4.5 orders of magnitude over 5400 time steps. Navier execution times required for the three gas models were \(7.9 \cdot 10^{-5}\), \(1.1 \cdot 10^{-4}\), and \(1.1 \cdot 10^{-4}\) cpu-seconds per mesh-cell-point per time-step-iteration, respectively.

Computed wall pressure distributions for turbulent corner flow are compared to experimental data [28] in Fig. 14. The present results again show good correlation with the data. Compared to the previous laminar corner flow predictions, these results indicate that the proper trending is produced by use of the two-corner wall turbulence model.

CONCLUDING REMARKS

The results presented herein show good correlation for all of the test cases considered. Since the equilibrium gas flux splitting schemes make use of the "equivalent" specific heat ratios, \(\tilde{\gamma}\) and \(\Gamma\), which are stored in the \(q\)-vector as additional dependent variables, these results validate not only the implementation of the flux difference and flux vector splitting schemes, but also the restructuring of the present code to permit an arbitrary number of independent and dependent variables.
Both the Srinivasan and Tannehill equilibrium air model and the Liu and Vinokur generalized equilibrium gas model reproduce perfect gas results or, where appropriate, exhibit the proper real gas trends. With full vectorization, the equilibrium gas calculations were possible with only a small (~20%) increase in execution time. Successful coupling of the equilibrium air/equilibrium gas models with the one- or two-wall algebraic turbulence model was also demonstrated.

Although not all of the code's capabilities were exercised, the results are indicative of the success of a substantial portion of the current effort. The resulting method should prove to be a valuable tool for use by the National Aero-Space Plane program, as well as a good starting point for future efforts aimed at incorporating nonequilibrium/chemistry effects, more sophisticated turbulence and transition models, or a variety of other physical phenomena.
REFERENCES


Figure 1: Supersonic Flat Plate Boundary Layer, Profiles at $x = 1$ m.

(a) Velocity

(b) Temperature
Figure 2: Supersonic Flat Plate Boundary Layer, Surface Distributions.
Figure 3: Hypersonic Flat Plate Boundary Layer, Profiles at $z = 1m$. 
Figure 4: Hypersonic Flat Plate Boundary Layer, Surface Distributions.
Figure 4: Hypersonic Flat Plate Boundary Layer, Surface Distributions (Conc'd).
Figure 5: High Speed Inlet, Wall Distributions; Flux Difference Splitting.
(c) Temperature

Figure 5: High Speed Inlet, Wall Distributions; Flux Difference Splitting (Conc'd).

(a) Density

Figure 6: High Speed Inlet, Wall Distributions; Flux Vector Splitting.
Figure 6: High Speed Inlet, Wall Distributions; Flux Vector Splitting (Conc'd).
Figure 7: Schematic of Computational Grid for Bent Nose Biconic.

Figure 8: Bent Nose Biconic, Surface Heat Transfer Distribution.
Figure 9: Schematic of Computational Grid for Flared Cone; Laminar Flow.

Figure 10: Flared Cone, Surface Distributions; Laminar Flow.
Figure 10: Flared Cone, Surface Distributions; Laminar Flow (Conc'd).
Figure 11: Schematic of Computational Grid for Flared Cone; Turbulent Flow.

![Schematic of Computational Grid](image)

Figure 12: Flared Cone, Surface Distributions; Turbulent Flow.

(a) Heat Transfer

![Heat Transfer Graph](image)

- - - CFL3D (perfect gas, turbulent)
- - CFL3D (equilibrium air, turbulent)
- - - CFL3D (equilibrium air, laminar)

* Experiment

$M_{\infty} = 7.850$  \hspace{2cm} $Re = 4.697 \times 10^6/ft$

Heat Transfer, QDOT (Btu/sec-ft$^2 \cdot 10^{-1}$)

Axial Location, $X/L$ ($L=24$ in)
Figure 12: Flared Cone, Surface Distributions; Turbulent Flow (Conc’d).
Figure 13: Laminar Corner Flow, Wall Pressure Distribution.

Figure 14: Turbulent Corner Flow, Wall Pressure Distribution.
APPENDIX A

Modified CFL3D Input Data File Description
******************************* LINE TYPE ONE *******************************

title describing case

******************************* LINE TYPE ONE.FIVE *******************************

(DATA FOR LINE TYPE ONE.FIVE REPEATED FOR EACH FILE)

path/name of binary grid file (unit 01)
path/name of binary restart file (unit 02)
path/name of binary PLOT3D grid file (unit 03)
path/name of binary PLOT3D flowfield file (unit 04)
path/name of binary Liu & Vinokur equilibrium air coefficient file (unit 07)
path/name of primary output file (unit 11)
path/name of FIXI/FIXJ output file (unit 12)
path/name of wing pressure output file (unit 14)
path/name of secondary output file (unit 15)
path/name of flowfield output file (unit 17)
path/name of unsteady cp output file (unit 20)

