NASA TM-81877

NASA Technical Memorandum 81877

NASA-TM-81877 19800025208

A COMPENDIUM OF COMPUTATIONAL FLUID DYNAMICS AT THE LANGLEY RESEARCH CENTER

FOR REFERENCE

NOT TO BE TAKEN FROM THIS ROOM

PREPARED BY THE STAFF OF THE
LANGLEY RESEARCH CENTER

AUGUST 1980

LIBRARY COPY
OCT 3 1980
LANGLEY RESEARCH CENTER
LIBRARY, NASA
HAMPTON, VIRGINIA

NASA
National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

3 1176 01324 7896

PREFACE

This document contains a set of summary presentations of research in progress at the Langley Research Center in Computational Fluid Dynamics (CFD). While this document does not include all or even most of the work going on in CFD at Langley, it does display in summary form the general scope and nature of research in this area.

Computational Fluid Dynamics is, of course, an important and substantial research activity at the Langley Research Center. No single organization has the primary responsibility for the CFD effort at the Center, but rather research activities in CFD are carried out within many different organizational units as an integral part of a wide variety of research programs.

The purpose of this compendium is to help identify, through numerous summary examples, the scope and general nature of the CFD effort at Langley. This document will help inform researchers in CFD and line management at Langley of the overall CFD effort. In addition to the in-house efforts and work at ICASE and JIAFS, out-of-house CFD work supported by Langley through industrial contracts and university grants are included. Researchers were encouraged to include summaries of work in preliminary and tentative states of development as well as current research approaching definitive results.

Wayne D. Erickson
Senior Scientist, Office of the Director

Jerry C. South, Jr.
Head, Theoretical Aerodynamics Branch, STAD

Jerome T. Foughner, Jr.
Technical Assistant to the Senior Scientist

N80-33716#

CONTENTS

1. SUB/SUPERSONIC LINEAR THEORY METHODS

Page

1.0 Development of a Free Vortex Sheet Theory for Estimation of Aerodynamic Loads on Wings With Separation Induced Vortex Flows
James M. Luckring 2

1.1 Extension of the Free Vortex Filament Theory to Include Core Modeling
James F. Campbell 4

1.2 The Development of Superior Singularities for Modeling Potential Flow Aerodynamic Problems at Subsonic and Supersonic Speeds
F. A. Woodward, Charles H. Fox, Jr., and Emma Jean Landrum . . 6

1.3 Extension of Two-Dimensional Analysis and Design Method to Three Dimensions
James L. Thomas 8

1.4 Development of a Vector-Continuous Loading Concept for Aerodynamic Panel Methods
James L. Thomas 10

1.5 Rotor Body Interference
Carl E. Freeman 12

1.6 Application of Subpaneling Concepts to the Modeling of Vortex/Surface Interactions
W. Elliott Schoonover, Jr. 14

1.7 Inviscid Analysis of Tip Vortex Formation on Oscillating Wings
Warren H. Young, Jr. 16

1.8 Steady, Oscillatory, and Unsteady Subsonic and Supersonic Aerodynamics
E. Carson Yates, Jr. 18

1.9 Calculation of Lifting Pressures and Generalized Forces on a Thin Lifting Surface With Leading- and Trailing-Edge Flap-Type Controls Undergoing Harmonic Motions
H. J. Cunningham 20

1.10 Hybrid Vortex Method for Lifting Surfaces With Free-Vortex Flow
Osama A. Kandil, Li Chuan Chu, and E. Carson Yates, Jr. . . . 22

1.11 A Nonlinear Continuous-Vorticity Model
Ali H. Nayfeh and Dean T. Mook 24

2. BOUNDARY LAYERS

Page

2.0 Advanced 2-D Turbulent Boundary-Layer Separation Prediction Method
Suresh H. Goradia and Harry L. Morgan, Jr. 28

2.1 Calculation of Heating Rates on Three-Dimensional Configurations
H. Harris Hamilton II 30

3. VISCOUS/INVISCID INTERACTION

Page

3.0 Calculation of Aeroelastic Loads on Wings in Transonic Flow
Woodrow Whitlow, Jr., and Robert M. Bennett 34

3.1 Interaction of 3-D Wing Boundary-Layer Code With 3-D Transonic Inviscid Wing Codes
Craig L. Streett 36

3.2 Calculation of Transonic Flow About Airfoils With Trailing-Edge Separation
Richard W. Barnwell 38

3.3 Computation of Leading-Edge Separation Bubbles on Airfoils
Richard W. Barnwell 40

3.4 Patched Viscous/Inviscid Interaction Techniques for Predicting Transonic Flow Over Boattail Nozzles
Richard G. Wilmoth and Lawrence E. Putnam 42

3.5 Approximate Convective Heating Equations for Hypersonic Flow
E. V. Zoby, J. N. Moss, and K. Sutton 44

4. TRANSONIC POTENTIAL FLOW

Page

Steady Flows

4.0 Analysis of Transonic Flow Problems Via Parametric Differentiation
Woodrow Whitlow, Jr. 50

4.1 Analysis of Iterative Methods for the Transonic Potential Equation
John C. Strikwerda 52

4.2 Design of Shockless Transonic Airfoils by the Method of Complex Characteristics
José Sanz 54

4.3 Application and Evaluation of Transonic Theories for Designing Supercritical Fighter Wings for Transonic Maneuver
Michael J. Mann 56

4.4 Improvements in the Artificial Density Method for Computational Transonics
Jerry C. South, Jr. 58

4.5 Development of a Fully Conservative, Axisymmetric Transonic Flow Code
Lawrence L. Green 60

4.6 Development of Vector Algorithms for Computational Transonics
James D. Keller 62

4.7 Basic Research in CFD at NYU
Jerry C. South, Jr. 64

4.8 Computation of Rotational Transonic Flow Using the Stream Function
Jerry C. South, Jr. 66

4.9 Development of a Small-Disturbance Embedded-Grid Code for Transonic Wing/Body/Nacelle/Pylon Configurations
Perry A. Newman 68

4.10 Validation of a Small-Disturbance Embedded-Grid Code for Transonic Wing/Body/Nacelle/Pylon Configurations
Perry A. Newman 70

4.11 Method for Numerical Design of a Contoured Wind-Tunnel Liner for Test of a Laminar-Flow-Control System for a Yawed Supercritical Airfoil Model
Perry A. Newman and E. Clay Anderson 72

4.12 Development of a Coupled Finite-Volume/Panel Code for Analysis of LFC Swept Wing Tests
Perry A. Newman 74

4.13 Multiple Level Techniques for Computational Fluid Dynamics
D. R. McCarthy and R. C. Swanson 76

4.14 Algorithm for Solving the Full Transonic Potential Flow Equations for a Nacelle/Wing/Pylon Geometry Using the CDC Cyber-203 Computer
T. A. Reyhner and D. E. Reubush 78

4.15 A Small Disturbance Technique for Predicting the Effects of Jet Exhaust Flow on the Characteristics of Transport Airplanes
Chen Sun and William K. Abeyounis 80

Unsteady Flow

4.16 Development of Frequency Plane Perturbation Method for Transonic Unsteady Flows
Robert M. Bennett 84

4.17 Development of a Finite Element Method for Transonic Unsteady Potential Flow
Kenneth R. Kimble 86

4.18 An Evaluation of Two-Dimensional Shock Compatibility Conditions for Linear Unsteady Transonic Flow Perturbations
Woodrow Whitlow, Jr. 88

4.19 A Finite Element Method for the Periodically Oscillating Thin Airfoil
G. Fix, M. D. Gunzburger, R. A. Nicolaides, and C. Cox 90

4.20 Improved Sonic-Box Computer Programs for Calculating Transonic Aerodynamic Loads on Oscillating Wings With Thickness
Song Y. Ruo, E. Carson Yates, Jr., and Jerome G. Theisen . . . 92

5. INVISCID EULER EQUATIONS

Page

5.0 Upstream Differencing for Compressible Flow
Bram van Leer 96

5.1 Finite Difference Solution of a Galactic Flow Problem
B. van Leer, T. Zang, M. Y. Hussaini, G. D. van Albada, and W. W. Roberts 98

5.2 Computation of the Shape of Slender Streams of Fluid
James F. Geer and John C. Strikwerda 100

5.3 Study of Transonic Flow Over a Circular Cylinder Using the Euler Equations
Manuel D. Salas 102

5.4 Study of Flowfields About Asymmetric External Corners
Manuel D. Salas 104

5.5 Interaction of a Two-Dimensional Shock Wave With a Vortex
Manuel D. Salas and Sui-Kwong P. Pao 106

5.6 Calculation of Supersonic Flow Past Aircraft With Shocks
Manuel D. Salas and James C. Townsend 108

5.7 Development of a Multi-Grid Method for the Euler Equations
Stephen F. Wornom 110

5.8 Solutions to the Euler Equations Using a Streamline-Characteristics Coordinate System
Lawrence Sirovich and James C. Townsend 112

5.9 Calculation of Two-Dimensional Inlet Flow Fields in a Supersonic Free Stream
Wallace C. Sawyer 114

5.10 The Flow Resulting From the Intersection of an Oblique Shock Wave With a Cylinder Aligned With a Steady Supersonic Flow
James C. Townsend 116

5.11 Comparison of Results From a Finite Difference Code With Results From Experiments and Other Methods
Emma Jean Landrum 118

5.12 Application of a Solution on the Euler Equations to the Prediction of Static Performance of 2-D C-D Nozzles
Mary L. Mason 120

5.13 A Solution of the Euler Equations for the Flow in the Engine Nacelle-Pylon-Wing Region of Transport Airplanes
William B. Compton III 122

6. NAVIER-STOKES EQUATIONS

Page

Incompressible Flows

6.0 Boundary-Layer Resolving Difference Equations for Steady, Incompressible Navier-Stokes Equations
Thomas B. Gatski, Chester Grosch, and Milton E. Rose 128

6.1 Split-Velocity Navier-Stokes Techniques for High Reynolds Number Flows
Douglas L. Dwoyer 130

6.2 Fully Viscous Multi-Element Airfoil Solutions
Frank C. Thames 132

Compressible Flows

6.3 Navier Stokes Solution for the Dynamic Stall of Helicopter Airfoils
Warren H. Young, Jr. 136

6.4 Viscous Analysis of Tip Vortex Formation
Warren H. Young, Jr. 138

6.5 Computation of the Perpendicular Injector Flow Field in a Hydrogen Fueled Scramjet
J. Philip Drummond and Elizabeth H. Weidner 140

6.6 Numerical Analysis of the Inlet Flow of a Scramjet Engine
Ajay Kumar 142

6.7 GIM/STAR: External and Internal Flows for Hypersonic Vehicles
L. W. Spradley, J. J. Stalnaker, A. W. Ratliff,
J. L. Hunt, and J. P. Drummond 144

6.8 Mixed Spectral/Finite Difference Method for Compressible Navier-Stokes Equations
Thomas A. Zang and M. Y. Hussaini 146

6.9 Outflow Boundary Conditions for Fluid Dynamics
Alvin Bayliss and Eli Turkel 148

6.10 Computation of the Flow at a Shuttle-Type Wing Elevon Juncture
Joanne L. Walsh and John C. Strikwerda 150

6.11 Computation of the Flow in Slotted Nozzles
John C. Strikwerda 152

6.12 A Class of Implicit Difference Schemes for Compressible Navier-Stokes Equations
Milton E. Rose 154

6.13 Complete Viscous Flowfield Solutions About a Blunt Parabolic Body in a Supersonic Stream
K. J. Weilmuenster and R. A. Graves, Jr. 156

6.14 Viscous Compressible Flow About Blunt Bodies Using a Numerically Generated Orthogonal Coordinate System
Randolph A. Graves, Jr., and H. Harris Hamilton II 158

6.15 Prediction of 3-D Viscous Flows in the Vicinity of Transonic Wind-Tunnel Wall Slots
Douglas L. Dwoyer 160

6.16 Prediction of Viscous Flow in Aerodynamic Juncture Regions With the Parabolized Navier-Stokes Equations
Douglas L. Dwoyer 162

6.17 Implicit Methods for the Navier-Stokes Equations
Stephen F. Wornom 164

6.18 Vector Computer Applications Software for the Solution of the Compressible Navier-Stokes Equations
Robert E. Smith and Joan I. Pitts 166

6.19 Finite Element Techniques for High Reynolds Number Flow Prediction
A. J. Baker 168

6.20 General Aerodynamic Simulation Three-Dimensional, Compressible, Navier-Stokes Equations
Julius E. Harris 170

6.21 Unsteady Open Cavity Flow/Two-Dimensional Incompressible Navier-Stokes Equations
Joshua C. Anyiwo 172

6.22 A Two-Dimensional Navier-Stokes Solver Using Spectral Methods for Flows Over Moving Wall Geometries
R. Balasubramanian and Steven A. Orszag 174

6.23 Time-Dependent Coordinate Systems for the Numerical Solution of Fluid Flow
Liviu Lustman 176

6.24 Prediction of the Transonic Flow Over Axisymmetric Boattail Nozzles Using a Navier-Stokes Code Developed for the STAR Computer
R. C. Swanson 178

6.25 Spline Collocation for Viscous Flow Over Afterbody Configurations
S. J. Rubin and R. C. Swanson 180

6.26 A Two-Stream Solution of the Navier-Stokes Equations for Uninstalled Nozzles
Michael C. Cline and Richard G. Wilmoth 182

6.27 A 3-D Solution of the Navier-Stokes Equations for the Flow Characteristics and Performance of Nonaxisymmetric Nozzles
P. D. Thomas and Lawrence E. Putnam 184

6.28 Boundary Conditions for Subsonic Compressible Navier-Stokes Calculations
David H. Rudy and John C. Strikwerda 186

6.29 Numerical Simulation of High Reynolds Number, Three Dimensional, Compressible Flow
M. Y. Hussaini and Steven A. Orszag 188

Parabolic Approximation

6.30 Application of Parabolic Analysis to Practical Scramjet Flow Fields
John S. Evans 192

6.31 Turbulent Mixing and Reaction in Three-Dimensional Partially Elliptic Flows of Supersonic Combustors
R. Clayton Rogers 194

6.32 Blunt-Body Viscous-Shock-Layer Analysis That Includes Turbulence, Mass Injection, and Radiation
James N. Moss 196

6.33 Preliminary Thermal Analysis for Saturn Entry
E. V. Zoby and J. N. Moss 198

7. TURBULENCE EFFECTS AND TURBULENCE MODELING

Page

7.0 Turbulence Models for Supersonic Combustors
P. T. Harsha and R. B. Edelman 202

7.1 Merging of Toroidal Vortices
C. H. Liu 204

7.2 Finite-Element Solution Algorithm for Aeroacoustic Trailing Edge Flow Prediction
A. J. Baker 206

7.3 Time Dependent Numerical Solution of Turbulent Jet Flows Using Reynolds Stress Closure Models
T. B. Gatski 208

7.4 Computed Evolution of Large Scale Wave-Like Structures in Turbulent Jet Flows and Their Sound Production
P. J. Morris 210

8. VISCOUS FLOW STABILITY AND TRANSITION

Page

8.0 Calculation of 3-D Laminar Boundary Layer Transition in the Presence of Free-Stream Turbulence
E. Clay Anderson 214

8.1 Nonlinear Stability Theory and Optimum Numerics for Laminar Flow Control Analysis
Steven A. Orszag 216

8.2 Three-Dimensional, Compressible, Nonparallel, Linear Stability Theory for Laminar Flow Control Analysis
Ali H. Nayfeh 218

8.3 Two - and Three - Dimensional Boundary-Layer Stability Analysis
Mujeeb R. Malik and Steven A. Orszag 220

8.4 On the Stability of the Free Shear Layer for an Axial Symmetric Jet and Plug Flow Jets
Steven A. Orszag, Lucio Maestrello, and Stan Lamkin 222

8.5 On the Stability of Rotating Disk Flow
Mujeeb R. Malik 224

8.6 Stability of Flows Over Bodies With Suction Through Porous Strips
Ali H. Nayfeh 226

9. ATMOSPHERIC MODELING

Page

9.0 Primitive Equation Spectral Model of the Atmospheric General Circulation
William L. Grose, W. Thomas Blackshear, and
Richard E. Turner 230

9.1 Summary of a Zonally-Averaged Circulation Model of the Atmosphere
Richard E. Turner 232

10. AEROACOUSTIC METHODS

Page

10.0 Extension of Farassat Theory to Include Helicopter Blade Vortex Interaction Noise
Danny R. Hoad 236

10.1 Nonlinear Sonic-Boom Propagation Including Asymmetric Effects and Shock Coalescence
Christine M. Darden 238

10.2 Vortex Modeling of Turbulent Flows
Jay C. Hardin 240

10.3 The Interaction Between a Sound Pulse and a Jet Shear Layer
I - Far Field
Lucio Maestrello, Alvin Bayliss, and Eli Turkel 242

10.4 The Interaction Between a Sound Pulse and a Jet Shear Layer
II - Inflow Field
Alvin Bayliss and Lucio Maestrello 244

11. GRID GENERATION

Page

11.0 Development of Mesh Generation Techniques in Computational Fluid Dynamics for Applications to Flow Field Analyses of Aeronautical Components and Systems With Complex Geometric Configurations
Peter R. Eiseman 248

11.1 Application of a Numerical Orthogonal Coordinate Generator to Axisymmetric Blunt Bodies
Randolph A. Graves, Jr. 250

11.2 Algebraic Techniques for Grid Generation
Robert E. Smith 252

11.3 Effects of Coordinate Systems on the Numerical Solution of Fluid Flow
Joe F. Thompson and C. Wayne Mastin 254

11.4 Time Dependent Coordinate Systems for the Numerical Solution of Fluid Flow
Liviu Lustman 256

12. ADDITIONAL TOPICS

Page

12.0 Development of a Rarefied Flow-Field Analysis Using the Direct Simulation Monte-Carlo Method
James N. Moss 260

1. Sub/Supersonic Linear Theory Methods

Methods which yield solutions to aerodynamic problems by superposition of fundamental solutions of the Prandtl-Glauert linear equation for the potential.

