

NASA-CR-176903  
19860020300

**A Reproduced Copy**  
**OF**

*NASA CR-176,903*

---

Reproduced for NASA  
*by the*  
**NASA Scientific and Technical Information Facility**

**LIBRARY COPY**

MAY 4 1987

LANGLEY RESEARCH CENTER  
LIBRARY, NASA  
HAMPTON, VIRGINIA



NF01685

*21 pages*

J. C. Tennant  
April 1984

(NASA-CR-176903) [NUMERICAL COMPUTATION OF  
VISCOUS FLOW AROUND BODIES AND WINGS MOVING  
AT SUPERSONIC SPEEDS] Final Report, 15 Nov.  
1971- 31 Mar. 1984 (Iowa State Univ. of  
Science and Technology) 21 p

N86-29772

Unclas  
43269

CSCL 01A G3/02

Final Report  
NASA Grant N5216-09-238  
November 15, 1971 - March 31, 1984

Submitted to:

Director of Research and Special Administration  
Aerospace Research Center  
Iowa State University

NS-301 Ames 61-83  
ERI Project 730  
CRD-730

Engineering Research Institute  
Iowa State University  
Ames, Iowa 50011

#  
N86-29772

Submitted to:

National Aeronautics and Space Administration  
Ames Research Center  
Moffett Field, California

ISU-ERI-Ames-84483  
ERI Project 930  
CFD-9

Final Report

NASA Grant NGR 16-002-038  
November 15, 1971 - March 31, 1984

John C. Tannehill, Principal Investigator

Department of Aerospace Engineering  
and Computational Fluid Dynamics Institute



engineering  
research institute

iowa state university

## TABLE OF CONTENTS

	page
INTRODUCTION	1
DESCRIPTION OF PERFORMED RESEARCH	2
1. Development of equilibrium air curve fits	2
2. Computation of hypersonic rarefied leading edge flows	3
3. Computation of 2-D and 3-D blunt body laminar flows with an impinging shock	5
4. Development of a two-dimensional or axisymmetric, real gas, blunt body code	7
5. Study of an over-relaxation procedure for the MacCormack finite-difference scheme	8
6. Computation of 2-D blunt body turbulent flows with an impinging shock	8
7. Computation of supersonic viscous flow over delta wings at high angles of attack	10
8. Development of a generalized PNS code	12
9. Computation of the Space Shuttle Orbiter flowfield	13
CONCLUDING REMARKS	14
REFERENCES	16

## INTRODUCTION

This final report summarizes the research accomplished under NASA Grant NGR 16-002-038 during the entire funded period which extended from November 15, 1971 to March 31, 1984. The total funding amounted to \$261,160. The Technical Officers for this grant were the late John V. Rakich and Joe G. Marvin of NASA Ames Research Center.

The research performed during the grant period can be divided into the following major areas:

1. Development of equilibrium air curve fits
2. Computation of hypersonic rarefied leading edge flows
3. Computation of 2-D and 3-D blunt body laminar flows with an impinging shock
4. Development of a two-dimensional or axisymmetric real gas blunt body code
5. Study of an over-relaxation procedure for the MacCormack finite-difference scheme
6. Computation of 2-D blunt body turbulent flows with an impinging shock
7. Computation of supersonic viscous flow over delta wings at high angles of attack
8. Development of a generalized PNS code
9. Computation of the Space Shuttle Orbiter flowfield

These major areas of research are discussed separately in the next section.

## DESCRIPTION OF PERFORMED RESEARCH

1. Development of equilibrium air curve fits

The thermodynamic and transport properties of equilibrium air are required for the numerical computation of flowfields around bodies in high-speed flight. In particular, when using a finite-difference technique to solve the unsteady Navier-Stokes equations in conservative form, it becomes necessary to determine the thermodynamic properties as a function of density ( $\rho$ ) and internal energy ( $e$ ). This requirement led to the present study in which two different approaches were devised for the case of equilibrium air. In the first approach, the NASA-Ames RGAS program was modified<sup>1</sup> to allow density and internal energy to be the independent variables. This approach permits a very accurate determination of the thermodynamic properties of air. Unfortunately, the table-lookup feature of the RGAS program is too cumbersome to be effectively employed on advanced computers. For this reason, and also to reduce computation time on conventional serial computers, simplified curve fits were developed<sup>2-3</sup> in the second approach.