******************************* LINE TYPE TWO *******************************

xmach - freestream Mach number
alpha - angle of attack
beta - side-slip angle
reue - freestream Reynolds number per unit length (millions)
tinf - freestream temperature (degrees Rankine)
isnd - wall temperature boundary condition flag
   0 adiabatic wall temperature
   1 specified wall temperature

c2spe - wall temperature (temperature at wall divided by temperature of freestream)
   if c2spe<=0, c2spe taken as freestream stagnation temperature

******************************* LINE TYPE THREE *******************************

sref - reference area
cref - reference length
bref - reference span
xmc - moment center in x-direction
ymc - moment center in y-direction
zmc - moment center in z-direction

****************************** LINE TYPE THREE.FIVE ******************************

igas - perfect gas/equilibrium air flag
  1 perfect gas
  2 Tannehill equilibrium air
  3 Liu & Vinokur equilibrium air

gamma - perfect gas, ratio of specific heats

rgas - perfect gas, gas constant

prgas - perfect gas, prandtl number

scalex - meters per unit length
  - 1.0 if using meters
  - 0.3048 if using feet, default
  - 0.02540 if using inches

scalent - degrees Kelvin per unit degree
  - 1.0 if using degrees Kelvin
  - 0.5556 if using degrees Rankine, default

scalem - kilograms per unit mass
  - 1.0 if using kilograms
  - 14.59 if using slugs, default
  - 0.453472 if using pounds (mass)

****************************** LINE TYPE FOUR ******************************

dt - time step
  < 0 local time stepping, CFL=abs(dt)
  > 0 constant time step (=dt)

irest = 0 no restart
  = 1 restart

iflagts = 0 constant dt
  > 0 dt ramped over iflagts steps to dt*fmax

fmax - maximum increase in dt

iunst = 0 steady
  = 1 sinusoidal plunging
  = 2 sinusoidal pitching

rfreq - reduced frequency

alphau - pitching alpha

cloc - pitching center

****************************** LINE TYPE FIVE ******************************
ngrid - number of grids input

nplot3d - number of flowfield data sets to be written in plot3d format
**nprint** - number of data sets to be sent to an output file

**nwrest** - number of iterations between updates of the binary restart file

______________________________ LINE TYPE SIX ______________________________
(DATA FOR LINE TYPE SIX REPEATED NGRID TIMES)

**ncg** - number of coarser grids to construct for multigrid/mesh sequencing (= 0 for embedded mesh)

**iem** - embedded mesh flag
- 0 for global grid
- 1 level of this embedded grid above global grid level

**iadvance** - flag to skip any residual/update calculations
>=0  proceed as usual
< 0  skip residual/update calculations

**iforce** - flag to skip the force routine
>=0  proceed as usual
< 0  skip force calculations

**imesh** - mesh flag for grids topologically similar to:
- 0  no singularities in mesh
- 1  delta wing  (AIAA 87-0207)
- 2  prolate spheroid  (AIAA 87-2627CP)
- 3  prolate spheroid with sting  (AIAA 87-2627CP)
- 10  wing (o-h)
- 11  wing (c-h)  (AIAA 86-0274)
- 12  wing (c-o)  (AIAA 86-0274)

**ivisc(m)** - viscous/inviscid interaction flag  m= 1 : I-direction
- 0  inviscid
- 1  laminar
- 2  turbulent

**NOTE:** The thin layer viscous terms can be included in either the j-, k-, or i-directions, separately. The viscous terms can be included simultaneously in, at most, two directions, either j-k or i-k, for any particular grid. It is preferable to let k be the primary viscous direction and j be the secondary viscous direction.

______________________________ LINE TYPE SEVEN ______________________________
(DATA FOR LINE TYPE SEVEN REPEATED NGRID TIMES)

**grid dimensions:**

**idim** - number pf points in i-direction
- for imesh = 1  axial direction (along chord)  (h-mesh)
- for imesh = 2  circumferentially along body  (o-mesh)
- for imesh = 3  cir. along body/sting  (c-mesh)
- for imesh = 10,11  spanwise direction  (h-mesh)
- for imesh = 12  spanwise:wrapping around wing tip  (o-mesh)

**jdim** - number of points in j-direction
- for imesh = 1,2,3  circumferentially along body/wing  (c-mesh)
- for imesh = 10  circumferentially along chord  (o-mesh)
- for imesh = 11,12  cir. along wing chord and wake  (c-mesh)

**kdim** - number of points in k-direction
- for all imesh, radial direction

**itel** - i location on body
- i at apex for imesh = 1
- 1 for imesh = 2,3,10,11,12

ite2 - i location on body
- 1 at trailing edge for imesh = 1
- idim for imesh = 2,3,12
- 1 at wing tip for imesh = 10,11

jte1 - j location on body
- 1
- j at trailing edge on lower surface for imesh = 1,2,3,10
- jdim for imesh = 11,12
- j at trailing edge on upper surface for imesh = 1,2,3,10

jte2 - j location on body
- jdim

************************************************************* LINE TYPE EIGHT *************************************************************
(Data for Line Type Eight Repeated NGRID Times)

inewg - restart flag for grid (not needed if irest=0)
- 0 read flowfile data from restart file
- 1 initialize at freestream or by linear interpolation from coarser grids

igridc - grid to which this grid connects (input 0 for global mesh(iem=0) and the grid number in which the embedded mesh fits for embedded meshes(iem>0))

js,ks,is - starting indices in connecting grid for placement of embedded mesh (input 0 for global meshes)

je,ke,ie - ending indices in connecting grid for placement of embedded mesh (input 0 for global meshes)

NOTE: The embedded meshes must be a regular refinement in all directions of the grid to which it connects.