DEVELOPMENT OF A FREE VORTEX SHEET THEORY FOR ESTIMATION OF AERODYNAMIC LOADS ON WINGS WITH SEPARATION INDUCED VORTEX FLOWS

James M. Luckring*

Separation induced vortex-flows from the leading and side edges play an important role in the high angle-of-attack aerodynamic characteristics of a wide range of modern aircraft. In the analysis and design of high-speed aircraft, a detailed knowledge of this type of separation is required, particularly with regard to critical wing loads, optimized vortex lift characteristics, and the stability and performance at various off-design conditions. The purpose of this research is to develop an analytical method based on advanced panel technology capable of estimating three-dimensional pressure distributions for wings with separation induced vortex flows. The code development is being performed by the Boeing Company, Seattle, Washington, under contract to NASA Langley. Numerical and experimental evaluation studies are being conducted jointly.

Figure 1 shows a typical panel arrangement and a brief description of the method follows. The wing, free sheet, and near wake are paneled with biquadratic doublet singularities. Thickness effects are modeled with bilinear surface source singularities. The fed sheet is a simplified model of the vortex core and is merely a kinnematic extension of the free sheet. Zero mass flux boundary conditions are imposed on the wing and free sheet; zero pressure jump boundary conditions are imposed on the near wake. Because the vortex sheet must be locally force free, the zero pressure jump boundary condition is additionally imposed. As a consequence, the resulting system of equation is nonlinear (both the strength and shape of the free vortex sheet are unknown) and an iterative solution procedure is necessitated.

A fairly wide range of application of this method has been conducted. In figure 2, comparison of the experimental surface pressure distributions with the free vortex sheet theory as well as attached flow theory is presented for a delta wing at angle of attack and sideslip. The free vortex sheet theory provides a reasonable estimate of the experimental loads. Also apparent in figure 2 is the ability of the theory to predict the appreciable nonconical nature of the flow as well as the inadequacy of attached flow methods.

Current research efforts are focused on resolving convergence sensitivities encountered for additional geometries of interest such as cropped wings, wing-body configurations, and wings with appreciable twist and camber. Additional efforts are directed at improving code efficiency.

---

*STAD, 505-31-43, (804) 827-3711

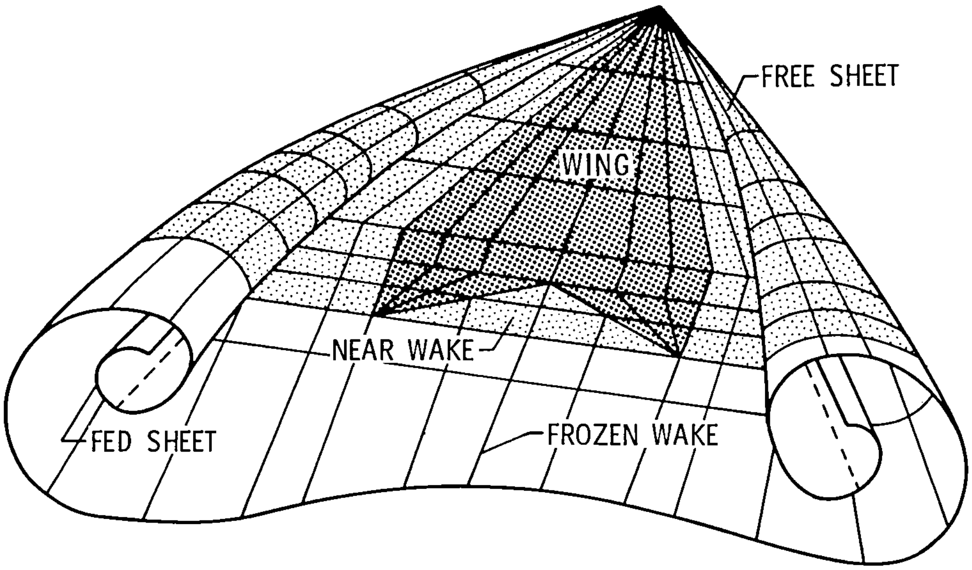

FREE SHEET
WING
NEAR WAKE
FED SHEET
FROZEN WAKE

Figure 1.-Typical panel arrangement.

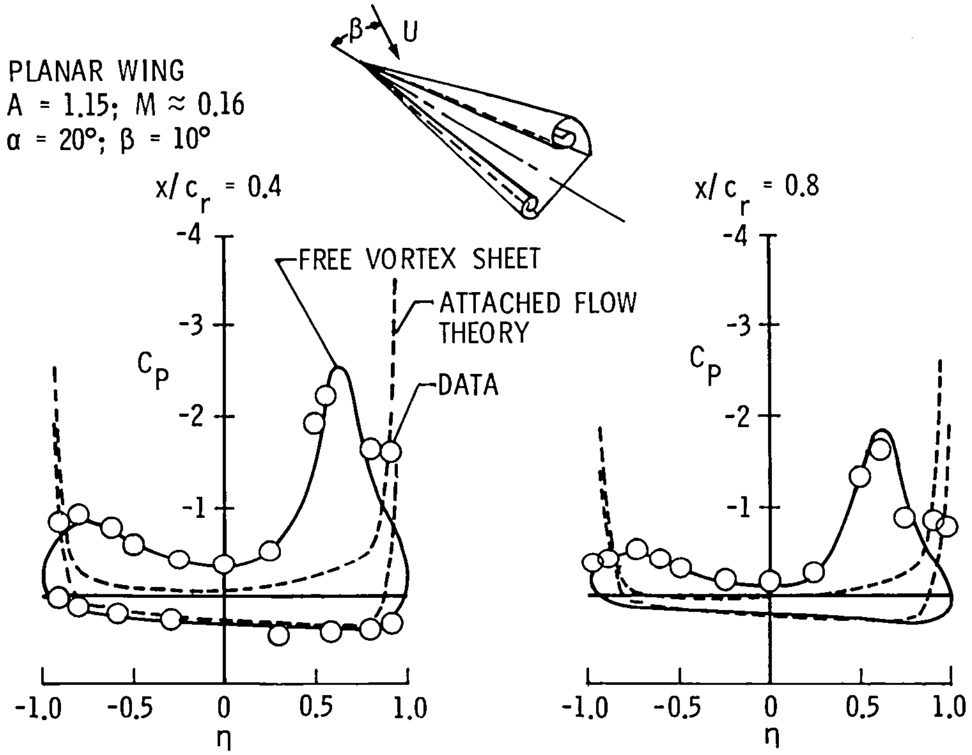


Figure 2.- Spanwise pressure distributions in sideslip.

EXTENSION OF THE FREE VORTEX FILAMENT THEORY TO INCLUDE CORE MODELING

James F. Campbell

Recently, lifting-surface methods have been applied to calculate the aerodynamics of wings having leading-edge vortex-flows. Two popular methods use doublet-panels and free-vortex filaments. The panel method can predict accurate results, but has not been extended to treat complex wing geometries which can have multiple vortex systems. Although the free-vortex filament method is easier to extend to the more complex geometries, its predicted pressure distributions are not as accurate because the free vortices are more diffused than the concentrated vortex found experimentally.

The current effort is being conducted by Dr. E. C. Lan of the University of Kansas to improve the pressure predicting capability of Mehrotra's free vortex filament model (ref. 1) by allowing the filaments to merge into a concentrated vortex core. Mehrotra's model is used to start the solution, then the centroid of the vortex filament system determines the vortex core. The boundary conditions are: (1) no flow through the wing, (2) Kutta conditions at leading and trailing edges, and (3) force-free conditions on the free vortex filaments, concentrated core, and trailing wake elements. Figure 1 shows a sketch of the vortex filament model and figure 2 illustrates the improved pressure predictions after the core is added. The core model also obtains the converged solution in about one half the computer time.

The method is currently being exercised on cambered wings and being extended to the wing-strake arrangement having multiple vortex cores.

Reference:
1. Mehrotra, S. C.; and Lan, C. E.: "A Theoretical Investigation of the Aerodynamics of Low-Aspect-Ratio Wings With Partial Leading-Edge Separation," NASA CR-145304, January 1978.

---

*STAD, 505-31-13, 804-827-3711

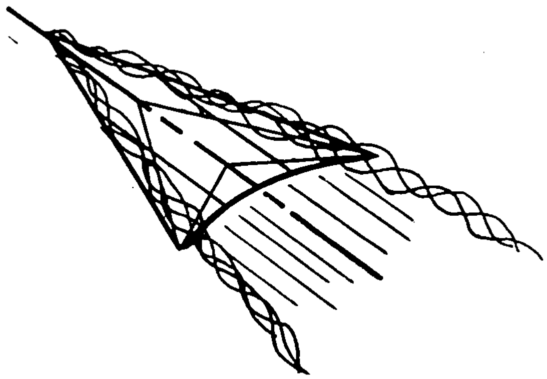


Figure 1.- Sketch of Mehrotra's vortex filament model.

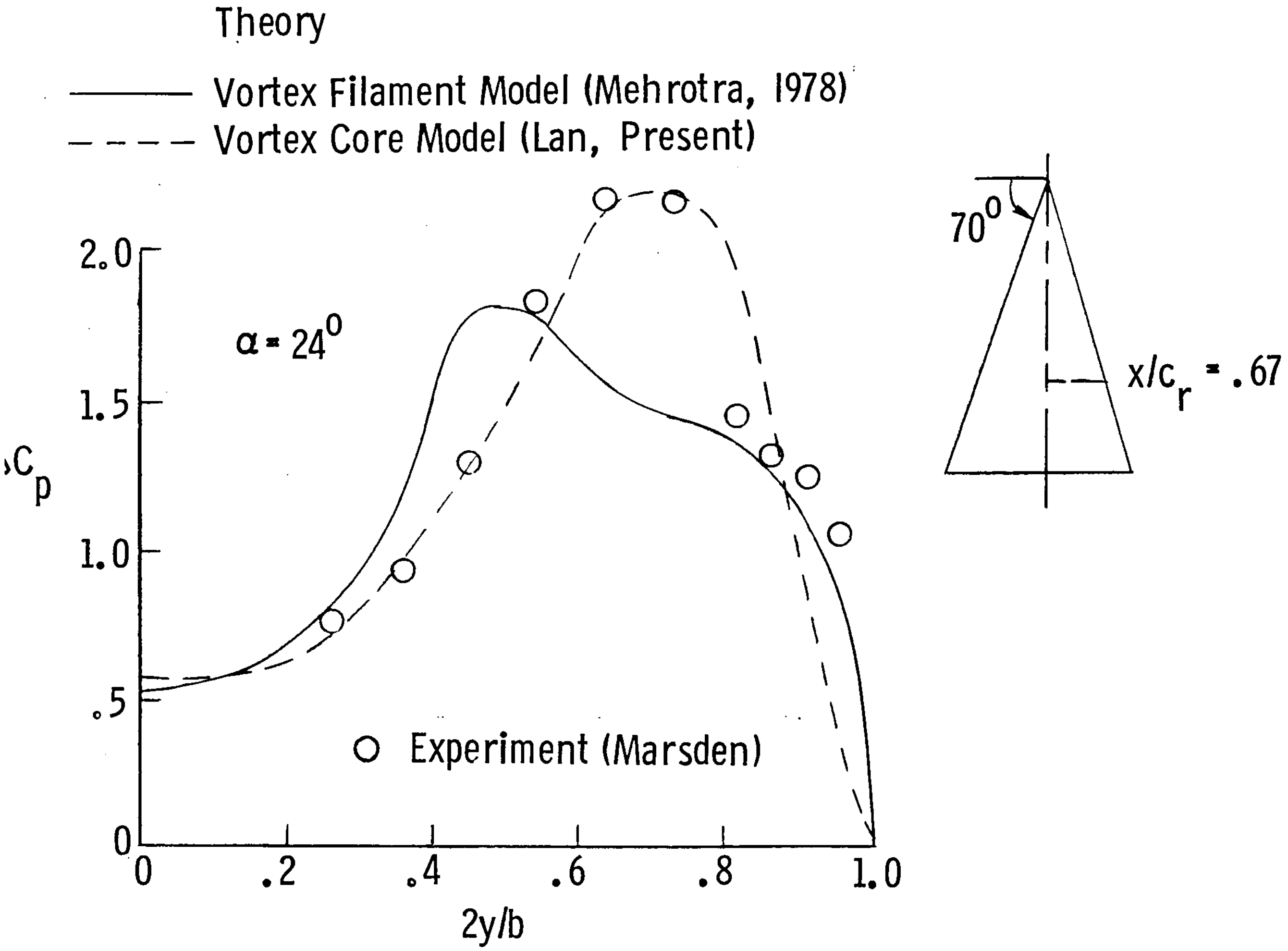


Figure 2.- Effect of leading-edge vortex core model on pressure predictions for a delta wing.

THE DEVELOPMENT OF SUPERIOR SINGULARITIES FOR MODELING POTENTIAL FLOW AERODYNAMIC PROBLEMS AT SUBSONIC AND SUPERSONIC SPEEDS

F.A. Woodward[1]
Charles H. Fox, Jr.[2]
Emma Jean Landrum[3]

The potential flow aerodynamic programs available in the mid-1960's employed relatively simple singularity models. In 1969, analytical studies were initiated to support the development of improved singularity models for use in a unified subsonic-supersonic potential flow aerodynamic program. This computer program, USSAERO, has been used as the test bed for these analytical studies. The USSAERO program has also been widely distributed in the United States as well as in Europe and South Africa. The currently released versions of USSAERO utilize various combinations of constant and linearly varying source and vortex distributions.

After 10 years of working with source and vortex singularities, it was discovered that a very special combination of them resulted in something with highly desirable properties. This significant discovery is considered a new singularity concept and is called a triplet. The triplet possesses numerous analytical properties which make it superior to other singularities. These properties are currently being explored. A computer program utilizing triplet singularities should have significant advantages over existing programs.

References:

1. Contract, NAS1-10408 "Analytical Study of Aerodynamic Characteristics of Wing-Body-Tail Combination at Subsonic and Supersonic Speeds."
2. Contract, NAS1-12900 "Calculation of Aerodynamic Characteristics of Wing-Body-Tail Configurations in Subsonic, Transonic, and Supersonic Flow."
3. Contract NAS1-15792 "Generalized Triplet Singularity, Analytical Methods, Inc."
4. Woodward, F. A.: "An Improved Method for the Aerodynamic Analysis of Wing-Body-Tail Configurations in Subsonic and Supersonic Flow, Part II - Computer Description," NASA CR-2228, Part II, May 1973.
5. Woodward, F. A.: "USSAERO Computer Program Development, Versions B and C," April 1980, NASA CR-3227.
6. Winter, O. A.: "The Incorporation of Plotting Capability into the Unified Subsonic Supersonic Aerodynamic Analysis Program, Version B," NASA CR-3228, 1980.
7. Fox, C. H., JR and Breedlove, W. J., Jr.: "Application of an Improved Unified Subsonic-Supersonic Potential Flow Method for the Aerodynamic Analysis of Aircraft Configurations," AIAA Paper No. 74-186, 1974.
8. Woodward, F. A. and Landrum, E. J.: "The Supersonic Triplet-A New Aerodynamic Panel Singularity with Directional Properties," AIAA Paper No. 79-0273R, Vol. 18, No. 2, Page 138, February 1980.
9. Landrum, E. J. and Miller, D. S.: "Assessment of Analytic Methods for the Prediction of Aerodynamic Characteristics of Arbitrary Bodies at Supersonic Speeds," AIAA Paper No. 80-0071, 1980.