The simplified curve fits were constructed using Grabau-type transition functions. A transition function of this type can be used to smoothly connect two surfaces together with or without a line of inflection between them. The coefficients in the transition functions were determined using a least squares computer program to fit the data from the NASA RGAS program. The resulting curve fits include the following correlations:

$$p = p(e, \rho)$$

$$T = T(p, \rho)$$

$$T = T(e, \rho)$$

$$\mu = \mu(e, \rho)$$

$$a = a(e, \rho)$$

$$k = k(e, \rho)$$

$$h = h(p, \rho)$$

The ranges of validity for these curve fits are the same as the NASA RGAS program, namely, temperatures up to 25,000°K and densities from  $10^{-7}$  to  $10^{-3}$  amagats (i.e.,  $\rho/\rho_0$ ). The maximum percentage differences between the simplified curve fits and the RGAS program for the primary variables  $p = p(e, \rho)$  and  $h = h(p, \rho)$  are 4.7% and 4.6%, respectively.

## 2. Computation of hypersonic rarefied leading edge flows

The hypersonic leading edge problem has been the subject of a considerable number of both experimental and theoretical investigations over the past three decades. The theoretical investigations can be divided into two groups, depending on whether a kinetic theory approach or a continuum flow approach is used. Prior to the present study, the continuum approach was primarily based on solving simplified forms of the complete Navier-Stokes equations. The accuracy of these solutions was somewhat questionable because of the neglected terms. In the present study, the hypersonic rarefied flow near the sharp leading edge of a flat plate was computed<sup>4-6</sup> using a finite-difference solution of the complete unsteady Navier-Stokes equations. McCormack's original explicit, second-order accurate, finite-difference scheme was used to solve the equations at each interior grid point. The computational region extended from the freestream ahead of the plate to the strong-interaction

regime. Both wall slip and temperature jump were included in the calculations. The numerical results compared favorably with experimental data and Monte Carlo simulations. In addition, it was shown for the first time that a numerical solution of the Navier-Stokes equations predicts that the wall pressure asymptotically approaches the free-molecule limit at the leading edge.

In addition to this study, a related investigation was conducted<sup>7-8</sup> to determine if a numerical solution of the complete Burnett equations would yield more accurate results for the hypersonic leading edge problem. The Burnett equations are a higher-order set of equations (in terms of Knudsen number) than the Navier-Stokes equations, and thus it might be reasonable to assume that they would yield better results than the Navier-Stokes equations in rarefied regions. However, it is believed by many that while the Burnett equations can be used to improve the solution in regions where the Navier-Stokes equations give good accuracy, it is not safe to assume that they can be used to progress into more rarefied regions where the Navier-Stokes equations are invalid. The present investigation was undertaken to test this latter assertion. The complete Burnett equations were integrated using MacCormack's explicit finite-difference scheme. The resulting solution was compared with experimental data, a numerical solution of the complete Navier-Stokes equations, and a Monte Carlo simulation. Based on this comparison, it was evident that the Burnett equations in conjunction with Schamberg's boundary condition gave a much less accurate description of the rarefied flowfield near the leading edge of the flat plate. This result tends to confirm the belief



that the Burnett equations cannot be used to progress into rarefied regions where the Navier-Stokes equations are already invalid.

3. Computation of 2-D and 3-D blunt body laminar flows with an impinging shock

An extraneous shock wave impinging on a blunt body in a hypersonic flow has been observed to greatly increase both the heat transfer rate and pressure near the impingement point. Flowfields of this type occur on the Space Shuttle and other maneuverable re-entry vehicles. The intense heating and high pressures occur over a small region where a disturbance, originating at the intersection of the impinging shock and bow shock, strikes the body. The disturbance may be a free shear layer, a supersonic jet, or a shock wave depending on the strength and location of the impinging shock and the shape of the body. Edney has described six different types of shock interference patterns which can occur.