************************************************************* LINE TYPE NINE *************************************************************
(Data for Line Type Nine Repeated NGRID Times)

idiag(m) - matrix inversion flag
- 0 5x5 block tridiagonal inversion
- 1 scalar tridiagonal inversions (recommended)

iflim(m) - flux limiter flag
- 0 unlimited
- 1 smooth limiter
- 2 min-mod scheme (recommended)

************************************************************* LINE TYPE TEN *************************************************************
(Data for Line Type Ten Repeated NGRID Times)

ifds(m) - spatial differencing parameter for Euler fluxes
- 0 flux-vector splitting
- 1 flux-difference splitting (Roe's scheme) (recommended)

rkap0(m) - spatial differencing parameter for Euler fluxes
- -1 fully upwind
- 0 Fromme's scheme
- 1 central
- 1/3 upwind-biased third order (recommended)
boundary condition flags:
mtypes(1) - boundary flag for i=0 boundary
mtypes(2) - boundary flag for i=1dim boundary
mtypes(3) - boundary flag for j=0 boundary
mtypes(4) - boundary flag for j=1dim boundary
mtypes(5) - boundary flag for k=0 boundary
mtypes(6) - boundary flag for k=kdim boundary

NOTE: Particular choices of mtypes/j/k determine the type of boundary conditions used at the edges of the computational grids and are best determined by inspection of subroutine BC. Additional boundary condition types can be incorporated into the algorithm by modifying subroutine BC according to the conventions outlined there.

---

nbli - number of block boundary conditions

---

nblon - block boundary condition on or off ( >0 or <0 )

---

mseq - mesh sequencing flag for global grids (maximum 5)
1 single solution on finest grid
2 solution on second finest grid advanced ncyc(1) cycles followed by ncyc(2) cycles on finest grid. The solution on the finest grid is obtained by interpolation from the coarser grid. If ncyc(2)=0, solution terminated on second finest grid after ncyc(1) steps
with restart file written for second finest grid at that point.
> 2 sequencing from coarest to finest mesh as above

mgflag - multigrid flag
  - 0 no multigrid
  - 1 multigrid on coarser global meshes
  - 2 multigrid on coarser global meshes and on embedded meshes

iconsf - conservation flag
  - 0 nonconservative flux treatment for embedded grids
  - 1 conservative flux treatment for embedded grids

mtt  - 0 no additional iterations on the "up" portion of the multigrid cycle
  > 0 mtt additional iterations on the "up" portion of the multigrid cycle

ngam - multigrid cycle flag
  - 1 V-cycle
  - 2 W-cycle

******************************* LINE TYPE THIRTEEN *******************************
(REPEATED FOR EACH SEQUENCE 1 THROUGH MSEQ (COARSEST TO FINEST))

ncycl - number of cycles
mglevg - number of grids to use in multigrid cycling for the global meshes
  - 1 for single grid
  - 2 for two levels
  - m for m levels
nemgl - number of embedded grid levels above the finest global grid (= 0 for global grids coarser than the finest global grid)
  - 0 no embedded grids
  - 1 one embedded grid
  - m m embedded grids
nltfol - number of first order iterations

******************************* LINE TYPE FOURTEEN *******************************
(REPEATED FOR EACH SEQUENCE 1 THROUGH MSEQ (COARSEST TO FINEST))

mitL - iterations on level L for each level L from coarsest to finest (mitL=1 recommended)

******************************* LINE TYPE FIFTEEN *******************************
(REPEATED NPRINT3D TIMES)

block - designated block number for output
istart - starting location in i-direction
iend   - ending location in i-direction
iinc   - increment factor in i-direction
jstart - starting location in j-direction
jend   - ending location in j-direction
jinc   - increment factor in j-direction
kstart - starting location in k-direction
kend   - ending location in k-direction
kinc   - increment factor in k-direction

******************************* LINE TYPE SIXTEEN *******************************
(REPEATED NPRINT TIMES)
block - designated block number for output
istart - starting location in i-direction
iend - ending location in i-direction
iinc - increment factor in i-direction
jstart - starting location in j-direction
jend - ending location in j-direction
jinc - increment factor in j-direction
kstart - starting location in k-direction
kend - ending location in k-direction
kinc - increment factor in k-direction
APPENDIX B

Sample Modified CFL3D Input Data Files
Supersonic Laminar Flat Plate Boundary Layer