---

[1] Analytical Methods, Inc., Bellevue, Washington, 206-454-6119.

[2] STAD, 505-31-43-03, 804-827-3711

[3] HSAD, 533-01-43-03, 804-827-3181

EXTENSION OF TWO-DIMENSIONAL ANALYSIS AND DESIGN METHOD TO THREE DIMENSIONS

James L. Thomas*

At subsonic speeds, the only general inviscid calculation methods presently available are surface panel methods which distribute source and/or doublet singularities on the surface geometry. An analysis and design formulation, which has proven successful in two dimensions, is being extended to allow three-dimensional design and analysis applications. This work is being performed under contract to NASA Langley by the McDonnell Douglas Corporation.

The two-dimensional analysis method uses linear distributions of source and vortex singularities on planar panels with normal velocity boundary condition satisfaction achieved by an internal perturbation potential boundary condition. The method is numerically stable and the prediction accuracy for surface velocity calculated from singularity strength is competitive with more complex curved panel formulations.

The design method is iterative and uses the derivative matrix of velocity with respect to geometry perturbations in the inversion calculations. Since such an approach accounts for the underlying physics of the flow, the design iteration converges very rapidly. Figure 1 illustrates the rapid convergence of the design cycle for a two-dimensional airfoil with prescribed pressure distributions available from an exact conformal method solution.

Figure 2 shows an application of the analysis and design method to an airfoil with trailing-edge separation. The inviscid model is based on semi-empirical rules for the behavior of pressure distributions in separated flow. The separation point and velocity level at a downstream point are specified and the analysis and design program (MAAD) iterates for the wake shape. The simple model predicts the pressure distribution in good agreement with the experimental data.

The three-dimensional extension will incorporate the two-dimensional design formulation into a three-dimensional method which uses a least square quadratic doublet formulation similar to the two-dimensional linear vortex formulation.

---

*STAD, 505-31-43, 804-827-3611

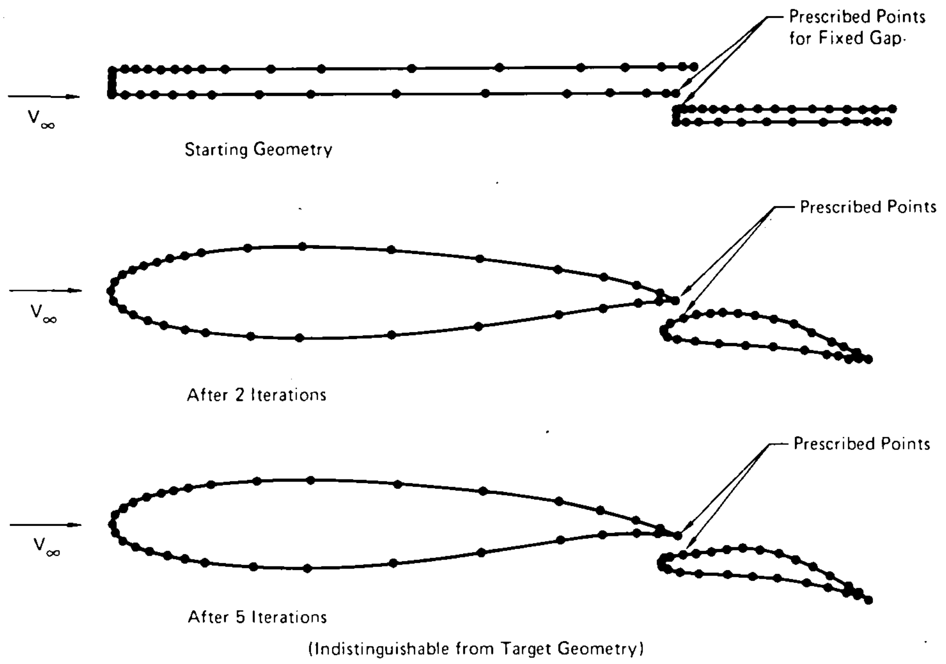

Prescribed Points
for Fixed Gap
V_∞
Starting Geometry
Prescribed Points
V_∞
After 2 Iterations
Prescribed Points
V_∞
After 5 Iterations
(Indistinguishable from Target Geometry)

Figure 1.- Iteration cycle for two-element airfoil inverse design.

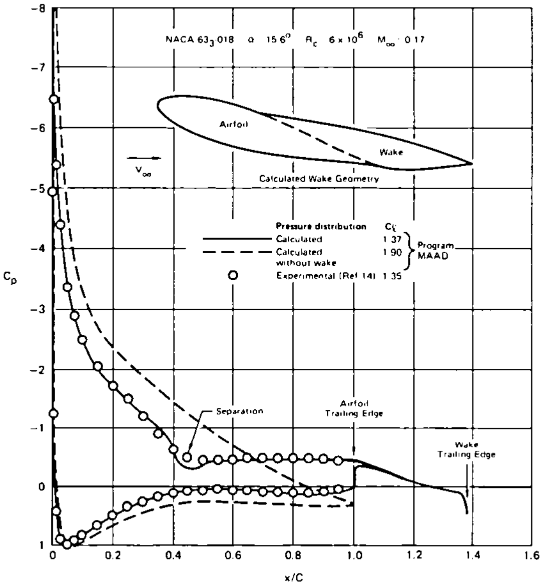


Figure 2.- Application of potential-flow combined analysis and design program to separated airfoil analysis.

DEVELOPMENT OF A VECTOR-CONTINUOUS LOADING CONCEPT FOR AERODYNAMIC PANEL METHODS

James L. Thomas*

For computation of inviscid flow over aerodynamic configurations at subsonic and supersonic speeds, linearized aerodynamic influence methods using surface panel methods have been used for over a decade with reasonable success even for analyses of very complex shapes. Much effort has been expended on the development of such methods to minimize discretization errors and problems with solution stability. Such problems have rendered many panel method formulations unreliable unless used with considerable finesse. A new approach to the reduction of discretization errors in such methods has been developed (ref. 1) and applied to two-dimensional incompressible analyses. The method is being contrasted with higher order curved panel formulations and also being studied for further development to three-dimensional applications.

The approach is based on preventing the occurrence of induced velocity singularities at panel slope discontinuities by maintaining continuity of the velocity jump vector, defined as the vector sum of panel normal and tangential velocity differences arising from the source and vortex singularities, respectively. The sketches in figure 1 illustrate several features of the loading vector concept. Triangular loadings of source and vorticity for a single panel are shown in figure 1(a). Loading vector continuity shown in figure 2(a) indicates that discontinuous source and vortex strengths are required across nonparallel panels. The individual source and vorticity strengths on a panel can be described by a general loading vector at panel intersections shown in figure 1(c). The overlapping of adjacent panels is illustrated in figure 1(d) and shows the continuous variation in both direction and magnitude of the loading vector along a panel.

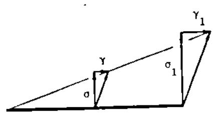
The approach has been evaluated by application to several external and internal flow problems. Figure 2 illustrates the error trends for a circular cylinder analysis using the load vector continuity formulation and a conventional source, vortex continuity formulation. The loading vector formulation exhibits a second order accuracy trend even in the prediction velocity at panel corners which becomes singular in the conventional source, vorticity continuity formulation. In other applications, the method was shown to be generally well-behaved and essentially insensitive to irregularities in panel size distribution.

References:

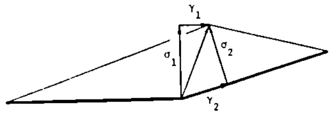
1. Kemp, W. B., Jr.: A Vector-Continuous Loading Concept for Aerodynamic Panel Methods. NASA TM-80104, May 1979.

---

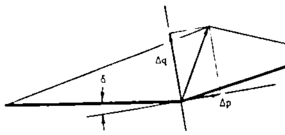
*STAD, 505-31-43, 804-827-3611



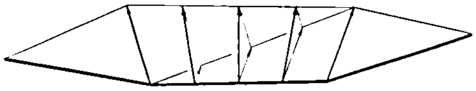
(a) Triangular combined loading



(b) Loading vector continuity

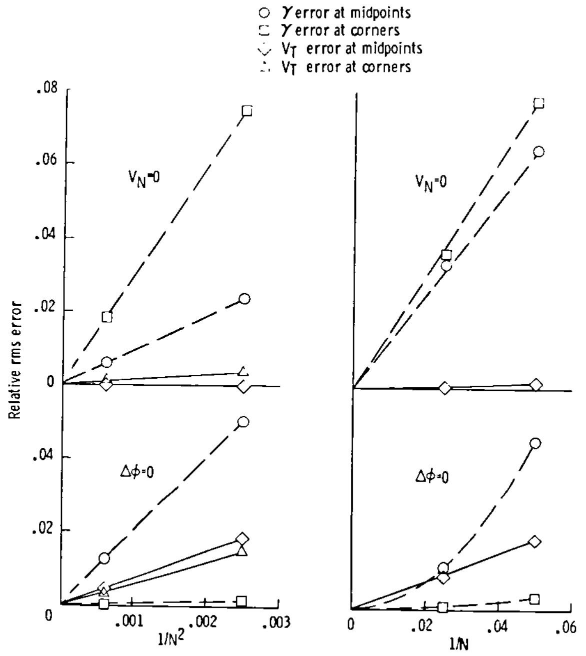


(c) Loading vector quantification



(d) Overlapping loading elements

Figure 1.- Properties of the vector-continuous loading element.



(a) Control points at panel midpoints, vector continuity

(b) Control points at panel midpoints, continuous σ and γ

Figure 2.- Discretization error trends in circular cylinder analysis.

ROTOR BODY INTERFERENCE

Carl E. Freeman*

The fuselage of a single main rotor helicopter operates in a flow field which is composed of its flight velocity, ambient wind conditions, induced flow from the engines, and induced velocities from the rotors. Likewise, the main rotor system is affected by the first three factors and induced velocities created by the fuselage altering the free-stream conditions.

The trend in modern helicopters is to decrease rotor-fuselage separation in order to increase transportability and decrease the overall vehicle size. Accompanying the decrease in rotor-fuselage separation has been a requirement to reduce the fuselage vibration level to provide a more comfortable cabin and cockpit, especially in nap-of-the-earth flight. The conflict is self evident. As separation has decreased, dynamic blade loads and rotor-induced pressure variations on the fuselage have increased. Therefore, the analysis of the diverse flow field at any condition from hover to high-speed flight is important in helping to minimize the dynamic blade and fuselage loadings.

Even though the dynamic nature of the rotor wake is important in analyzing certain problems such as vibration, there are many design details which can be analyzed using a time-averaged downwash field. Therefore, a computer code has been developed which combines a vortex tube rotor-wake theory with an incompressible, potential-flow panel method. This method calculates both on-body and off-body velocities, fuselage surface pressures, and total loads.

Figure 1 presents flow-field velocity vectors as calculated by the combined vortex-tube/panel method for a low-speed flight condition. Figure 2 presents experimental and calculated time-averaged surface pressures on the top center line of the fuselage. At this low-speed flight condition, correlation is good.

---

*STAD, 505-42-13, 804-827-3611

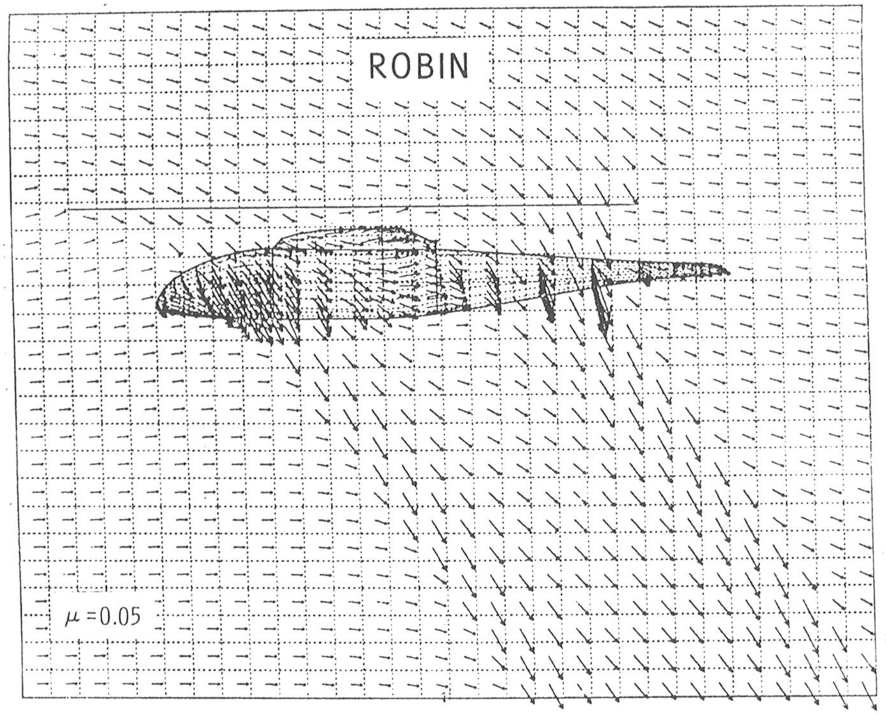


Figure 1.- Flow-field calculations using panel method.

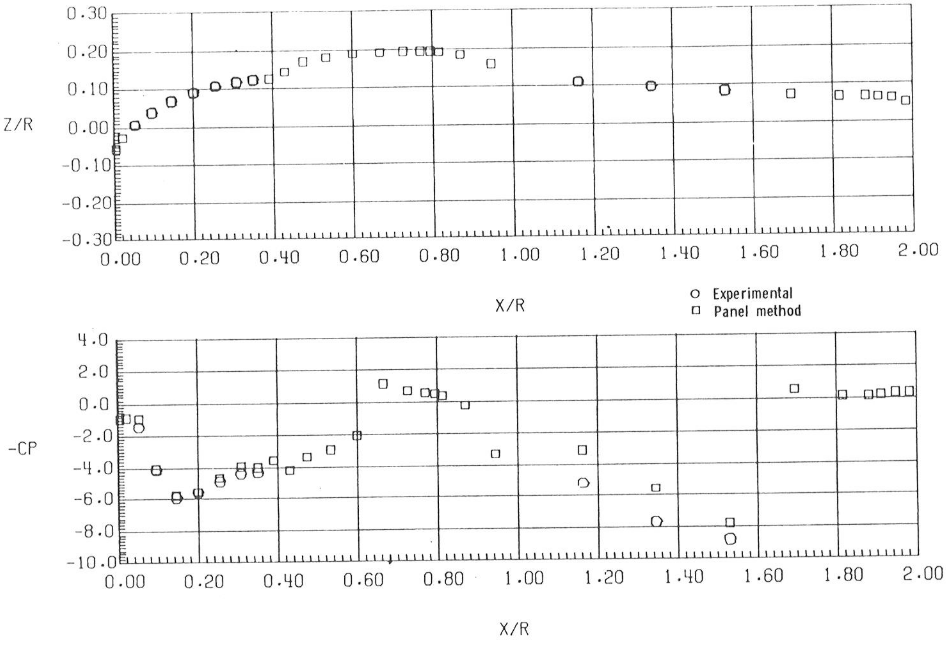


Figure 2.- Calculated and experimental surface pressures on the top center line of the ROBIN.

APPLICATION OF SUBPANELING CONCEPTS TO THE MODELING OF VORTEX/SURFACE INTERACTIONS

W. Elliott Schoonover, Jr.*

Many leading-edge-vortex flow situations are difficult to model with panel methods because they require close proximity of the panels of the vortex sheet and those of the wing surface. Close proximity relative to panel size leads to numerical instabilities and divergence of iterative solutions. Examples of such situations include leading-edge vortices at low angles of attack, vortex separations from rounded surfaces and the secondary-flow regions between a leading-edge vortex and the wing leading-edge. Circumventing this difficulty by simply increasing panel density quickly becomes prohibitively expensive in terms of computational costs. A technique is being developed which will allow localized increasing of panel density in those flow regions where increased density is required. This technique, called subpaneling (ref. 1), is expected to economically solve many current problems in vortex flow modeling.