Because of the very complicated nature of shock impingement flowfields, previous attempts at predicting the maximum heating rates and pressures were limited to semiempirical approaches. In the present study, this empiricism was eliminated by numerically computing the entire shock impingement flowfield. This was accomplished by using a "time-dependent," finite-difference method to solve the complete set of compressible Navier-Stokes equations for a laminar flow. The major reason for using the "time-dependent" method is that the resulting unsteady Navier-Stokes equations are a mixed set of hyperbolic-parabolic equations for both subsonic and supersonic flows. As a result, a very complicated shock impingement flowfield which contains both subsonic and supersonic regions

can be calculated as an initial-value problem. An additional advantage is that since the complete Navier-Stokes equations are solved, all shock waves, shear layers, jets, and the wall boundary layer are "captured" automatically in the solution without prior knowledge of their location or even existence.

For very low Reynolds number flows, the entire shock impingement flowfield, including the bow shock, can be "captured" using this so-called "shock-capturing" approach. This approach was used for computations made during the early stages of the present study.<sup>9</sup> However, it was found that for higher Reynolds number flows it was not practical to "capture" the bow shock because of the numerical difficulties associated with the large gradients at the bow shock. Instead, it was found to be more convenient to treat the bow shock as a discontinuity, across which the Rankine-Hugoniot equations could be applied, while leaving the boundary layer and interaction regions to be captured as before. This latter approach is the so-called "shock-fitting" method.

MacCormack's original, explicit, finite-difference algorithm was used to solve the Navier-Stokes equations at each interior grid point. Initially, the blunt body flow without the impinging shock was computed. The impinging shock was then introduced into the flowfield by specifying its location and strength. The computation was then restarted and continued until the final "steady-state" solution was reached.

Both two-dimensional and three-dimensional blunt body flows with planar impinging shocks were computed. In the two-dimensional cases, the planar impinging shock was parallel to the centerline of the infinite cylinder. Early 2-D results<sup>9-11</sup> were qualitatively compared with existing

3-D experiments because of the lack of suitable 2-D experiments. Later, it became possible to compare 2-D computations with 2-D experiments conducted by J. W. Keyes at NASA Langley. The numerical results<sup>12</sup> were in excellent agreement with the 2-D experimental values of shock shape, pressures, and heat transfer rates. The 3-D computations,<sup>13-15</sup> showed reasonable agreement with the experimental results. These computations were hampered by the lack of grid points which was the direct result of computer storage limitations.

4. Development of a two-dimensional or axisymmetric, real gas, blunt body code

A two-dimensional or axisymmetric, viscous, real gas, blunt body code was developed<sup>3</sup> in this study. The complete Navier-Stokes equations in curvilinear, orthogonal coordinates were employed as the governing equations. The use of orthogonal coordinates simplifies the application of body boundary conditions and the determination of a starting line for supersonic calculations downstream of the nose. The steady-state solution to the governing equations is obtained using the original, explicit MacCormack scheme. The bow shock is treated as a discontinuity using the Rankine-Hugoniot relations.

Both perfect gas and equilibrium air calculations can be performed with this code. The equilibrium air flow properties are introduced into the calculation procedure via the simplified curve fits described earlier. This code was tested by calculating both perfect gas and equilibrium air flows over standard sphere test cases.

5. Study of an over-relaxation procedure for the MacCormack finite-difference scheme

An over-relaxation procedure was developed<sup>16-17</sup> for the MacCormack finite-difference scheme in order to reduce the computation time required to obtain a steady-state solution. The implementation of this acceleration procedure to an existing computer program using the original explicit MacCormack method is extremely simple and does not require additional storage. The over-relaxation procedure does not alter the steady-state solution, which is second-order accurate. The method was first applied to Burgers' equation. A stability condition and an expression for the increase in the rate of convergence was derived. The method was then applied to the calculation of the hypersonic viscous flow over a flat plate using the complete Navier-Stokes equations, and the inviscid flow over a wedge using the Euler equations. Reductions in computing time by factors of 3 and 1.5, respectively, were obtained by the over-relaxation procedure.