2-d plate, cfl3dn - liu, lam
binary grid file
'/scr6/rosen/plt32/plt32.grd'
binary restart file
'/scr6/rosen/plt32/plt32c.bin'
plot3d binary grid file
'/scr6/rosen/plt32/plt32c.plg'
plot3d binary flowfield file
'/scr6/rosen/plt32/plt32c.plq'
Liu & Vinokur binary equilibrium air coefficient file
'/scr6/rosen/cfl3dn/liu/liubsr.cof'
primary output file
'/scr6/rosen/plt32/plt32c.out'
fixi/fixj output file
'/scr6/rosen/plt32/plt32c.fix'
wing pressure output file
'/scr6/rosen/plt32/plt32c.wng'
secondary output file
'/scr6/rosen/plt32/plt32c.sec'
flowfield output file
'/scr6/rosen/plt32/plt32c.prt'
unsteady cp output file
'/scr6/rosen/plt32/plt32c.ucp'

XMACH  ALPHA  BETA  REUE,MIL  TINF,dK  ISND  C2SPE
 2.000 0.000 0.0 1.650000 221.60 1 1.0000

SREF  CREF  BREF  XMC  YMC  ZMC
 1.0000 1.0000 1.0000 0.0 0.0 0.0

IGAS  GAMMA  RGAS  PRGAS  SCALEX  SCALEY  SCALEZ
 3 1.4 286.9 0.72 1.0 1.0 1.0

DT  IREST  IFLAGTS  FMAX  IUNST  RFREQ  ALPHAU  CLOC
-0.001 0 500 10.00 0 0.0000 0.0000 0.0000

NGRID  NPRINT  NWREST
  1 0 2 100

DG  IEM  IADVANCE  IFORCE  IMESH  IVISC(I)  IVISC(J)  IVISC(K)
  0 0 0 0 0 0 0 1

IDIM  JDIM  KDIM  ITE1  ITE2  JTE1  JTE2
  2 51 100 1 1 1 51

INEWG  IGRIDC  IS  JS  KS  IE  JE  KE
  1 0 0 0 0 0 0 0

IDIAI (I)  IDIAI (J)  IDIAI (K)  IFLIM(I)  IFLIM (J)  IFLIM(K)
  0 0 0 2 2 2

IFDS (I)  IFDS (J)  IFDS (K)  RKAP0(I)  RKAP0(J)  RKAP0(K)
  1 1 1 0.33333 0.33333 0.33333

MTYPEI (I)  MTYPEI (2)  MTYPEI (1)  MTYPEJ (2)  MTYPEK (1)  MTYPEK (2)
  11 11 27 27 67 67

NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
0

BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >=0 OR <0 )
BLCK1 IST JST KST IND JND KND IND1 IND2 BLCK2 IST JST KST IND JND KND IND1 IND2
MSEQ  MGFLAG  ICONSF  MTT  NGAM
  1 0 0 0 0 0 1

NCYC  MGLEV0  NEMGL  NTFO
500 01 000

MIT1  MIT2  MIT3  MIT4  MIT5
01 01 01 01 01

PRINT OUT:

BLOCK ISTART  IEND  IINC JSTART  JEND  JINC KSTART  KEND  KINC
  1 1 1 1 1 1 1 1 1
  1 1 1 1 1 1 1 1 1

42
Hypersonic Laminar Flat Plate Boundary Layer

2-d plate, cfl3dn : llu,lam
binary grid file
'/scr6/rosen/plt20/plt20.grd'

binary restart file
'/scr6/rosen/plt20/plt20c.bin'
plot3d binary grid file
'/scr6/rosen/plt20/plt20c.plg'
plot3d binary flowfield file
'/scr6/rosen/plt20/plt20c.plq'
Liu & Vinokur binary equilibrium air coefficient file
'/scr6/rosen/cfl3dn/llu/llubsr.cof'

primary output file
'/scr6/rosen/plt20/plt20c.out'
fixi/fixj output file
'/scr6/rosen/plt20/plt20c.fix'
wing pressure output file
'/scr6/rosen/plt20/plt20c.wng'

secondary output file
'/scr6/rosen/plt20/plt20c.sec'
flowfield output file
'/scr6/rosen/plt20/plt20c.prt'
unsteady cp output file
'/scr6/rosen/plt20/plt20c.ucp'

XMACH ALPHA BETA REUE,MIL TINF,DR ISND C2SPE
20.00 0.000 0.0 0.200000 100.00 1 10.0000

SREF CREF XMC YMC ZMC
0.1000 1.0000 0.1000 .05 .5 0.