An example of the use of subpaneling is shown in the attached figure. In the lower left is shown a Clark-Y airfoil at incidence with a vortex fixed above its upper surface by a sink. The resulting pressure distribution is shown in the upper left. The solid line is an exact solution developed by Rossow (ref. 2), using a transformation technique and is shown as a benchmark for evaluating the improvements made by subpaneling. The circular symbols indicate calculated pressures obtained by modeling the airfoil with 40 panels. The right half of the figure shows a detailed scan of the upper surface pressures in the vicinity of the suction peak beneath the vortex. Although the pressures elsewhere around the airfoil were calculated quite accurately, without subpaneling (circular symbols), the suction peak is substantially underestimated. The square symbols indicate a more accurate solution obtained by subpaneling; specifically, each panel within four panel widths of the calculation point is replaced by five subpanels. Each subpanel control point lies on an interpolation of the airfoil contour and each subpanel singularity strength is a biquadratic interpolation of a set of surrounding panel singularity strengths. The improvement in accuracy is dramatic. The key point, however, is that this accuracy improvement was obtained after the forty panel singularity strengths had been determined and, thus, at a cost significantly less than by simply modeling the entire airfoil with 200 panels.

All effort to date has been two-dimensional; the long-range objective, however, is extension of subpaneling techniques to the 3-dimensional paneling used to model leading-edge vortex flows. This research is being conducted by Analytical Methods, Inc., Bellevue, Washington, under NASA contract NAS1-15495.

References:
1. Maskew, Brian: "A Subvortex Technique for the Close-Approach to a Discretized Vortex Sheet." NASA TM X-62487, September 1975.
2. Rossow, V. J.: "Lift Enhancement by an Externally Trapped Vortex, Journal of Aircraft, Vol. 15, No. 9, September 1978, pp. 618-625.

*STAD, 505-31-43, (804) 827-3711

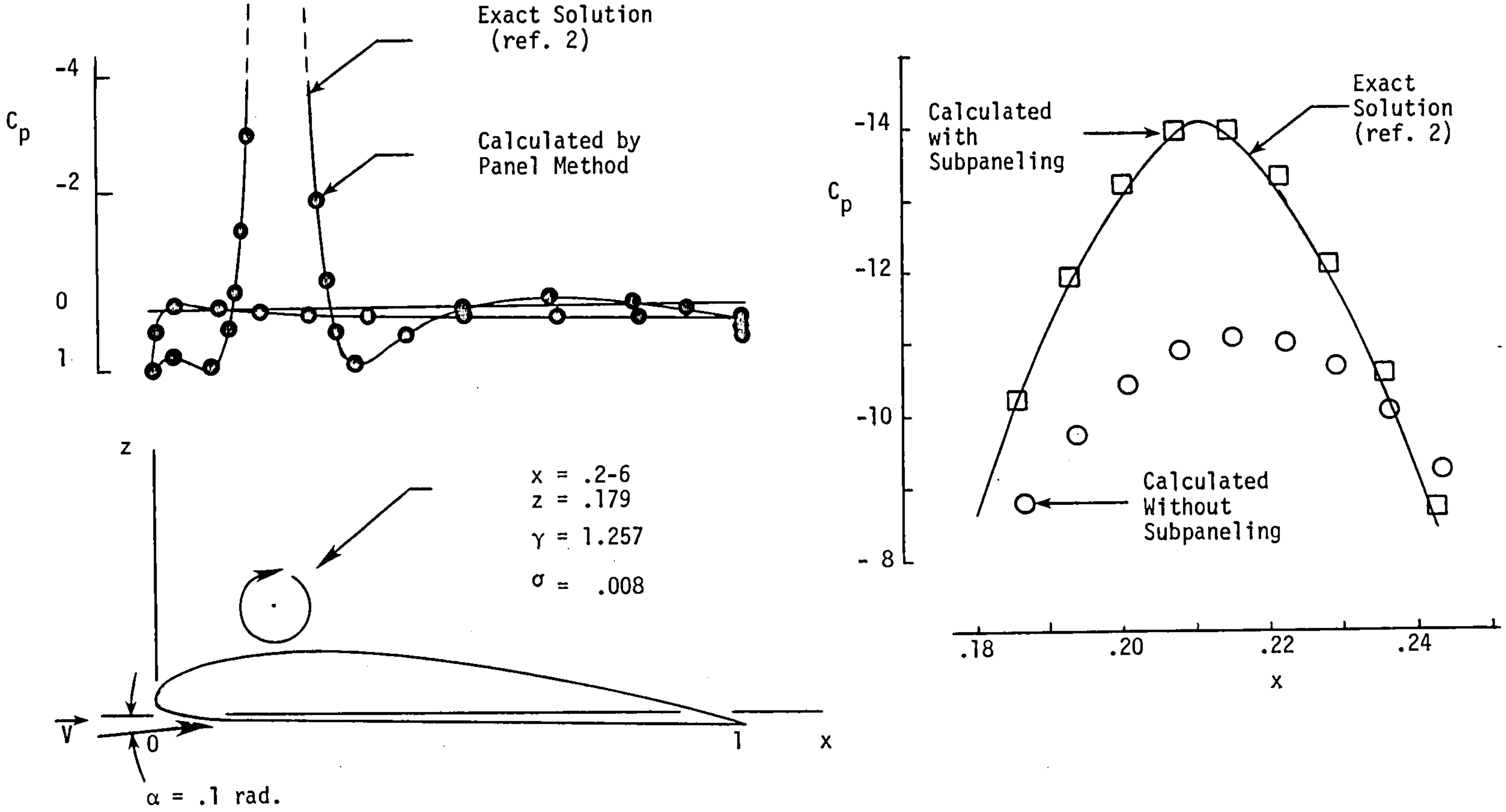


Figure 1.- Improvement of Calculated Pressure Distribution Accuracy Through Subpaneling.

INVISCID ANALYSIS OF TIP VORTEX FORMATION ON OSCILLATING WINGS

Warren H. Young, Jr.*

The lack of analytical design tools for helicopter rotor blade tips has hindered the development of improved tip planforms for decreased control loads, reduced rotor noise, and optimized performance. This program is one of the required tools. The purposes of this analysis are studies of competing tip shapes and to provide guidance for improved tip vortex modeling in future rotor wake programs.

The analysis (ref. 1) is an unsteady potential flow panel method using planar quadrilateral panels to represent the surface of thick wing tips (fig. 1). Each panel has a constant source and doublet distribution. The Dirichlet boundary condition on velocity potential is applied at a central control point. The method includes detailed paneling around the tip edge. As the calculation is stepped in time, a wake separation line may be specified on the tip. Both the usual unsteady wake shed from the trailing edge and the wake shed from the tip are convected downstream at the local velocity. This gives the time history of the rollup of the wake to form the tip vortex. Fourier analysis is used to find the real and imaginary coefficients (fig. 2) after a complete angle of attack oscillation has been calculated.

The analysis is designed to calculate rigid body pitching motion of an unstalled wing in incompressible flow. The resolution of the details of the tip vortex requires several hundred panels and 30 to 50 time steps. Another limitation of the program is that the position of the tip separation line (fig. 1) is an input. Three measures to avoid error in placing the seperation line are planned. First, the calculated pressures will be correlated to unsteady surface pressure measurements in the DFVLR 3 x 3-meter wind tunnel for four tip shapes (fig. 3). Second, an integral boundary layer code will be added to the analysis. Third, a quasi-steady parabolized Navier-Stokes solution will be developed under a separate program. It will calculate the boundary layer, free shear layer, and vorticity diffusion.

The development of this analysis is continuing at Analytical Methods, Inc. under contract NAS1-15472. In addition to the boundary layer, plans include a compressible approximation by restricting the magnitude of the angle of attack variation, motions other than pitching, and wind tunnel wall corrections.

References:

1. Maskew, B.: Influence of Rotor Blade Tip Shape on Tip Vortex Shedding--An Unsteady, Inviscid Analysis. AHS Paper 80-6, May 1980.

---

*SDD and Structures Laboratory, USARTL, 505-42-13-08, 804-827-2661

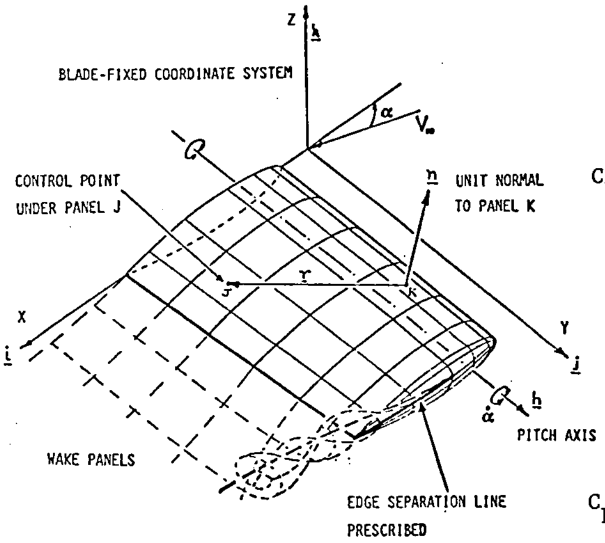

Z
k
BLADE-FIXED COORDINATE SYSTEM
α
V∞
CONTROL POINT UNDER PANEL J
n UNIT NORMAL TO PANEL K
r
J
K
X
i
Y
j
α̇
h
PITCH AXIS
WAKE PANELS
EDGE SEPARATION LINE PRESCRIBED

Figure 1.- Finite elements on a semi-span wing with tip edge separation.

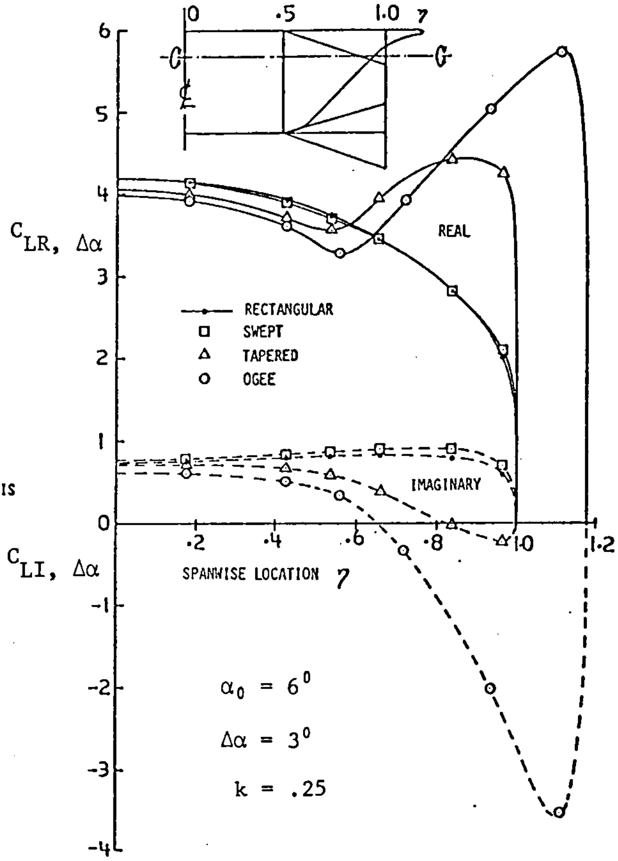


Figure 2.- Section lift curve slopes for four tips based on oscillation amplitude.

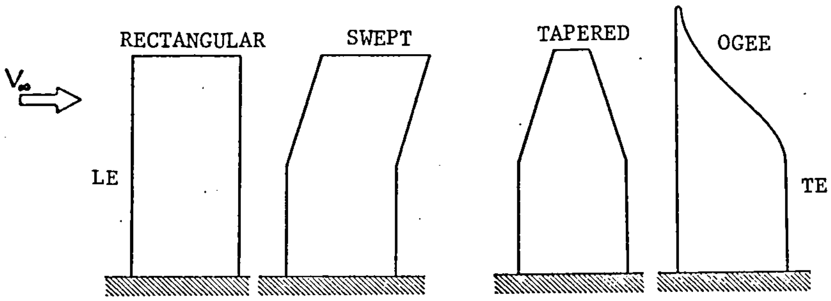

V∞
RECTANGULAR
SWEPT
TAPERED
OGEE
LE
TE

Figure 3.- Four model tip shapes for oscillation about quarter-chord line.

STEADY, OSCILLATORY, AND UNSTEADY SUBSONIC AND SUPERSONIC AERODYNAMICS (SOUSSA)

E. Carson Yates, Jr.*

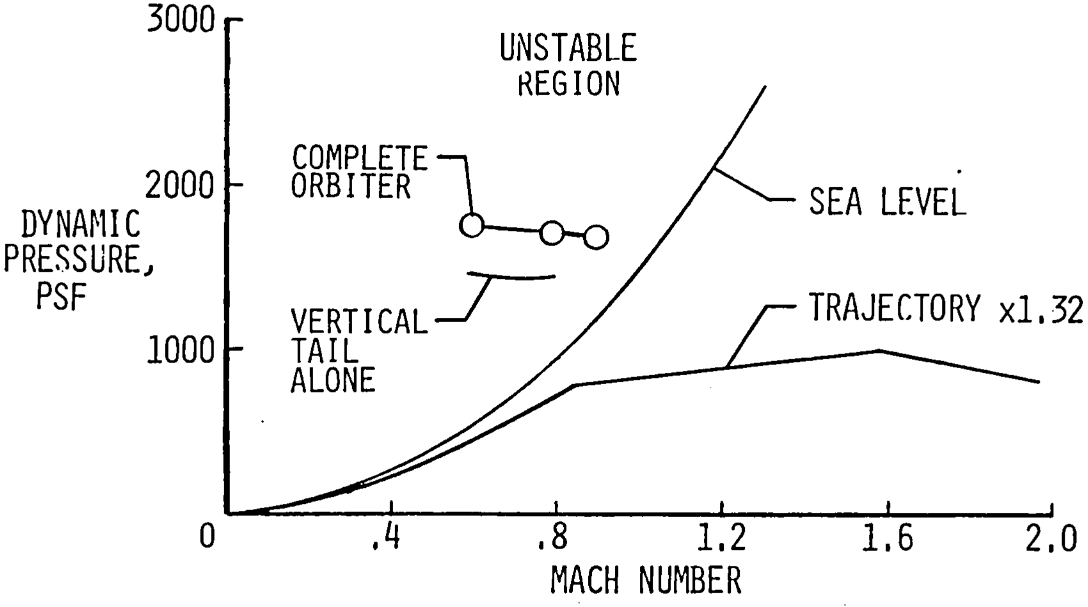
Under NASA grant, Dr. Luigi Morino and his associates at Boston University have developed the theory and methodology for a very general potential-flow analysis that can generate pressure distributions and loads on complete aircraft or other bodies having arbitrary shapes, motions, and deformations. The analysis generated an integral equation for the velocity potential by application of Green's theorem to the governing partial differential equation. This integral equation has been implemented for computation by surface-panel discretization and Laplace transform time solution. The method has been incorporated into a computer program called SOUSSA (Steady, Oscillatory, and Unsteady Subsonic and Supersonic Aerodynamics). The program is flexible, user-oriented, and modular to facilitate inclusion of new capabilities as they become available (e.g., transonic flow which is under current development). The program should be a versatile and effective tool for aerodynamic, aeroelastic, and stability analyses as well as for structural-design optimization.