6. Computation of 2-D blunt body turbulent flows with an impinging shock

During this study, a computer program for computing two-dimensional turbulent blunt body flows with an impinging shock wave was developed. This code utilizes the time-dependent, time-averaged Navier-Stokes equations with algebraic eddy viscosity and turbulent Prandtl number models employed in the expressions for shear stress and heat flux. For the turbulent boundary layer, the two-layer mixing length model of Cebeci and Smith is used. This model contains a correction for the streamwise pressure

gradient which occurs in blunt body flows. For the turbulent shear layer a modified Prandtl mixing length model is used. In the region where the shear layer impinges on the boundary layer, a combination of the two models is employed.

Originally, the standard, unsplit, MacCormack finite-difference scheme was used to integrate the governing equations. This proved unsuccessful, however, because the allowable time step is proportional to the grid spacing. Since, a very fine mesh is required to properly resolve the turbulent boundary layer it was found that excessively long computer times were necessary to obtain a steady-state solution. Because of this difficulty, the Beam-Warming implicit scheme was incorporated into the turbulent code. This scheme increases the computation time per step by a factor of 1.6 but permits a time step which is many times greater than the time step of the previous explicit scheme. Thus, steady-state turbulent solutions can be obtained with substantially fewer iterations. The present implicit scheme is noniterative, spatially factored, and employs Euler time differencing which makes the scheme second-order accurate in space but first-order accurate in time. For the present steady flows, time accuracy is not important. The irregular-shaped bow shock is treated as a discontinuity across which the Rankine-Hugoniot equations are applied. The code was used<sup>18-19</sup> to compute a Type III turbulent shock interference flowfield. The computed results were in excellent agreement with the experimental values of shock shape, wall pressures, and heat transfer rates.

In a related study, the present code (with the MacCormack finite-difference scheme) was used<sup>20</sup> to calculate the turbulent hypersonic flow over severely indented nose tips. The computed results were compared with experimental data from wind-tunnel tests. Qualitative agreement was obtained for the surface pressure distribution and flowfield structure, including the separated bubble in the indented region.

7. Computation of supersonic viscous flow over delta wings at high angles of attack

Two complementary procedures were developed<sup>21-23</sup> to calculate the supersonic, viscous flow over conical shapes at large angles of attack, with application to cones and delta wings. In the first approach, the flow was assumed to be conically self-similar. This approximation is suggested by the results of experiments for supersonic flows around conical bodies and wings. The resulting Navier-Stokes equations are solved at a given Reynolds number with a time-marching explicit finite-difference algorithm. These equations are referred to as the conical Navier-Stokes equations. These equations had been used previously to calculate the flow around cones at high angles of attack and delta wings with supersonic leading edges. However, in those calculations the bow shock was captured and the body shapes were limited by a rather restrictive geometry. The present method treats the shock as a sharp discontinuity and allows for a completely general cross-sectional shape and distribution of the finite-difference grid points.

In the second approach, only the streamwise viscous derivatives were neglected in the steady Navier-Stokes equations. The resulting equations

are referred to as the "parablized" Navier-Stokes (PNS) equations. The PNS equations are hyperbolic-parabolic with respect to the streamwise direction. This permits the solution to be marched downstream from a given initial data surface provided that certain conditions are met. Previous investigators had used this approach in conjunction with an implicit, iterative finite-difference scheme to compute the supersonic flow over circular cones at angle of attack. In the present study, a new implicit, noniterative algorithm was developed which provides better computational efficiency than previous algorithms and is not restricted to conical shapes.

The two approaches were used to calculate the supersonic flow over a cone at high angle of attack and a delta wing with a sharp subsonic leading edge. Both of the approaches accurately predicted the viscous and inviscid features of the two flowfields. In particular, the location of the main vortex above the delta wing was correctly predicted.

During this study, an important fact concerning the mathematical behavior of the PNS equations was discovered. It was found that the PNS equations remain hyperbolic-parabolic only if a fraction ( $\omega$ ) of the streamwise pressure gradient is retained in the subsonic portions of the flowfield such as boundary layers. This fraction is given by

$$\omega = \frac{\gamma M_x^2}{1 + (\gamma - 1) M_x^2}$$

where  $M_x$  is the local streamwise Mach number. Using this knowledge, a technique was developed to prevent departure solutions which are attributable to the presence of the streamwise pressure gradient term in the streamwise momentum equation.