IGAS GAMMA RGAS PFRGAS SCALEX SCALET SCALEM
3 1.4 286.9 0.72 1.0 1.0 1.0

DT IREST IFLAGTS FMAX IUNST RFREQ ALPHAU CLOC
-0.001 0 1500 100.00 0 0.00000 0.00000 0.00000

NGRID NPLOT3D NPRINT NWREST
1 0 2 250

NCG IEM IADVANCE IFORCE IMESH IVISC(I) IVISC(J) IVISC(K)
0 0 0 0 0 0 0 0 0 1

DIM JDIM KDIM ITE1 ITE2 JTE1 JTE2
2 65 65 1 2 1 65

INEWG IGRIDC IS JS KS IE JE KE
1 0 0 0 0 0 0 0 0 0

IDIAG(I) IDIAG(J) IDIAG(K) IFLIM(I) IFLIM(J) IFLIM(K)
0 0 0 0 0 0 0 0 2 2

IFDS(I) IFDS(J) RPKAO(I) RPKAO(J) RPKAO(K)
1 1 1 0.33333 0.33333 0.33333

MTYPEI(I) MTYPEI(2) MTYPEI(3) MTYPEI(4) MTYPEI(5) MTYPEI(6)
11 11 27 27 67 67

NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
0

BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >=0 OR <0 )
BLCK1 IST JST KST IND JND KND IND1 IND2 BLCK2 IST JST KST IND JND KND IND1 IND2
MSEQ MFLAG ICONSF MTT NGAM
1 0 0 0 0 0 1

NCYC MGLEVG NEMGL NITFO
1500 01 00 000

MITI MIT2 MIT3 MIT4 MIT5
01 01 01 01 01

PRINT OUT:
BLOCK ISTART IEND IINC JSTART JEND JINC KSTART KEND KINC
1 2 2 1 64 64 1 65 2
1 2 2 1 2 64 64 2 1 1

43
High Speed Inlet

2-D INLET - AIAA 87-1117 (LIU, VAN LEER)

binary grid file
'/scr2/rosen/in/in.grd'

binary restart file
'/scr2/rosen/in/in3.bin'

binary plot3d grid file
'/scr2/rosen/in/in3.plg'

binary plot3d flowfield file
'/scr2/rosen/in/in3.plq'

Liu & Vinokur binary equilibrium air coefficient file
'/scr2/rosen/cfl3dn/liu/liubsr.cof'

primary output file
'/scr2/rosen/in/in3.out'

fixi/fixj output file
'/scr2/rosen/in/in3.fix'

wing pressure output file
'/scr2/rosen/in/in3.wng'

secondary output file
'/scr2/rosen/in/in3.sec'

flowfield output file
'/scr2/rosen/in/in3.prt'

unsteady cp output file
'/scr2/rosen/in/in3.ucp'

XMACH ALPHA BETA REUE,MIL TINF,DK ISND C2SPE
5.000 0.000 0.0 4.940578 3573.0 1 1.0
SREF CREF BREF XMC YMC ZMC
0.1 1.0 0.1 0.5 0.05 0.1
IGAS GAMMA RGAS Pegas SCALEX SCALET SCALEM
3 1.4 286.9 0.72 1.0 1.0 1.0
DT IREST IFLAGTS FMAX IUNST RFREQ ALPHAU CLOC
-0.010 0 300 10.00 0 0.00000 0.00000 0.00000
NGRID NPRINT NWREST
1 0 1 100
NCG IEM IADVANCE IFORCE IMESH IVISC(I) IVISC(J) IVISC(K)
0 0 0 0 0 0 0 0
IDIM JDIM KDIM ITE1 ITE2 JTE1 JTE2
2 201 51 1 2 1 2 201
INEWG IGRIDC IS JS KS IE JE KE
1 0 0 0 0 0 0 0
IDIGA(I) IDIAG(J) IDIAG(K) IFLIM(I) IFLIM(J) IFLIM(K)
0 0 0 0 0 0 0
IFDS(I) IFDS(J) IFDS(K) RKAP0(I) RKAP0(J) RKAP0(K)
0 0 0 0.33333 0.33333 0.33333
MSTYPE(I) MSTYPE(2) MTYPEJ(1) MTYPEJ(2) MTYPEK(1) MTYPEK(2)
27 27 27 27 27 27
NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
0

BLOCK INTERFACE BOUNDARY CONDITION OR OFF ( >-0 OR <0 )
BLCK1 IST JST KST IND JND KND IND1 IND2 BLCK2 IST JST KST IND JND KND IND1 IND2
0 0 0 0 0 0 0
MSEQ MGILAG ICONSF MTT NGAM
1 0 0 0 0 1
NCYC MGBEVG NEMCL NITFO
300 0 0 0
MIT1 MIT2 MIT3 MIT4 MIT5
01 01 01 01
PRINT OUT:
BLOCK ISTART IEND IINC JSTART JEND JINC KSTART KEND KINC
1 2 2 1 1 201 1 1 1 1

44
Bent Nose Biconic

BENT-BICONIC AT LOW-RE ALPHA=0 (NASA-TP-2334)

binary grid file
'/scr/rosen/bnb/bnb.grd'

binary restart file
'/scr/rosen/bnb/b3.bin'

binary plot3d grid file
'/scr/rosen/bnb/b3.plg'

binary plot3d flowfield file
'/scr/rosen/bnb/b3.plq'

Liu & Vinokur binary equilibrium air coefficient file
'/liu/liubsr.cof'

primary output file
'/scr/rosen/bnb/b3.out'

fixi/fixj output file
'/scr/rosen/bnb/b3.fix'

wing pressure output file
'/scr/rosen/bnb/b3.wng'

secondary output file
'/scr/rosen/bnb/b3.sec'

flowfield output file
'/scr/rosen/bnb/b3.prt'

unsteady cp output file
'/scr/rosen/bnb/b3.ucp'