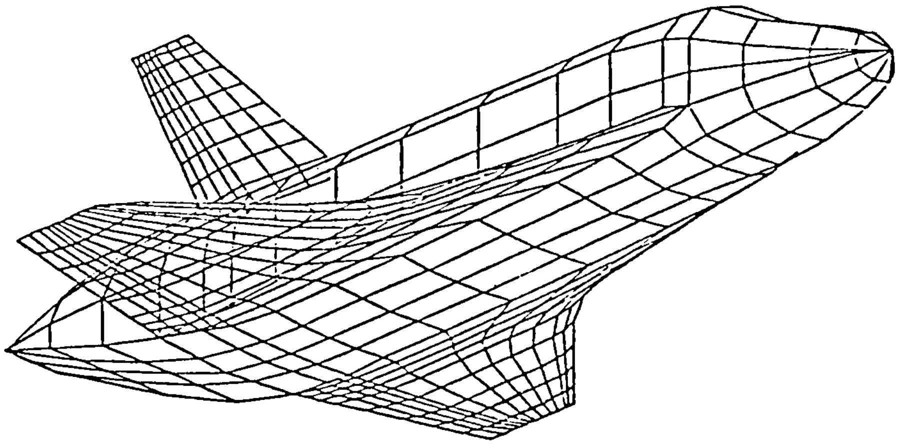
Rockwell International recently used SOUSSA aerodynamics in a 762-panel, 30-mode subsonic flutter analysis of the entire shuttle orbiter. Both symmetric and antisymmetric flutter were investigated with modes of motion that included rudder and elevon rotation as well as rigid-body motions and structural deformations. The entire external surface of the vehicle was paneled, as shown in the attached figure, so that finite thickness of lifting surfaces and aerodynamic interferences were accounted for. The figure alsoshows some representative results for antisymmetric flutter which involved primarily vertical-tail motion. Comparison with the flutter boundary calculated with aerodynamics for the isolated vertical tail shows that the aerodynamic interference of the orbiter on vertical-tail flutter is favorable. The orbiter is indicated to be flutter-free by a substantial margin.

---

*Aeroelasticity Branch, 533-01-13-07, 804-827-2661

SHUTTLE ORBITER FLUTTER ANALYSIS WITH SOUSSA AERODYNAMICS
762 PANELS, 30 VIBRATION MODES





CALCULATION OF LIFTING PRESSURES AND GENERALIZED FORCES ON A THIN LIFTING SURFACE WITH LEADING- AND TRAILING-EDGE FLAP-TYPE CONTROLS UNDERGOING HARMONIC MOTIONS

H. J. Cunningham*

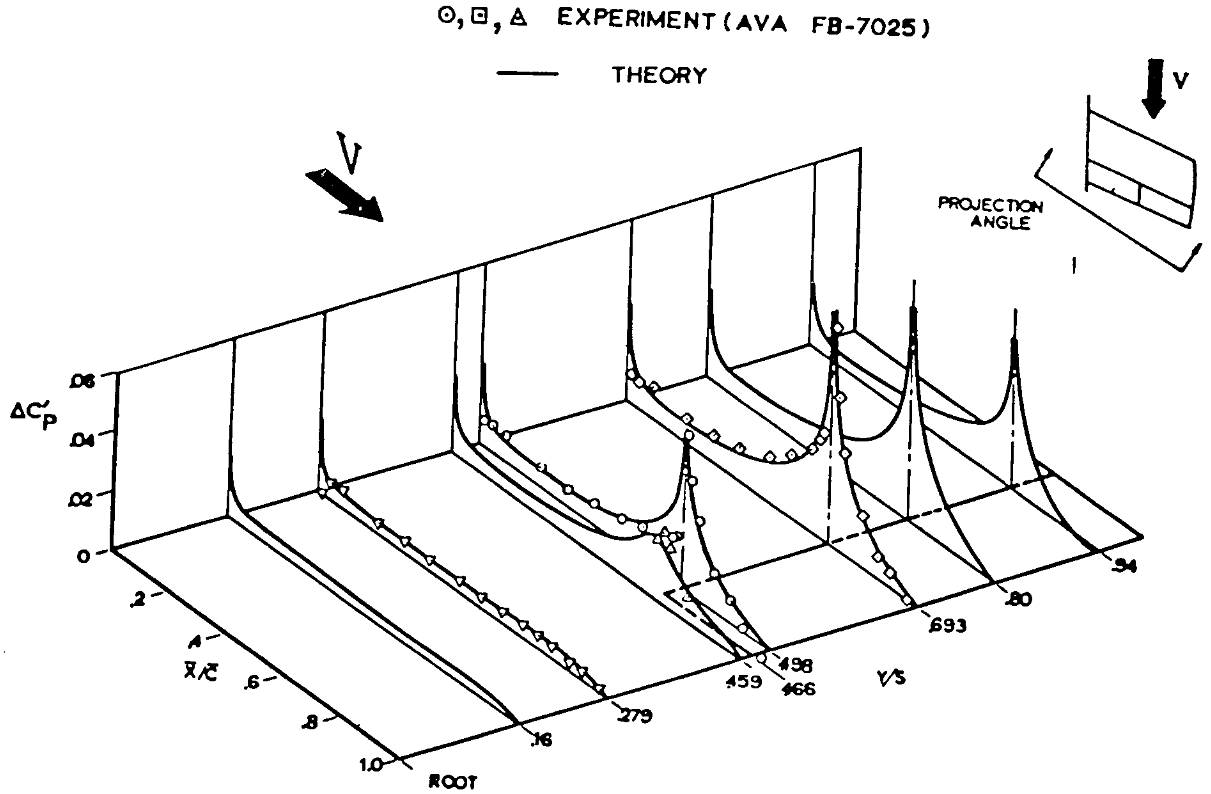
Analysis is made, based on linear potential-flow theory, by a subsonic-kernal-function method with downwash collocation on the surface. Each term in the lifting pressure series is smooth and continuous on the surface except for appropriate singularities at edges and hinge lines.

Leading and trailing edges can each be made up of from one to nine straight-line segments. The edges are continuous, but can have discontinuous slopes, at segment juncture points. Spanwise symmetry or antisymmetry of motion is user specified. From zero to six controls can be accommodated. The user supplies the data to describe the modal deformation and motion. Lifting pressure differences, section generalized forces, and total generalized forces are calculated at user-specified points and span stations. The calculated results are written on formatted files that can be saved and subsequently input to the user's post-processor program, such as, for flutter analysis or plotting.

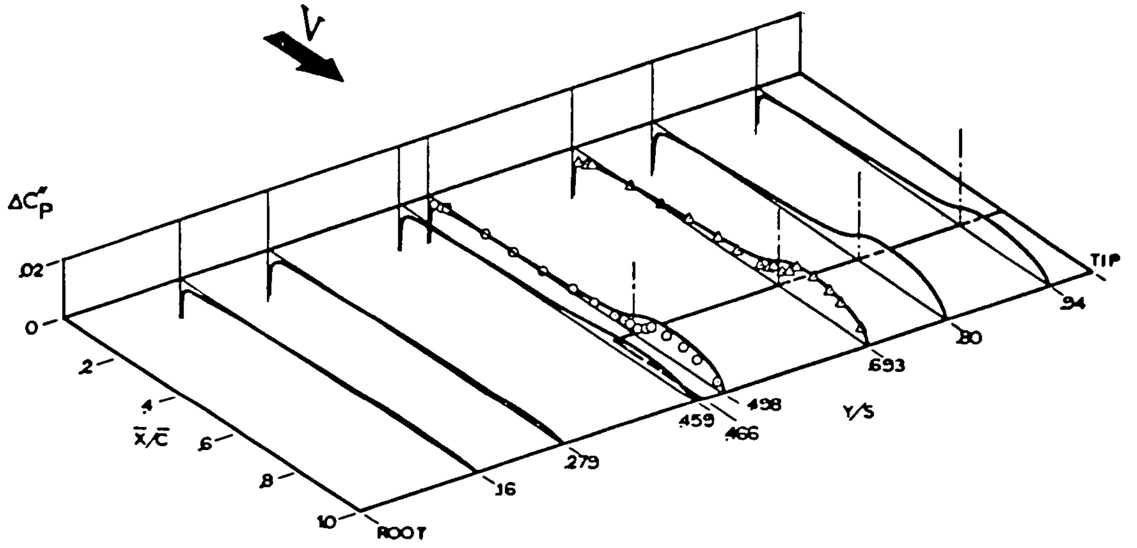
The figure show the theoretical and experimental in-phase and out-of-phase lifting-pressure distributions over a wing with two trailing-edge controls. THe flow Mach number is $M = 0$, and the outer control is oscillating with a reduced frequency $k = 0.372$.

---

*Aeroelasticity Branch, 505-02-23-01, 804-827-2661



In-Phase Pressure Distributions Resulting from Motions of Outer Flap. $A_{i.f.}$ = 0°, $A_{o.f.}$ = 0.66°, A_W = 0°, k = 0.372, and $M \simeq 0$



Out-of-Phase Chordwise Pressure Distributions Resulting from Motions of Outer Flap. $A_{i.f.}$ = 0°, $A_{o.f.}$ = 0.66°, A_W = 0°, k = 0.372, and $M \simeq 0$

HYBRID VORTEX METHOD FOR LIFTING SURFACES WITH FREE-VORTEX FLOW

Osama A. Kandil and Li-Chuan Chu*
E. Carson Yates, Jr.**

A Nonlinear Hybrid Vortex method (NHV-method) has been developed for predicting the aerodynamic characteristics of wings exhibiting leading- and side-edge separations. This method alleviates the drawbacks of the Nonlinear Discrete Vortex method (NDV-method, also known as the multiple line vortex method.) The NHV-method combines continuous-vorticity and vortex-line representations of the wing and its separated free shear layers. Continuous vorticity is used in the near-field calculations, while discrete vortex-lines are used in the far-field calculations. The wing and its free shear layers are divided into quadrilateral vortex panels having second-order vorticity distributions. The aerodynamic boundary conditions and continuity of the vorticity distributions are satisfied at certain nodal points on the vortex panels. An iterative technique is used to satisfy these conditions in order to obtain the vorticity distribution and the wake shape. Distributed and total aerodynamic loads are then calculated. The method will be extended to unsteady flow for use in flutter analyses of wings and aircraft at high angle of attack.

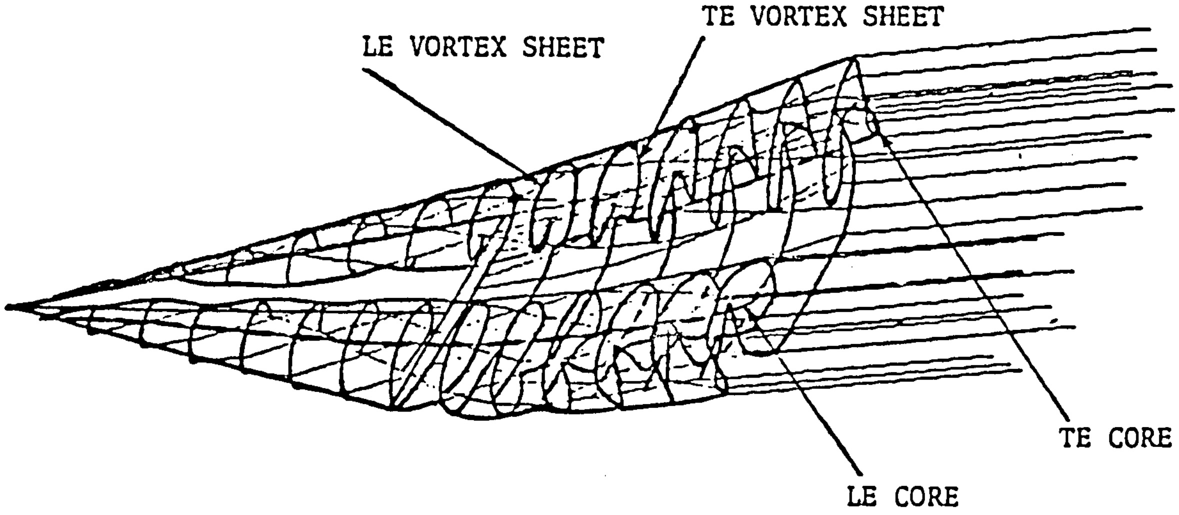
The attached figure illustrate the kind of vortex system that is modeled by the hybrid method for a simple delta wing. The lower left shows the shape of the right-hand side of the leading-edge vortex system at the longitudinal position of the trailing edge. The lower right shows flow at a cross section 8 percent of root chord downstream from the trailing edge. At this location the portion of the vortex sheet shed from the trailing edge has started to deform (roll up) in a clockwise sense which is opposite to that for a trailing-edge vortex sheet generated in the absence of leading-edge flow separation. The arrows represent velocities measured by Hummel, and the dash-dot line indicates his estimate of the vortex-sheet location. Agreement with the results calculated for the low-order (coarse) paneling arrangement used for this illustration is good.

---

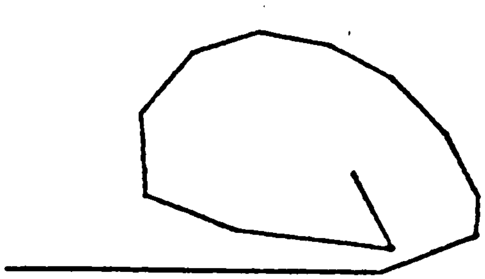
*Old Dominion University

**Aeroelasticity Branch, 505-33-53, 804-827-2661

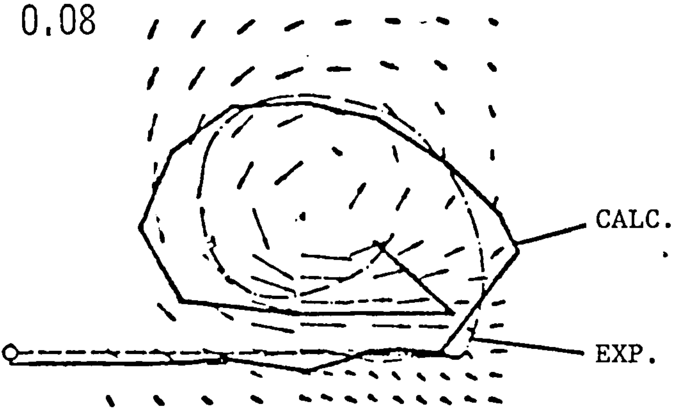
VORTEX FLOW AROUND AR = 1. DELTA WING


TE VORTEX SHEET
LE VORTEX SHEET
TE CORE
LE CORE

ξ = 0. (TE)



ξ = 0.08



A NONLINEAR CONTINUOUS-VORTICITY MODEL

Ali H. Nayfeh* and Dean T. Mook*

Because the lifting surfaces of many modern vehicles are designed for supersonic flight, they generally have small aspect ratios, are highly swept, and have sharp leading edges and/or tips. Consequently, one must expect aerodynamically influential vortex systems to be generated by leading-edge and tip separation (these are called the leading-edge and tip wakes) even at moderate (4^o) angles of attack or deflection.

For a single wing without interactions, the need to model the leading-edge and tip wakes accurately is clearly illustrated by comparisons of the aerodynamic loads predicted by various lifting-surface methods with experimental data. For multi-component interactions, the need to model all the wakes accurately is compounded by the possibility of one component being in the wake of another. It is not reasonable to suppose that the strength, direction and position of the wake can be assigned accurately by employing some general assumptions; rather, one should expect the solution to provide these.

In earlier papers, we developed an accurate, discrete-vortex model of the wakes and coupled this with a vortex-lattice model of the lifting surface. This compound model, which provides the strength, direction and position of the vorticity in the wake, can be used to predict accurately the aerodynamic loads on far-coupled and close-coupled wings, wing-body combinations and wings in unsteady flows.

Though this discrete-vortex technique yields the total loads accurately, it does not always lead to accurate predictions of the pressure. Thus, a continuous-vorticity technique is being developed to treat lifting and non-lifting subsonic flows. Such an approach will result in significant improvements in the accuracy of the predicted pressure. Then this technique will be extended to treat general unsteady lifting flows and finally to modify the general unsteady technique to treat small, harmonic oscillations efficiently.

---

*Engineering Science and Mechanics Department
Virginia Polytechnic Institute and State University

2. Boundary Layers

Methods for solving the 2-D or 3-D parabolic-type boundary-layer equations. (Application of established boundary-layer codes to a problem would usually be omitted from this compendium.)

ADVANCED 2-D TURBULENT BOUNDARY-LAYER SEPARATION PREDICTION METHOD

Suresh H. Goradia*

Harry L. Morgan Jr.**

For the past several years, the Lockheed-Georgia Company has been developing methods for the prediction of separation and stall on single-component airfoils. For leading-edge stall, Goradia's weighted multiparameter method for the prediction of laminar separation and reattachment has been shown to give accurate stall angle-of-attack predictions for a wide variety of airfoils. Application of these leading-edge stall airfoils to practical aircraft wings have generally produced undesirable abrupt stall characteristics. The trailing-edge stall airfoils which are characterized by large regions of separated flow near the trailing-edge have very nonlinear lift versus angle-of-attack curves and produce more desirable gradual wing stall characteristics. Several theoretical methods are currently being developed to model this trailing-edge separation region by determining the effective displacement of the fluid streamlines in inviscid flow. These methods require an accurate prediction of point of turbulent boundary layer separation.