#### 8. Development of a generalized PNS code

The original parabolized Navier-Stokes (PNS) computer code, described in the previous section, was modified<sup>24</sup> to permit the calculation of supersonic viscous flows around arbitrary body shapes at high angles of attack. The previous code advanced the solution using axis-normal surfaces which allowed for most body geometries. However, for blunted bodies with large surface slope, the axial component of velocity in the inviscid part of the shock layer may be subsonic which prevented the previous code from computing this case. Therefore, it was necessary to generalize the code to march with solution surfaces that are more nearly normal to the local body surface. This was done in a perfectly general manner with a non-orthogonal three-dimensional coordinate frame.

The generalized PNS code was used to compute the laminar flow over a 70° sweep slab delta wing having a spherical nose and a cylindrical leading edge. The body shape and flow conditions were chosen to match the experiments of Bertram and Everhart. Calculations were performed at angles of attack up to 41.5° and Mach numbers of 6.8 and 9.6. The computed shock shapes, surface pressures, and heat transfer coefficients are in excellent agreement with the experiment. The results of these computations are reported in Refs. 24-26.

The complex turbulent flow in a three-dimensional corner was also computed using the generalized PNS code. Two simple eddy viscosity models were used for the turbulence modeling. The computed results were compared with the experiment of Peake and the complete Navier-Stokes computations of Horstman and Hung. Detailed comparisons of surface pressure, skin



friction, surface flow angle, velocity profiles and flow angle profiles show reasonable agreement. The results of these computations are presented in Refs. 27-28.

#### 9. Computation of the Space Shuttle Orbiter flowfield

The calculation of the complete inviscid-viscous flow around the Space Shuttle Orbiter has been the aim of recent research efforts. While inviscid flow solutions and other solution techniques that couple the inviscid solution with the matching boundary-layer analysis have resulted in limited solutions for the windward side of Shuttle-like bodies, they have failed in general to predict the correct solution on the lee side and near the wing root region. In the present study, the laminar flow around the Space Shuttle Orbiter forebody was computed using the generalized PNS code discussed previously. This generalized code is ideally suited for the complex flow that develops around the leading edge and in the vicinity of the wing-body juncture of the Orbiter. The code will accept the initial solution on any starting data surface and it will allow the marching to proceed from one solution surface to another without any restrictions on the orientation of these solution surfaces. The only condition that must be satisfied is that the velocity component normal to the solution surface must be supersonic outside the boundary layer.

The initial solution for the nose part of the Orbiter geometry was obtained with a 3-D time-dependent Navier-Stokes solver. It was necessary to employ a wind axis oriented coordinate system to obtain the initial solution with the time-dependent code. A grid point clustering scheme<sup>29</sup>

was employed to accurately describe the body shape by clustering points at the wing tip and at the wing-body juncture. The computed heat transfer coefficients, pressure coefficients, and shock shapes compared favorably with the available experimental data for  $0^\circ$  and  $30^\circ$  angle of attack. The results of these computations were reported in Refs. 26, 30, and 31.

Work was initiated to modify the PNS code to permit real gas (equilibrium air) calculations. This modification was complicated by the presence of the Jacobians which must be evaluated for the finite-difference scheme. The real gas PNS code is currently being used to compute a Space Shuttle Orbiter flowfield. The results of this computation will be compared with STS-3 flight data in a forthcoming paper.<sup>32</sup>

#### CONCLUDING REMARKS

The research performed during this grant is described by the 32 documents listed in the next section. This documentation consists of:

- 8 Journal articles
- 11 Technical papers presented and preprinted
- 5 NASA Contractor Reports/Technical Memorandums
- 4 Ph.D. dissertations
- 4 M.S. theses

In addition, the present grant supported the following students as Research Assistants at Iowa State University:

1. Robert A. Mohling
2. Terry L. Holst
3. Paul H. Mugge

4. G. Richard Eisler
5. Yvon C. Vigneron
6. J.-Antoine Désidéri
7. Ethiraj Venkatapathy
8. Iho K. Tree
9. Dinesh K. Prabhu
10. David C. Stookesberry

The late John V. Rakich of NASA Ames Research Center served as Technical Monitor for this grant until his death on April 9, 1983. The research in this grant was greatly enhanced by his inspiration and guidance. During the remainder of the grant period, Joe C. Marvin of NASA Ames Research Center served as the Technical Monitor.