**XMACH** 9.860  
**ALPHA** 0.000  
**BETA** 0.0  
**REUE,MIL** 1.842000  
**TINF,DK** 49.75  
**ISND** 1  
**C2SPE** 6.030151

**SREF** 0.001013  
**CREF** 0.121680  
**XMACH** 9.860  
**SREF** 0.001013  
**IGAS** 3  
**DT** -0.001  
**NGRID** 85  
**NCG** 1  
**IDIM** 1  
**NEMGL** 0  
**INEWG** 0  
**IDIAG** 0  
**IFDS** 0  
**MTYPEI** 3  

**NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS** 0  

**BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >-0 OR <0 )**

**MSEQ** 2  
**MGFLAG** 0  
**MICNSF** 0  
**MIT** 0  

**NCYC** 400  
**MGLEV** 0  
**NEMGL** 0  

**MIT1** 0  
**MIT2** 0  
**MIT3** 0  

**PRINT OUT:**

**BLOCK ISTART IEND IINC JSTART JEND JINC KSTART KEND KINC**

1 1 85 1 1 23 1 1 1 1
**Flared Cone (Laminar)**

```
A50-FLARED-CONE AT LOW-RE RUN-17 (AFFDL-TR-65-199)

binary grid file
'/scr2/rosen/a50/a50.grd'

binary restart file
'/scr2/rosen/a50/a50.bin'

binary plot3d grid file
'/scr2/rosen/a50/a3.plg'

binary plot3d flowfield file
'/scr2/rosen/a50/a3.plg'

Liu & Vinokur binary equilibrium air coefficient file
'/scr2/rosen/cfl3dn/liu/liubsr.cof'

primary output file
'/scr2/rosen/a50/a50.out'

fixi/fixj output file
'/scr2/rosen/a50/a50.fix'

wing pressure output file
'/scr2/rosen/a50/a50.wng'

secondary output file
'/scr2/rosen/a50/a50.sec'

flowfield output file
'/scr2/rosen/a50/a50.prt'

unsteady cp output file
'/scr2/rosen/a50/a50.ucp'

XMACH  ALPHA  BETA  REUE, MII  TINF, DR  ISND  C2SPE
16.930  0.000  0.0  0.197600  83.73  1  6.329870

SREF  CREF  BREF  XMC  YMC  ZMC
1.0  1.0  1.0  1.0  0.0  0.0

IGAS  GAMMA  RGAS  PRGAS  SCALEX  SCALET  SCALEM
3  1.4  1715.6  0.72  0.0  0.0  0.0

DT  IREST  IFLAGTS  FMAX  IUNST  RFREQ  ALPHAU  CLOC
-0.001  0  300  10.00  0  0.00000  0.00000  0.00000

NGRID  NPRINT  NWREST
1  0  100

NCG  IEM  IADVANCE  IFORCE  IMESH  IVISC(I)  IVISC(J)  IVISC(K)
1  0  0  0  1  0  0  0

IDIM  JDIM  KDIM  ITE1  ITE2  JTE1  JTE2
97  19  45  5  97  1  19

INEWG  IGRIDC  IS  JS  KS  IE  JE  KE
1  0  0  0  0  0  0  0

IDIA(1)  IIDI(1)  IIDIA(1)  IFLAG(1)  IFILM(I)  IFILM(J)  IFILM(K)
0  0  0  0  2  2  2

IFDS(I)  IFDS(J)  IFDS(K)  RKAP0(I)  RKAP0(J)  RKAP0(K)
1  1  0  0.33333  0.33333  0.33333

MTPYE(1)  MTPYE(2)  MTPYE(3)  MTPYE(4)  MTPYE(5)
67  67  1  1  67  77

NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
0

BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >=0 OR <0 )
BLCK1  INS  JLST  KST  IND  JND  KND  IND1  IND2  MSEQ  MFLAG  ICONSF  MIT  NGAM
2  0  0  0  0  0

NCYC  MGLEVEL  NEMGL  NITFO
300  01  00  00

000  01  00  00

MIT1  MIT2  MIT3  MIT4  MIT5
01  01  01  01  01

01  01  01  01  01

PRINT OUT:
BLOCK  ISTART  IEND  IINC  JSTART  JEND  JINC  KSTART  KEND  KINC
1  1  97  1  1  19  1  1  1  1

46
```
Flared Cone (Turbulent)

A50-FLARED-CONE AT HIGH-RE RUN-32 (AFFDL-TR-65-199)
binary grid file
'./scr/rosen/a32/a32.grd'
binary restart file
'./scr/rosen/a32/a3.bin'
binary plot3d grid file
'./scr/rosen/a32/a3.plg'
binary plot3d flowfield file
'./scr/rosen/a32/a3.plq'
Liu & Vinokur binary equilibrium air coefficient file
'liu/liubsr.cof'
primary output file
'./scr/rosen/a32/a3.out'
fixi/fixj output file
'./scr/rosen/a32/a3.fix'
wing pressure output file
'./scr/rosen/a32/a3.wng'
secondary output file
'./scr/rosen/a32/a3.sec'
flowfield output file
'./scr/rosen/a32/a3.prt'
unsteady cp output file
'./scr/rosen/a32/a3.ucp'