A recent contractual effort was initiated with Lockheed-Georgia to develop a new method based on energy dissipation principals and the use of inviscid pressure distributions to predict turbulent boundary layer growth, the location of separation, and the viscous pressure at the point of separation. Using moment of momentum and energy dissipation integral equations, transformations were performed so that terms containing gradients of pressure at the outer edge of the boundary layer did not appear in the governing equations. This resulted in a set of integral equations whose solution is not jeopardized by large pressure gradient errors that are induced by relatively small inaccuracies in the computed pressure distribution. These integral equations also include generalized terms which account for the effects of large velocity fluctuations near the point of separation and were formulated based on a set of laser velocimeter data obtained by Lockheed on a Wortmann-type airfoil. Parametric criteria for separation were then derived from dimensional analysis of the resulting integral equations. Several parameters were found effective in predicting the separation point and a large number of correlations were made to determine the most effective parameter. Additional parametric criteria were also found to accurately predict the viscous pressure at separation. The results of this method applied to a recently developed 17-percent thick, medium-speed general aviation airfoil are presented in figure 1 and show excellent agreement for both the location of separation and the viscous pressure at separation.

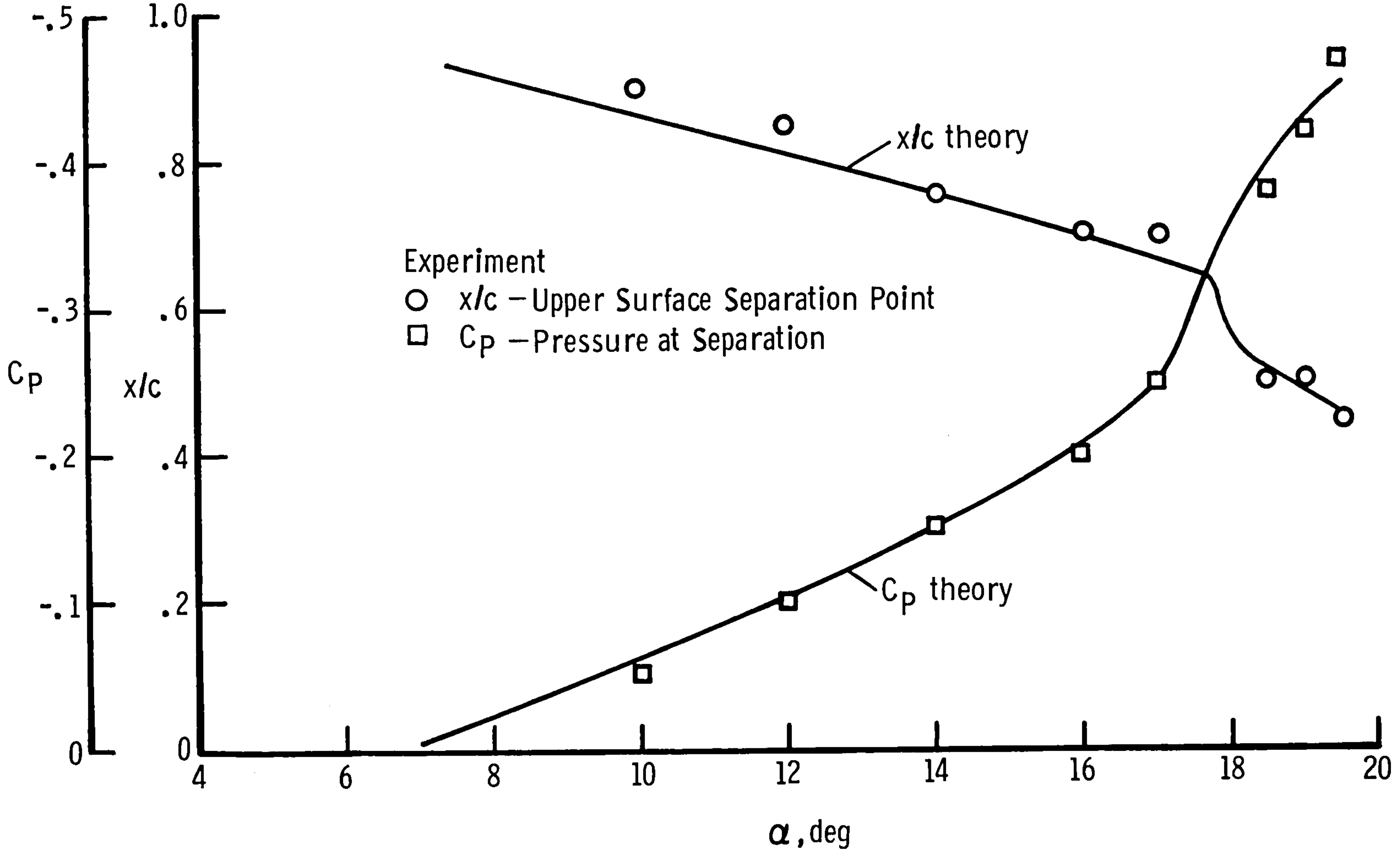
---

*Lockheed-Georgia employee

**NASA Langley employee, STAD, 505-11-13, 804-827-4514

SEPARATION PREDICTION FOR MS(1)-0317 AIRFOIL USING TURBULENT SEPARATION METHOD

($M_\infty = 0.15$)



CALCULATION OF HEATING RATES ON THREE-DIMENSIONAL CONFIGURATIONS

H. Harris Hamilton II*

A method has been developed for calculating laminar heating rates on three-dimensional configurations using the "axisymmetric analogue" (ref. 1). In this approach, the three-dimensional boundary-layer equations are written in inviscid surface-streamline coordinates and the crossflow in the boundary layer is assumed small, which is the case when the streamline curvature is small or when the wall is highly cooled. This reduces the three-dimensional boundary-layer equations to a form equivalent to the usual axisymmetric equations. Heating rates on a configuration are calculated along streamlines using any existing method of solving the "equivalent" axisymmetric boundary layer. When several streamlines are considered, the heating over the entire configuration can be obtained.

The inviscid surface streamlines, metric coefficients, and boundary-layer edge conditions are calculated from a three-dimensional inviscid flow-field solution obtained from a computer code (STEIN) developed by the Grumman Aerospace Corporation. Coupling the present boundary-layer method with the three-dimensional inviscid code means that heating rates can be calculated on any configuration where the flow is attached and an inviscid solution can be obtained.

The heating rates along the windward symmetry plane of an 80° sweep slab delta wing at an angle of attack of 20° are presented in figure 1. There is good agreement between the present theory and the experimental data. For the same configuration, the circumferential distribution of heating rates at three axial stations are presented in figure 2. Again, the agreement between the present theory and the experimental data is very good. Similar results have been obtained for other configurations such as blunt cones and the fuselage of the Space Shuttle Orbiter (ref. 2).

References:

1. Cooke, J. C., "An Axially Symmetric Analogue for General Three-Dimensional Boundary Layers," A.R.C. Technical Report, R&M No. 3200, June 1959.
2. Hamilton, H. Harris II, "Calculation of Heating Rates on Three-Dimensional Configurations," Report for Degree of Engineer, George Washington University, December 1979.

---

*SSD, Langley, 804-827-2921

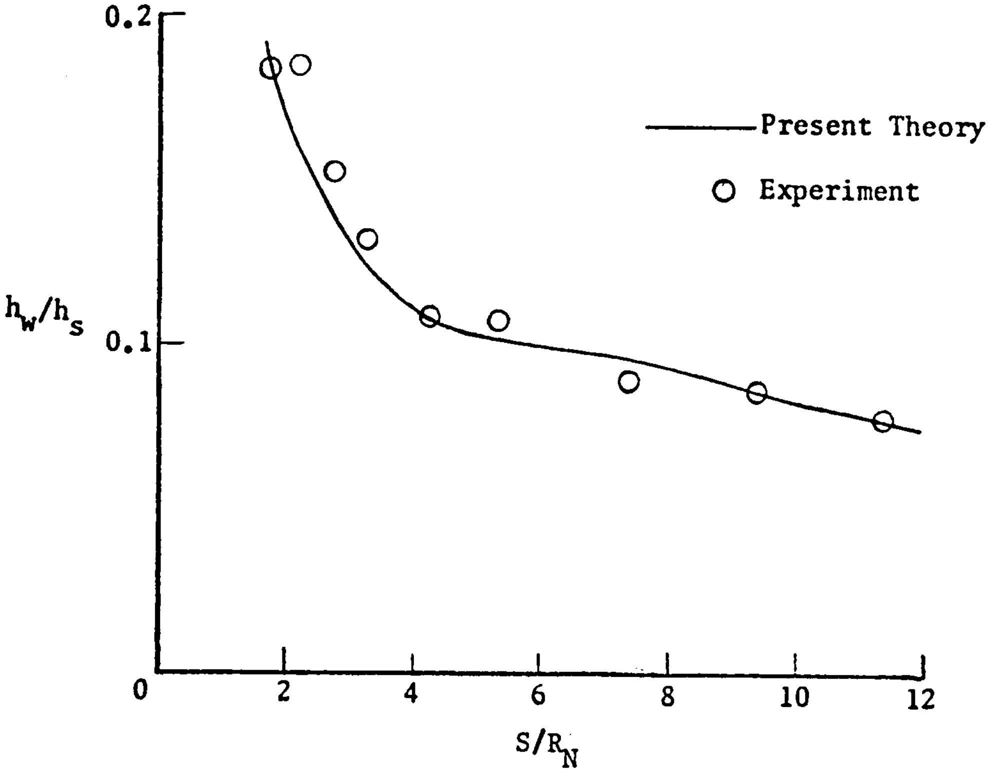
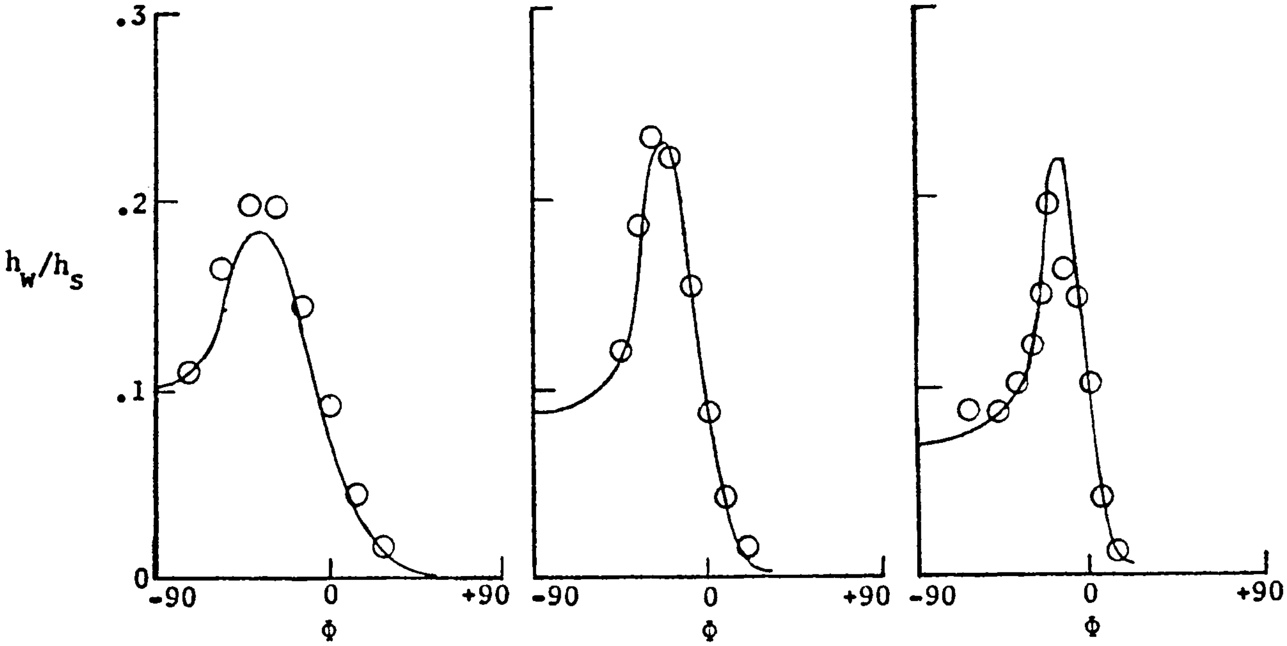


Fig. 1 Distribution of heating in windward symmetry plane of 80° sweep slab delta wing; $\alpha = 20°$; $M_\infty = 9.6$; $Re_\infty = 3.94 \times 10^6$/m.



(a) $\ell/R_N = 4.$ (b) $\ell/R_N = 8.$ (c) $\ell/R_N = 12.$

Fig. 2 Circumferential distribution of heating on 80° sweep slab delta wing; $\alpha = 20°$; $M_\infty = 9.6$; $Re_\infty = 3.94 \times 10^6$/m.

3. Viscous/Inviscid Interaction

Methods for interacting inviscid codes with viscous (usually boundary-layer) codes.

CALCULATION OF AEROELASTIC LOADS ON WINGS IN TRANSONIC FLOW

Woodrow Whitlow, Jr.*

Robert M. Bennett*

The aerodynamic loads acting on flexible wings depend on the downwash which, in turn, depends upon the elastic deformation of the wing. Thus, the elastic deformation, which itself is determined by the aerodynamic loads, can have a significant effect on wing loading. Specifically, transonic prediction methods that assume rigid wings--which almost all such methods do--may predict loads on flexible wings that differ drastically from the true values.

Consequently, this effort, which is to be conducted at Langley Research Center, is aimed at coupling an aeroelastic model with a steady transonic prediction method to yield the flexible wing loading. The wing which is currently being examined is a transport wing with a supercritical section. That wing is shown in Figure 1, which is taken from Reference 1. The transonic flow field for a rigid wing is computed using the FL022 computer program[2], which is a finite difference solution of the full potential equation. The wing loading obtained from that program may then be used to determine the aeroelastic correction to the current wing geometry. That geometry is then used as the new rigid wing and the process continued until a converged geometry and flexible wing loading is obtained. The effects of viscosity may also be included by incorporating the solution of the boundary layer equations into the iteration process.

The type of effects that are expected are shown in Figures 2 and 3 which are taken from Reference 1. Figure 2 shows the spanwise lift and pitching moment distributions on a wing operating at the cruise point, and Figure 3 shows those distributions for the same wing operating at a much higher dynamic pressure. These figures demonstrate the need to incorporate aeroelastic effects into the aerodynamic predictions for flexible wings.

References

1. Chipman, R.; Waters, C.; MacKenzie, D.: Numerical Computation of Aeroelastically Corrected Transonic Loads. AIAA Paper No. 79-0766.

2. Jameson, A.; and Caughey, D. A.: Numerical Calculation of the Transonic Flow Past a Swept Wing. ERDA Research and Development Report COO-3077-140, 1977.

---

*SDD, 505-02-23, 804-827-2661

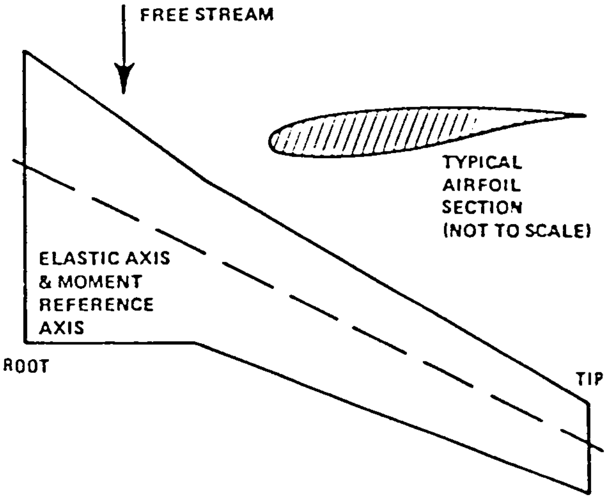

FREE STREAM
TYPICAL AIRFOIL SECTION (NOT TO SCALE)
ELASTIC AXIS & MOMENT REFERENCE AXIS
ROOT
TIP

Figure 1. Planform of a transport wing with supercritical sections.