## REFERENCES

1. Tannehill, J. C. and Mohling, R. A., "Development of Equilibrium Air Computer Programs Suitable for Numerical Computation Using Time-Dependent or Shock-Capturing Methods," NASA CR-2134, September 1972.
2. Tannehill, J. C. and Mugge, P. H., "Improved Curve Fits for the Thermodynamic Properties of Equilibrium Air Suitable for Numerical Computation using Time-Dependent or Shock-Capturing Methods," NASA CR-2470, October 1974.
3. Vigneron, Y. C., "Hypersonic Viscous Flow of Equilibrium Air Around a Blunt Body," M.S. Thesis, Iowa State University, Ames, Iowa, 1976.
4. Mohling, R. A., "Numerical Computation of the Hypersonic Rarefied Flow Past the Sharp Leading Edge of a Flat Plate," Ph.D. Dissertation, Iowa State University, Ames, Iowa, 1972.
5. Tannehill, J. C., Mohling, R. A., and Rakich, J. V., "Numerical Computation of the Hypersonic Rarefied Flow Near the Sharp Leading Edge of a Flat Plate," AIAA Paper 73-200, presented at the AIAA 11th Aerospace Sciences Meeting, Washington, D.C., January 1973.
6. Tannehill, J. C., Mohling, R. A., and Rakich, J. V., "Numerical Computation of Hypersonic Viscous Flow over a Sharp Leading Edge," AIAA Journal, Vol. 12, No. 2, February 1974, pp. 129-130.
7. Eisler, G. R., "Numerical Computation of the Hypersonic Leading Edge Problem Using the Burnett Equations," M.S. Thesis, Iowa State University, Ames, Iowa, 1974.
8. Tannehill, J. C. and Eisler, G. R., "Numerical Computation of the Hypersonic Leading Edge Problem Using the Burnett Equations," Physics of Fluids, Vol. 19, No. 1, January 1976, pp. 9-15.
9. Tannehill, J. C. and Holst, T. L., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," NASA CR-138594, February 1974.
10. Tannehill, J. C., Holst, T. L., and Rakich, J. V., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," AIAA Paper 75-154, presented at the AIAA 13th Aerospace Sciences Meeting, Pasadena, California, January 1975.
11. Tannehill, J. C., Holst, T. L., and Rakich, J. V., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," AIAA Journal, Vol. 14, No. 2, February 1976, pp. 204-211.