XMACH  ALPHA  BETA  REUE,MIL  TINE,DR  ISND  C2SPE
  7.850  0.000  0.0  4.697000  130.2  1  4.070661
SREF  CREF  BREF  XMC  YMC  ZMC
  1.0  1.0  1.0  1.0  0.  0.
IGAS  GAMMA  RGAS  PRGAS  SCALEX  SCALEY  SCALEZ
  3.0  1.4  1715.6  0.72  0.0  0.0  0.0
DT  IREST  IFLAGTS  FMAX  IUNST  RFREQ  ALPHAU  CLOC
-0.010  0  300.0  10.00  0  0.00000  0.00000  0.00000
NGRID  NPLOT3D  NPRINT  NWREST
  2  0  0  100
NCG  IEM  IADVANCE  IFORCE  IMESH  IVISC(I)  IVISC(J)  IVISC(K)
  1  0  0  0  1  1  0  0  0
  1  0  0  0  1  0  0  0  1
IDIM  JDIM  KDIM  ITE1  ITE2  JTE1  JTE2
  19  45  5  5  1  19
93  19  45  1  93  1  19
INEWG  IGRIDC  IS  JS  KS  IE  JE  KE
  1  0  0  0  0  0  0  0
  1  0  0  0  0  0  0  0
IDIGA(I)  IDIGA(J)  IDIGA(K)  IFLIM(I)  IFLIM(J)  IFLIM(K)
  0  0  0  2  2  2
  0  0  0  2  2  2
IFDS(I)  IFDS(J)  IFDS(K)  RKAP0(I)  RKAP0(J)  RKAP0(K)
  1  1  0.33333  0.33333  0.33333
  1  1  0.33333  0.33333  0.33333
MTYPEI(1)  MTYPEI(2)  MTYPEJ(1)  MTYPEJ(2)  MTYPEK(1)  MTYPEK(2)
  67  67  1  1  67  77
  67  67  1  1  67  77

NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
1
BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF (+>0 OR <0)
1
BLCK1  IST  JST  KST  IND  JND  KND  IND1  IND2  BLCK2  IST  JST  KST  IND  JND  KND  IND1  IND2
  1  5  1  1  5  19  45  2  3  2  1  1  1  1  19  45  2  3
MSEQ  MGFLAG  ICONSF  MIT  NGAM
  2  0  0  0  0  1
NCYC  MGLEV  NEMGL  NITFO
  300  01  00  000
  000  01  000
MIT1  MIT2  MIT3  MIT4  MIT5
  01  01  01  01  01
  01  01  01  01  01

47
Laminar Corner Flow

symmetric wedge corner : liu, lam

binary grid file
'./scr2/rosen/corner/lam.grd'
binary restart file
'./scr2/rosen/corner/l3.bin'
plot3d binary grid file
'./scr2/rosen/corner/l3.plg'
plot3d binary flowfield file
'./scr2/rosen/corner/l3.plq'
Liu & Vinokur binary equilibrium air coefficient file
'./liu/liubsr.cof'
primary output file
'./scr2/rosen/corner/l3.out'
fixi/fixj output file
'./scr2/rosen/corner/l3.fix'
wing pressure output file
'./scr2/rosen/corner/l3.wng'
secondary output file
'./scr2/rosen/corner/l3.sec'
flowfield output file
'./scr2/rosen/corner/l3.prt'
unsteady cp output file
'./scr2/rosen/corner/l3.ucp'