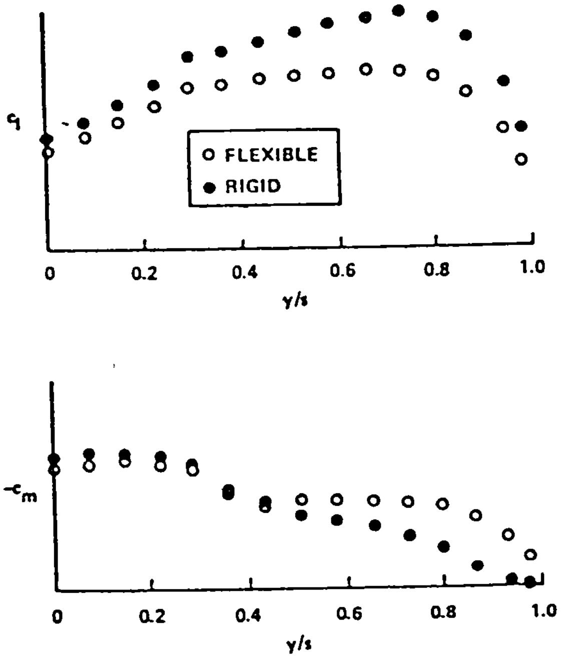


Figure 2. Predicted spanwise distribution of lift and pitching moments at the transonic cruise point.

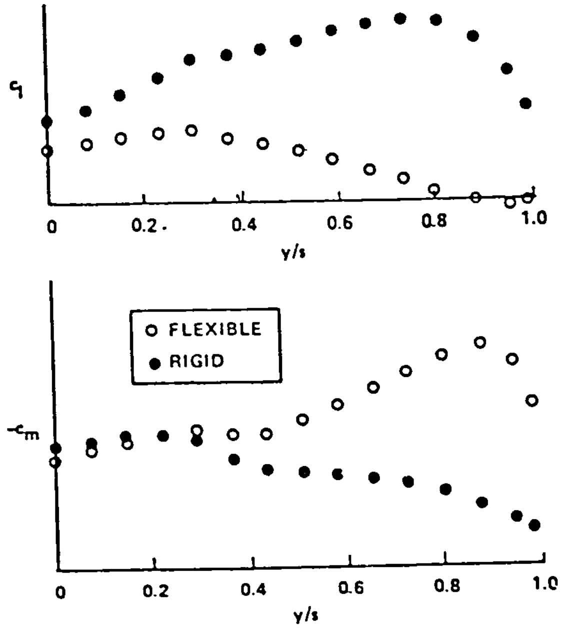


Figure 3. Predicted spanwise distribution of lift and pitching moments at a transonic low altitude point.

INTERACTION OF 3-D WING BOUNDARY-LAYER CODE WITH 3-D TRANSONIC INVISCID WING CODES

Craig L. Streett*

Transonic flow over three-dimensional wings, especially over those configurations using supercritical airfoil sections, is known to be strongly affected by viscous effects even when separated regions are very small. Both level and spanwise distribution of wing lift is changed by the decambering effect of the boundary layer. Although viscous correction using two-dimensional strip boundary-layer calculations shows fair agreement with experiment in some cases, the strong spanwise crossflow in the cove region of a supercritical wing dictates that any viscous correction used must be three-dimensional to have hope of correcting properly for these viscous effects.

Iteratively interacting inviscid and boundary-layer computations to produce an effective body-displacement surface and its corresponding pressure distribution is a technique which in principle should produce results of adequate accuracy. However, current three-dimensional finite-difference boundary-layer techniques require computation times and storage of at least the same order as presently used full-potential inviscid calculation techniques. Thus, the cost of the many complete boundary-layer calculations required for interaction is prohibitive. Present three-dimensional integral boundary-layer techniques, on the other hand, can reduce this cost by a factor of 20 or more and give results in good agreement with finite-difference codes even in difficult flow situations. In addition, integral methods are in general more robust than finite-difference methods, an important consideration during early iterations of viscous/inviscid interaction.

At present, a modern three-dimensional integral boundary-layer code has been successfully interacted with the transonic full-potential wing code FLO-27. Converged interacted results for three wing stations on the supercritical SCW-3 wing are compared with experiment in figure 1. Agreement is good, with much of the decambering effect of the boundary layer in the cove region predicted. Body effects were not accounted for in this calculation; work to interact the wing-body code FLO-30 with the present boundary-layer method is underway, and agreement is expected to improve.

---

*STAD, 505-31-13, 804-827-2627

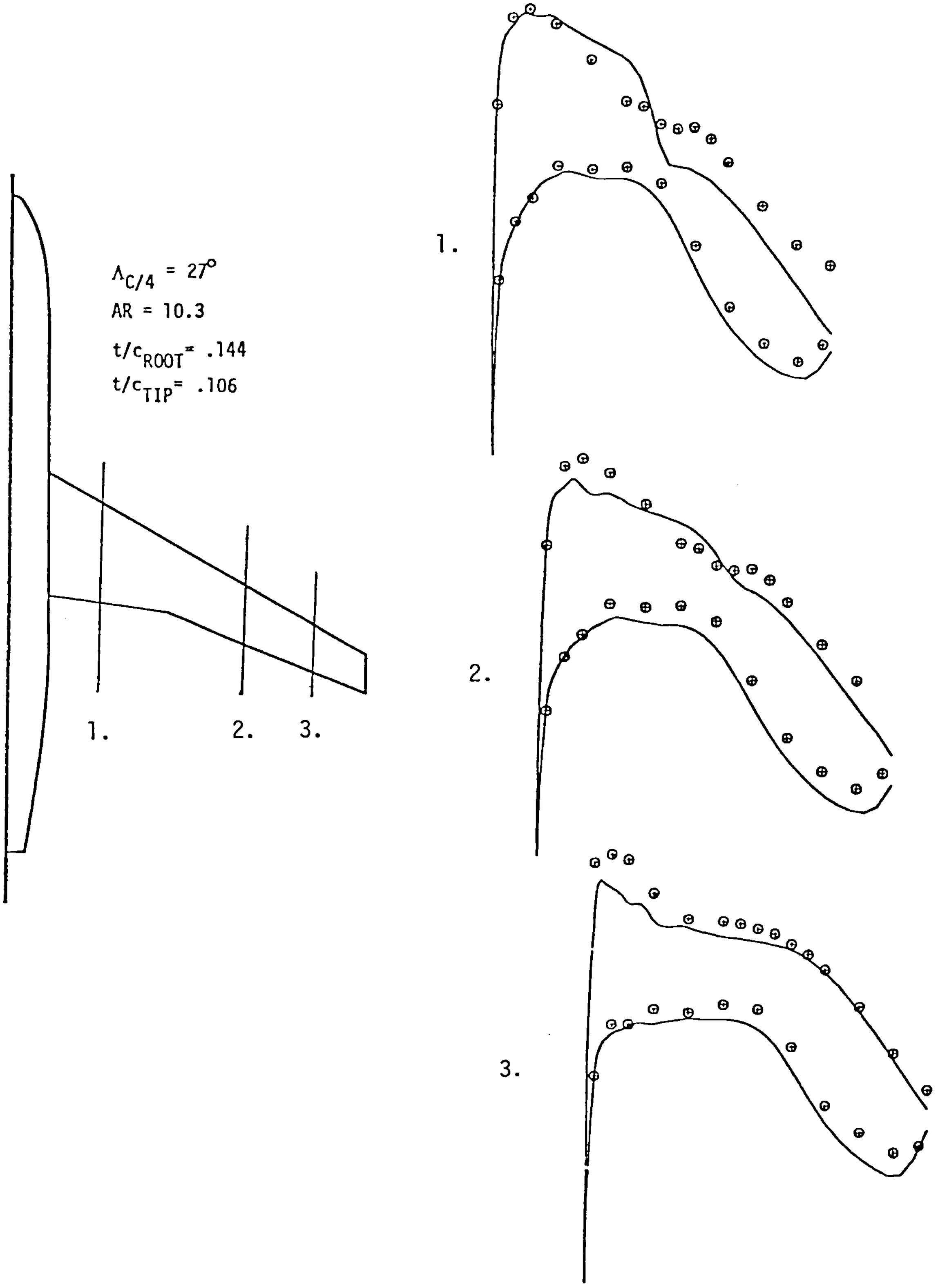


Figure 1.- SCW-3 wing; comparison of calculated and experimental pressure distributions.

CALCULATION OF TRANSONIC FLOW ABOUT AIRFOILS WITH TRAILING-EDGE SEPARATION

Richard W. Barnwell*

During the past decade, computational fluid dynamics methods have been developed to the point that reliable predictions can be made for attached viscous flow past airfoils traveling at subsonic and transonic speeds. However, despite some pioneering efforts in the 1960's, the development of methods for calculating separated flow has lagged considerably. As a result, extensive wind-tunnel experimentation is still required to determine airfoil stall characteristics and control effects although design-point characteristics can be predicted analytically.

Viscous airfoil flowfields at typical flight Reynolds numbers can generally be approximated as inviscid outer flowfields and viscous boundary layers. Therefore, the development of reliable separated-flow prediction methods depends, in principal, on the successful coupling of existing inviscid and boundary-layer techniques, which have reached a high degree of development during the past decade. At present, several grants and contracts are being supported by the Airfoil Aerodynamics RTOP to develop hybrid methods of this type to account for trailing-edge separation. F. A. Dvorak of AMI is incorporating a vortex lattice representation of the separation streamlines into the NASA-Langley/Lockheed multi-element vortex-lattice airfoil program (MCARF) for subsonic flow. This technique, which is similar to the single-element method described in reference 1, uses one of several integral boundary-layer methods. L. A. Carlson of Texas A & M is developing a finite-difference/integral boundary-layer method for both subsonic and transonic applications (ref. 2). O. R. Burggraf of the Ohio State University is developing a method in which finite-diference techniques are used both in the inviscid and boundary-layer regions. Barnwell has recently succeeded in calculating separated-trailing-edge flow about an airfoil traveling at transonic speeds. The methods used employed a simple finite-difference potential flow method and an integral boundary-layer method. Results are shown in Figure 1.

References:
1. Mashew, B; and Dvorak, F.A.: The Prediction of $C_{l,max}$ Using a Separated Flow Model. AHS Journal, April 1978.

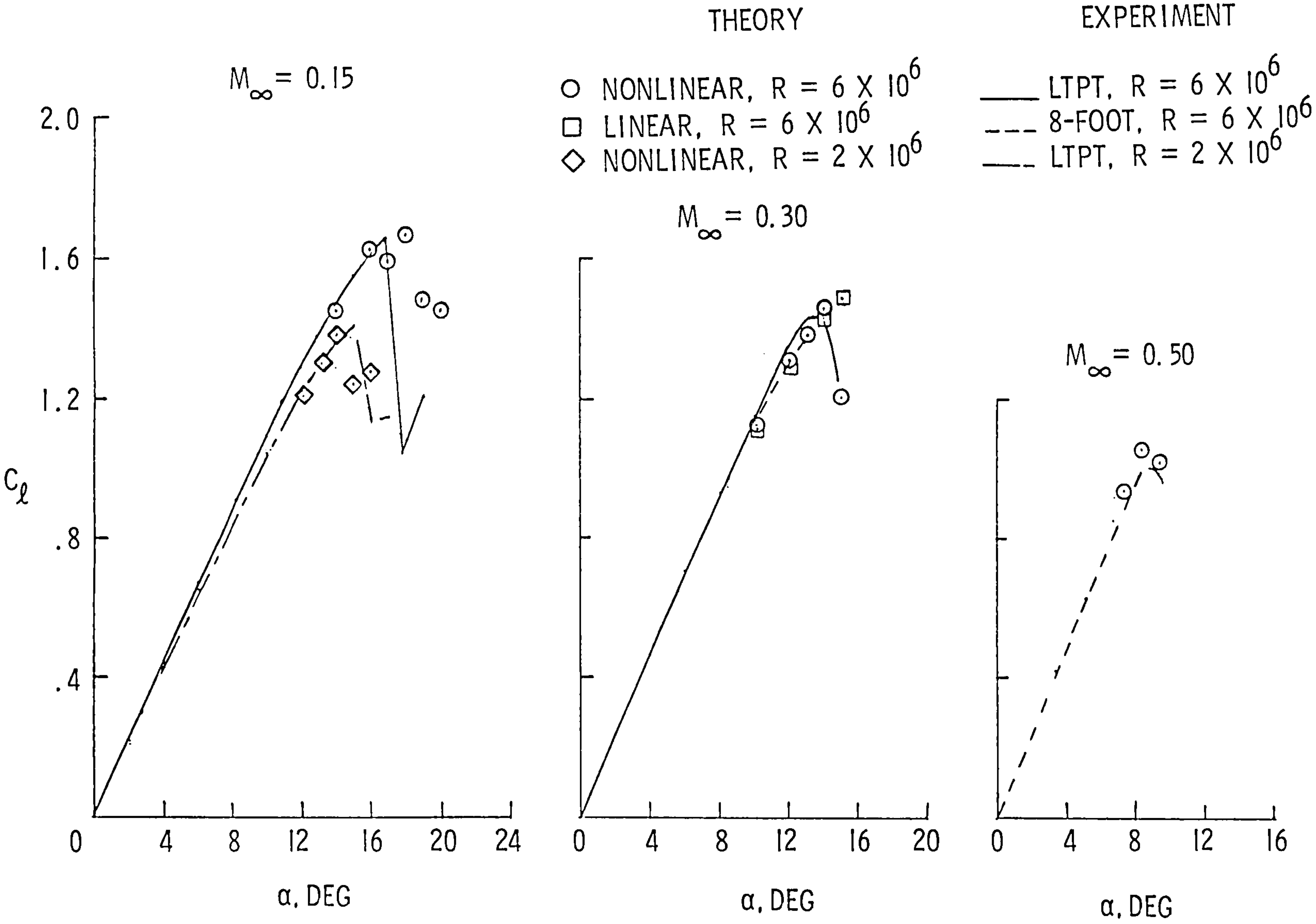
2. Carlson, Leland A; and Rochell, Bruce M.: Application of Direct-Inverse Techniques to Airfoil Analysis and Design. NASA SP-2045, Vol. 1, Part 1, 1978, pp. 55-72.

---

*STAD, 505-31-33, 804-827-4514

PREDICTION OF COMPRESSIBLE TRAILING-EDGE SEPARATION EFFECTS

NACA 0012 AIRFOIL



COMPUTATION OF LEADING-EDGE SEPARATION BUBBLES ON AIRFOILS

Richard W. Barnwell*

Laminar boundary-layer separation can occur in the leading edge region of a blunt airfoil if the Reynolds number is sufficiently low that the flow remains laminar from the stagnation point to the minimum pressure point. Downstream of this peak suction point, the laminar boundary-layer separate almost immediately resulting in a recirculating flow region bounded by a thin shear layer. This separated shear layer, which is very unstable, generally transitions to turbulent flow so that the lower portion of the shear layer is energized by turbulent mixing (fig. 1). If this mixing is sufficiently intense, the flow will reattach quickly and a "short" bubble results. If quick reattachment does not occur, then a "long" bubble results. The "short" bubble has a relatively small effect on airfoil performance, but the "long" bubble has a profound effect and generally indicates the onset of leading-edge stall. The change from "short" to "long" bubble (called bubble bursting) may occur gradually or abruptly depending on the airfoil shape.

J. E. Carter of UTSI will develop a prediction method to analyze leading-edge separation bubbles. In the first phase, a recently-developed subsonic viscous inviscid interaction procedure presented in reference 1 will be combined with a current airfoil analysis code and used to study the "short" bubble problem. In the second phase, the main thrust will be to implement an improved transition model and further develop this "short" bubble analysis to determine its potential for predicting the occurrence of bubble bursting. Parametric studies will be performed to determine the sensitivity of the leading-edge bubble to airfoil incidence shape, Reynolds number and free-stream turbulence level.

The study of O. R. Burggraf of the Ohio State University will also contribute to the understanding of leading-edge separation. Part of this study involves an analytical study of the region around a laminar separation point. From this analysis, insight can be gained which should lead to improved numerical techniques and procedures.