12. Tannehill, J. C., Holst, T. L., Rakich, J. V., and Keyes, J. W., "Comparison of a Two-Dimensional Shock Impingement Computation with Experiment," AIAA Journal, Vol. 14, No. 4, April 1976, pp. 539-541.
13. Holst, T. L., Tannehill, J. C., and Rakich, J. V., "Numerical Computation of Viscous Blunt Body Flows with a Planar Impinging Shock," presented at NASA Conference on Aerodynamic Analyses Requiring Advanced Computers, Hampton, Virginia, NASA SP-347, March 1975.
14. Holst, T. L., "Numerical Computation of Three-Dimensional Blunt Body Flow Fields with an Impinging Shock," Ph.D. Dissertation, Iowa State University, Ames, Iowa, 1975.
15. Holst, T. L. and Tannehill, J. C., "Numerical Computation of Three-Dimensional Blunt Body Flow Fields with an Impinging Shock," NASA CR-143312, July 1975.
16. Désidéri, J.-A., "Over-Relaxation of a Finite-Difference Technique to Solve Fluid Flow Problems," M.S. Thesis, Iowa State University, Ames, Iowa, 1976.
17. Désidéri, J.-A. and Tannehill, J. C., "Over-Relaxation Applied to the MacCormack Finite-Difference Scheme," Journal of Computational Physics, Vol. 23, No. 3, March 1977, pp. 313-326.
18. Tannehill, J. C., Vigneron, Y. C., and Rakich, J. V., "Numerical Solution of Two-Dimensional Turbulent Blunt Body Flows with an Impinging Shock," AIAA Paper 78-1209, presented at AIAA 11th Fluid and Plasma Dynamics Conference, Seattle, Washington, July 1978.
19. Tannehill, J. C., Vigneron, Y. C., and Rakich, J. V., "Numerical Solution of Two-Dimensional Turbulent Blunt Body Flows with an Impinging Shock," AIAA Journal, Vol. 17, No. 12, December 1979, pp. 1289-1290.
20. Rakich, J. V., Tannehill, J. C., and Vigneron, Y. C., "Navier-Stokes Calculations for Laminar and Turbulent Hypersonic Flow Over Indented Nosedtips," AIAA Paper 78-260, presented at 16th Aerospace Sciences Meeting, Huntsville, Alabama, January 1978.
21. Vigneron, Y. C., Rakich, J. V., and Tannehill, J. C., "Calculation of Supersonic Viscous Flow Over Delta Wings with Sharp Subsonic Leading Edges," AIAA Paper 78-1137, presented at AIAA 11th Fluid and Plasma Dynamics Conference, Seattle, Washington, July 1978.
22. Vigneron, Y. C., Rakich, J. V., and Tannehill, J. C., "Calculation of Supersonic Viscous Flow Over Delta Wings with Sharp Subsonic Leading Edges," NASA TM 78500, June 1978.

23. Vigneron, Y. C., "Calculation of Supersonic Viscous Flow Over Delta Wings with Sharp Subsonic Leading Edges," Ph.D. Dissertation, Iowa State University, Ames, Iowa, 1978.
24. Tannehill, J. C., Venkatapathy, E., and Rakich, J. V., "Numerical Solution of Supersonic Viscous Flow over Blunt Delta Wings," AIAA Paper 81-0049, presented at AIAA 19th Aerospace Sciences Meeting, St. Louis, Missouri, January 1981.
25. Tannehill, J. C., Venkatapathy, E., and Rakich, J. V., "Numerical Solution of Supersonic Viscous Flow over Blunt Delta Wings," AIAA Journal, Vol. 20, No. 2, February 1982, pp. 203-210.
26. Venkatapathy, E., "A Generalized Solution Technique for the Parabolized Navier-Stokes Equations," Ph.D. Dissertation, Iowa State University, Ames, Iowa, 1982
27. Tree, I. K., "Numerical Solution of Three-Dimensional Turbulent Corner Flow Using Parabolized Navier-Stokes Equations," M.S. Thesis, Iowa State University, Ames, Iowa, 1981.
28. Rakich, J. V., Tannehill, J. C., and Tree, I. K., "Comparison of Computation with Experiment," Proceedings 1980-1981 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows; Comparison of Computation and Experiment, September 1981, Stanford, California, pp. 1491-1494.
29. Venkatapathy, E., Tannehill, J. C., and Rakich, J. V., "Algebraic Grid Generation Scheme for Flow Field Around Complex Geometries," presented at AIAA 5th Computational Fluid Dynamics Conference, Palo Alto, California, June 1981.
30. Venkatapathy, E., Rakich, J. V., and Tannehill, J. C., "Numerical Solution of Space Shuttle Orbiter Flow Field," AIAA Paper 82-0028, presented at AIAA 20th Aerospace Sciences Meeting, Orlando, Florida, January 1982.
31. Rakich, J. V., Venkatapathy, E., Tannehill, J. C., and Prabhu, D., "Numerical Solution of Space Shuttle Orbiter Flowfield," Journal of Spacecraft and Rockets, Vol. 21, No. 1, January-February 1984, pp. 9-15.
32. Prabhu, D. K. and Tannehill, J. C., "Numerical Solution of Space Shuttle Orbiter Flow Field Including Real Gas Effects," AIAA Paper 84-1747, to be presented at AIAA 19th Thermophysics Conference, Snowmass, Colorado, June 1984.

END

DATE

FILMED

OCT 8 1986

**End of Document**