XMACH | ALPHA | BETA | REUE,MIL | TINF,DK | ISND | C2SPE
--- | --- | --- | --- | --- | --- | ---
3.00 | 0.000 | 0.0 | 3.07 | 105.0 | 1 | 2.8
SREF | CREF | BREF | XMC | YMC | ZMC | S
1.0000 | 1.0000 | 1.0000 | 0.0 | 0.0 | 0.0 | 0.0
IGAS | GAMMA | RGAS | PRGAS | SCALEX | SCALET | SCALEM | S
3 | 1.4 | 286.9 | 0.72 | 1.0 | 1.0 | 1.0
DT | IREST | IFLAGTS | FMAX | IUNST | RFREQ | ALPHAU | CLOC | S
-0.010 | 0 | 300 | 10.00 | 0 | 0.00000 | 0.00000 | 0.00000 | 0
NGRID | NPLOT3D | NPRINT | NWREST | S
1 | 0 | 0 | 300 | S
NCG | NEM | IADVANCE | IFORCE | IMESH | IVISC(I) | IVISC(J) | IVISC(K) | S
0 | 0 | 0 | 0 | 0 | 0 | 1 | 1 | S
IDIM | JDIM | KDIM | ITE1 | ITE2 | JTE1 | JTE2 | S
2 | 121 | 121 | 1 | 1 | 1 | 121 | S
INEWG | IGRIDC | IS | JS | KS | IE | JE | KE | S
1 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | S
IDIAG(I) | IDIAG(J) | IDIAG(K) | IFLIM(I) | IFLIM(J) | IFLIM(K) | S
0 | 0 | 0 | 0 | 0 | 0 | S
IFDS(I) | IFDS(J) | IFDS(K) | RKAP0(I) | RKAP0(J) | RKAP0(K) | S
1 | 1 | 1 | 0.33333 | 0.33333 | 0.33333 | S
MTPYE1(1) | MTPYE1(2) | MTPYEJ(1) | MTPYEJ(2) | MTPYEK(1) | MTPYEK(2) | S
1002 | 1002 | 1004 | 1002 | 1004 | 1002 | S
NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS | S
0 | S
BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >=0 OR <0 ) | S
BLCK1 | 1ST | JST | KST | IND | JND | KND | IND1 | IND2 | S
1 | 0 | 0 | 0 | 0 | 0 | S
MSEQ | MGFLAG | ICONSF | MTT | NGAM | S
1 | 0 | 0 | 0 | 0 | S
NCYC | MGLEVG | NEMGL | NITFO | S
300 | 01 | 00 | 000 | S
MIT1 | MIT2 | MIT3 | MIT4 | MIT5 | S
01 | 01 | 01 | 01 | 01 | S
symmetric wedge corner : liu,turbulent
binary grid file
'/scr2/rosen/corner/turb3.grd'
binary restart file
'/scr2/rosen/corner/t3.bin'
plot3d binary grid file
'/scr2/rosen/corner/t3.plg'
plot3d binary flowfield file
'/scr2/rosen/corner/t3.plq'
Li & Vinokur binary equilibrium air coefficient file
'liu/liubsr.cof'
primary output file
'/scr2/rosen/corner/t3.out'
fixi/fixj output file
'/scr2/rosen/corner/t3.fix'
wing pressure output file
'/scr2/rosen/corner/t3.wng'
secondary output file
'/scr2/rosen/corner/t3.sec'
flowfield output file
'/scr2/rosen/corner/t3.prt'
unsteady cp output file
'/scr2/rosen/corner/t3.ucp'

XMACH ALPHA BETA REUE,MIL TINF,DK ISND C2SPE
3.00 0.000 0.0 3.2189 105.0 1 2.8
SREF CREF BREF XMC YMC ZMC
1.0000 1.0000 1.0000 0.0 0.0 0.0
IGAS GAMMA RGAS PRGAS SCALEX SCATEL SCALEM
3 1.4 286.9 0.72 1.0 1.0 1.0
DT IREST IFLAGS FMAX IUNST RFREQ ALPHAU CLOC
-0.010 0 300 10.00 0 0.00000 0.00000 0.00000
NGRID NPLOT3D NPRINT NWREST
1 0 300
NCC IEM IADVANCE IFORCE IMESH IVISC(I) IVISC(J) IVISC(K)
0 0 0 0 0 0 0 1 1
IDIM JDIM KDIM ITE1 ITE2 JTE1 JTE2
2 121 121 1 2 1 121
INEWG IGRIDC IS JS KS IE JE KE
1 0 0 0 0 0 0 0
IDIAG(I) IDIAG(J) IDIAG(K) IFLIM(I) IFLIM(J) IFLIM(K)
0 0 0 2 2 2
IFDS(I) IFDS(J) IFDS(K) RKAP0(I) RKAP0(J) RKAP0(K)
1 1 1 0.33333 0.33333 0.33333
MYPEJ(1) MYPEJ(2) MYPEJ(1) MYPEK(1) MYPEK(2)
1002 1002 1004 1002 1004

NUMBER OF BLOCK INTERFACE BOUNDARY CONDITIONS
0

BLOCK INTERFACE BOUNDARY CONDITION ON OR OFF ( >0 OR <0 )
BLCK1 IST JST AST IND JND KND IND1 IND2 BLCK2 IST JST KST IND JND KND IND1 IND2
MSEQ MGFLAG ICONSF MTT NGAM
1 0 0 0 0 0
NCYC MGLEVG NEMGL NITFO
300 01 00 000
MIT1 MIT2 MIT3 MIT4 MIT5
01 01 01 01 01
An upwind 3-D finite volume Navier-Stokes code is modified to facilitate modeling of complex geometries and flow fields presented by proposed National Aero-Space Plane concepts. Code enhancements include an equilibrium air model, a generalized equilibrium gas model, and several schemes to simplify treatment of complex geometric configurations. The code is also restructured for inclusion of an arbitrary number of independent and dependent variables. This latter capability is intended for eventual use to incorporate nonequilibrium/chemistry gas models, more sophisticated turbulence and transition models, and other physical phenomena which will require inclusion of additional variables and/or governing equations. Comparisons of computed results with experimental data and with results obtained using the other methods are presented for code validation purposes. Good correlation is obtained for all of the test cases considered, indicating the success of the current effort. This work was conducted at the NASA Langley Research Center, during participation in the NASA/Industry Fellowship Program for the National Aero-Space Plane.