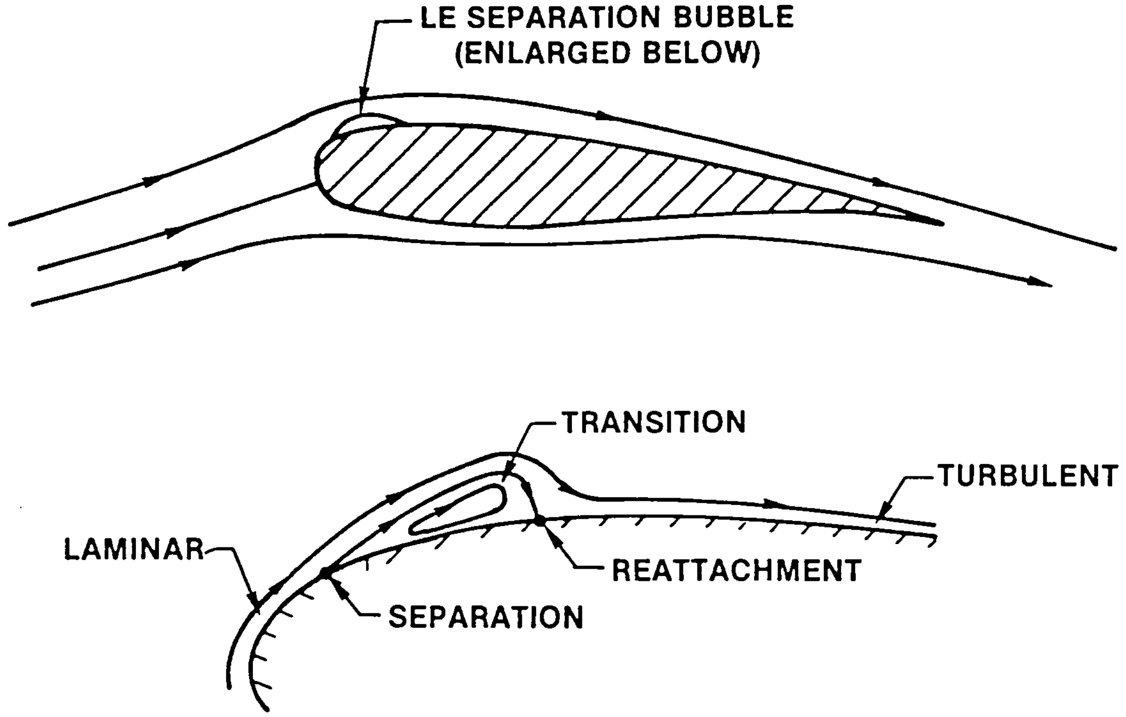
References:

1. Carter, James E,: A New Boundary-Layer Inviscid Iteraction Technique for Separated Flow. AIAA Paper No. 79-1450, 1979.

---

*STAD, 505-31-33, 804-827-4514

AIRFOIL LEADING EDGE SEPARATION BUBBLE


LE SEPARATION BUBBLE
(ENLARGED BELOW)
TRANSITION
TURBULENT
LAMINAR
REATTACHMENT
SEPARATION

PATCHED VISCOUS/INVISCID INTERACTION TECHNIQUES FOR PREDICTING TRANSONIC FLOW OVER BOATTAIL NOZZLES

Richard G. Wilmoth* and Lawrence E. Putnam*

A patched viscous/inviscid interaction technique is used to solve the subsonic/transonic flow over boattail nozzles including the effects of boundary-layer displacement, boundary-layer separation, jet plume blockage and jet entrainment (references 1 and 2). The technique combines (see figure 1) the South-Jameson full potential transonic flow solution for the inviscid external flow with an integral boundary solution, Presz's separation model and discriminating streamline analysis, a shock-capturing inviscid jet exhaust solution and an overlaid, parabolic-marching mixing layer solution. Boundary-layer effects are accounted for in the conventional manner by adding the displacement thickness to the body geometry. For separated flows, the discriminating streamline shape is also included to account for the streamline displacement caused by the separation bubble. Jet effects are accounted for by combining the inviscid jet and mixing layer solutions to define an effective viscous plume boundary which includes both the inviscid plume blockage and jet entrainment contributions. The entire flow field is solved iteratively using a viscous-inviscid underrelaxation procedure.

A comparison with experiments of boattail pressures predicted with this technique are shown in figure 2.

References:

1. Putnam, Lawrence E.: DONBOL: A Computer Program for Predicting Axisymmetric Nozzle Afterbody Pressure Distributions and Drag at Subsonic Speeds. NASA TM-78779, 1979.

2. Wilmoth, Richard G.: Viscous-Inviscid Calculations of Jet Entrainment Effects on the Subsonic Flow Over Nozzle Afterbodies. NASA TP-1626, 1980.

---

*HSAD, 505-32-13, 804-827-2675

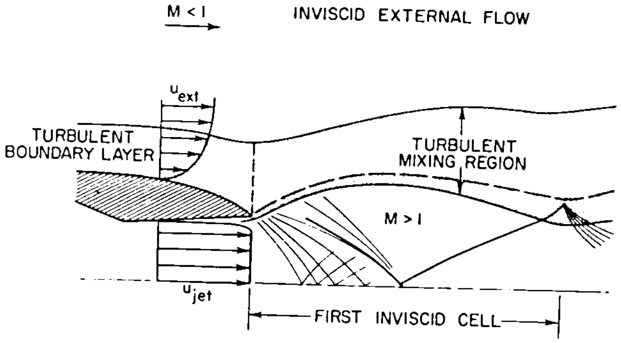
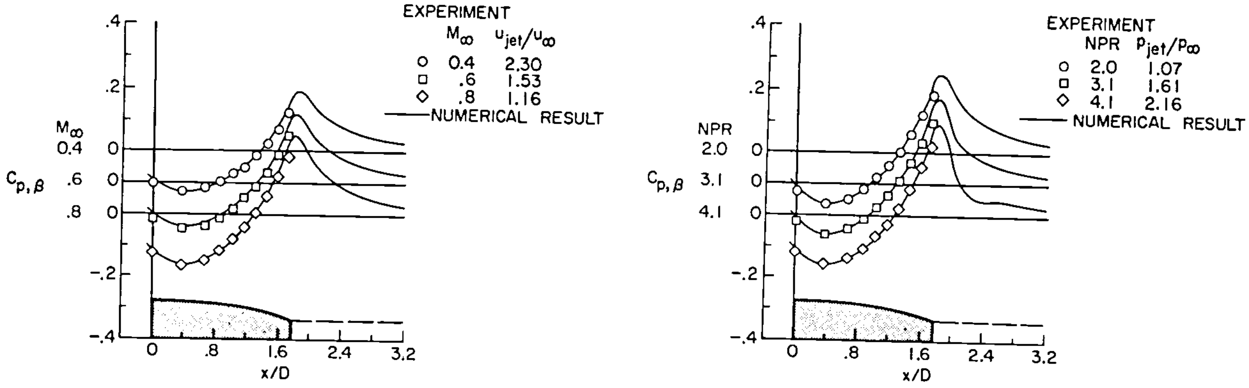

M < 1
INVISCID EXTERNAL FLOW
u_ext
TURBULENT BOUNDARY LAYER
TURBULENT MIXING REGION
M > 1
u_jet
FIRST INVISCID CELL

Figure 1. Schematic of nozzle afterbody flowfield.



(a) Effect of free-stream Mach number at NPR = 2.0.

(b) Effect of NPR at $M_\infty = 0.8$.

Figure 2. Comparisons between predicted and measured boattail pressure distributions for cold-air jet exhaust.

APPROXIMATE CONVECTIVE HEATING EQUATIONS FOR HYPERSONIC FLOW

E. V. Zoby, J. N. Moss, and K. Sutton*

A design problem for most reentry spacecraft is the prediction of convective heating rates. As a consequence of current interest in outer-planet entry and advanced transportation systems for Earth entry, the spacecraft designer now needs rapid, but reliable, heating methods which must be capable of also assessing the effects of such problems as arbitrary reactive gas compositions, complex three-dimensional and/or variable-entropy flows, as well as the possibility of turbulent flows over large surface areas on the spacecraft design.

Based on a recent investigation,[1] an approximate convective heating method has been developed for engineering calculations of laminar and turbulent heating rates. The equations are applicable to nonreacting or reacting gas flows for either constant or variable-entropy edge conditions. The variable-entropy effect on the heat transfer is approximated by defining the boundary-layer edge properties as the inviscid values located a distance from the surface equal to the boundary-layer thickness.

The results of the engineering method are in good agreement with available heat-transfer data, as well as boundary-layer and viscous-shock-layer[2] (VSL) solutions. Note the comparisons of the calculated results are in good agreement for Earth, Venusian, and Jovian entry conditions. Typical comparisons of the results of the present engineering method and the VSL solutions are presented in Figures 1 and 2. The convective heating rate results are presented about a blunt 40° cone in air at $M_\infty = 10$ in Figure 1 and about a blunt 45° cone at a typical Jovian entry condition in Figure 2.

References:

1. Zoby, E. V., Moss, J. N., and Sutton, K, "Approximate Convective Heating Equations for Hypersonic Flows," AIAA Paper 79-1078, June 1979.

2. Moss, J. N., "Stagnation and Downstream Viscous Shock Layer Solutions With Radiation and Coupled Ablation Injection," AIAA Paper 74-73, January 1974.

---

*SSD, Langley, 804-827-2707

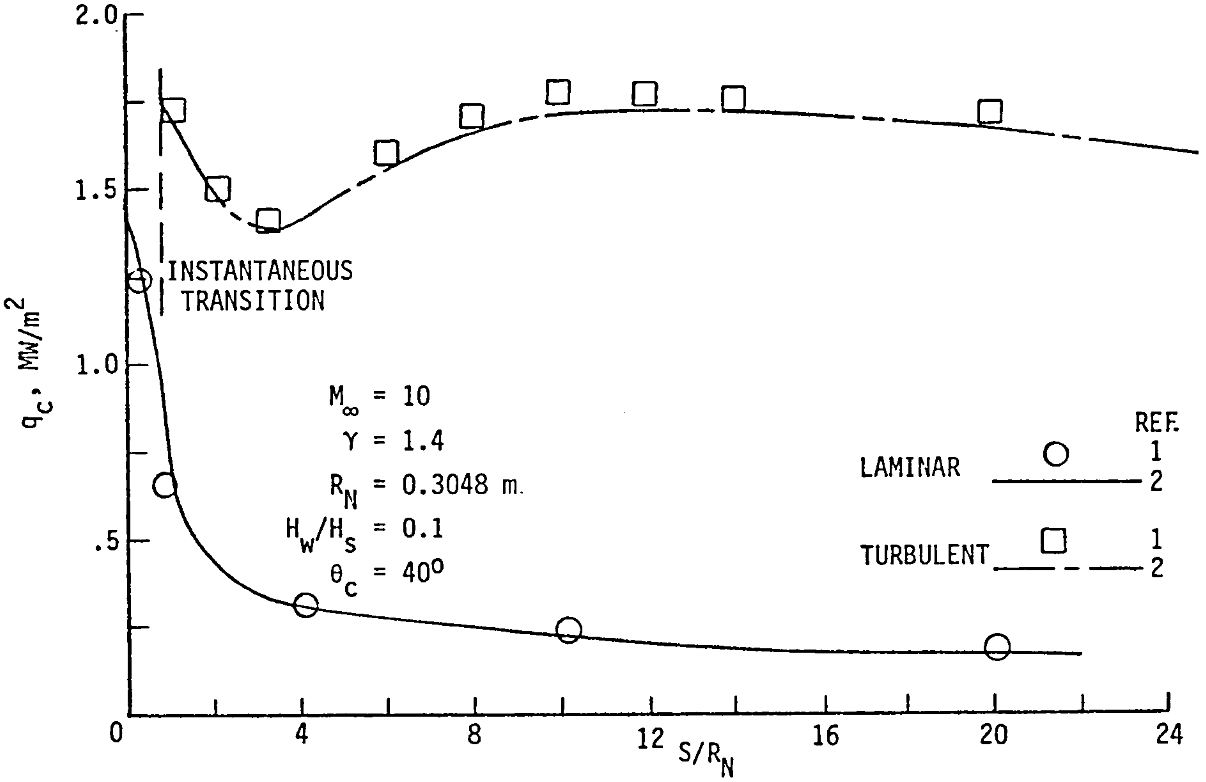


Figure 1.- Comparison of convective heating rate distributions (air-perfect gas).

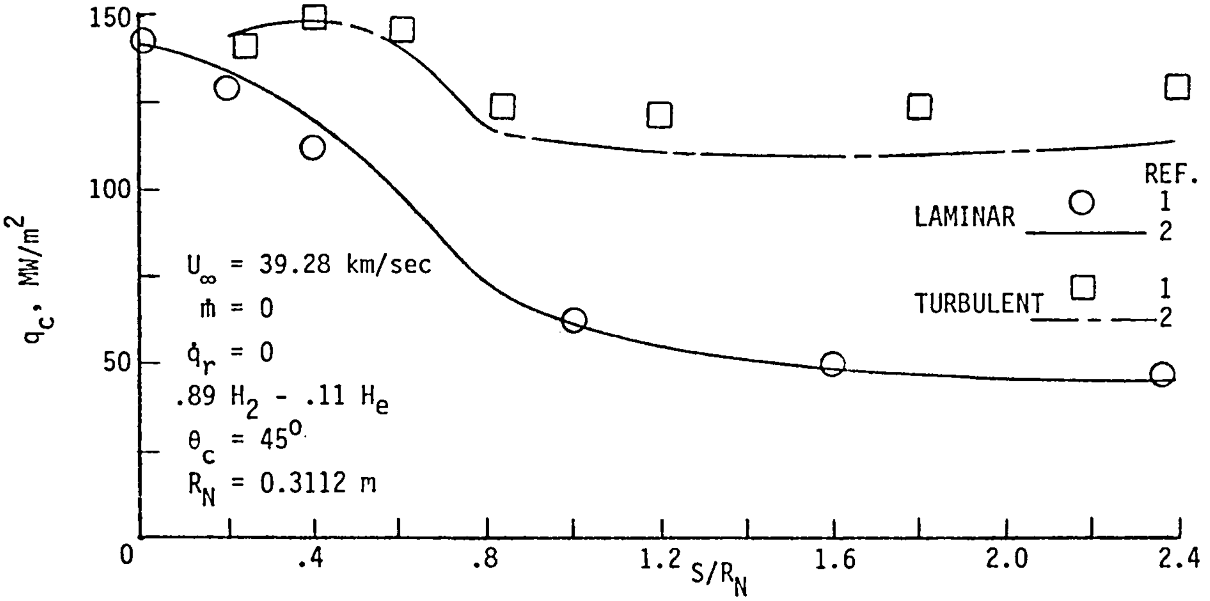


Figure 2.- Comparison of convective heating rate distributions for Jovian atmosphere.

4. Transonic Potential Flow

Methods for solving the irrotational approximation to transonic flow, usually involving the transonic velocity potential or the stream function. This category may often include methods, or validation of methods, which involve viscous/inviscid interaction as a secondary focus.

Steady Flows

Methods designed primarily for solution of the steady problem.

ANALYSIS OF TRANSONIC FLOW PROBLEMS VIA PARAMETRIC DIFFERENTIATION

Woodrow Whitlow, Jr.*

One of the primary difficulties associated with predicting transonic flow fields is that the governing equations are generally nonlinear. Thus, transonic flow equations don't readily lend themselves to formal mathematical analysis. To circumvent the problems caused by the nonlinearity, Rubbert and Landahl[1] used the method of parametric differentiation (MPD) to linearize the small disturbance transonic equation. That method yielded sets of solutions for flow past nonlifting airfoils.

In this study, MPD has been applied in a manner similar to that of Reference 1 but extended to include lifting as well as nonlifting airfoils. Relaxation solutions of the linearized problem were computed on a rectangular grid, and predictor-corrector methods were used to obtain solutions for a range of airfoil thicknesses and angles of attack. A typical set of nonlifting pressure distributions obtained using MPD is shown in Figure 1, and a comparison of a lifting pressure distribution with experimental data[2] is shown in Figure 2. Figure 1 summarizes a set of 21 nonlifting solutions, and Figure 2`summarizes seven nonlifting and two lifting solutions. Each set of solutions was obtained in one application MPD.

Current efforts, being conducted at Langley Research Center, are aimed at utilizing MPD to examine three-dimensional steady and two-dimensional unsteady transonic problems. The extension of MPD to those problems is conceptually straightforward. In addition to thickness and angles of attack, three-dimensional problems offer additional parameters such as wing sweep and twist, and reduced frequency is a natural parameter to use for unsteady problems.

References:

1. Rubbert, P. E.; and Landahl, M. T.: Solutions to the Transonic Airfoil Problem Through Parametric Differentiation. AIAA Journal, vol. 5, pp. 470-479, 1967.

2. Knechtel, E. D.: Experimental Investigation at Transonic Speeds of Pressure Distributions over Wedge and Circular-Arc Airfoil Sections and Evaluation of Perforated-Wall Interference. NASA TN D-15, 1959.

---

*SDD, 505-02-23, 804-827-2661

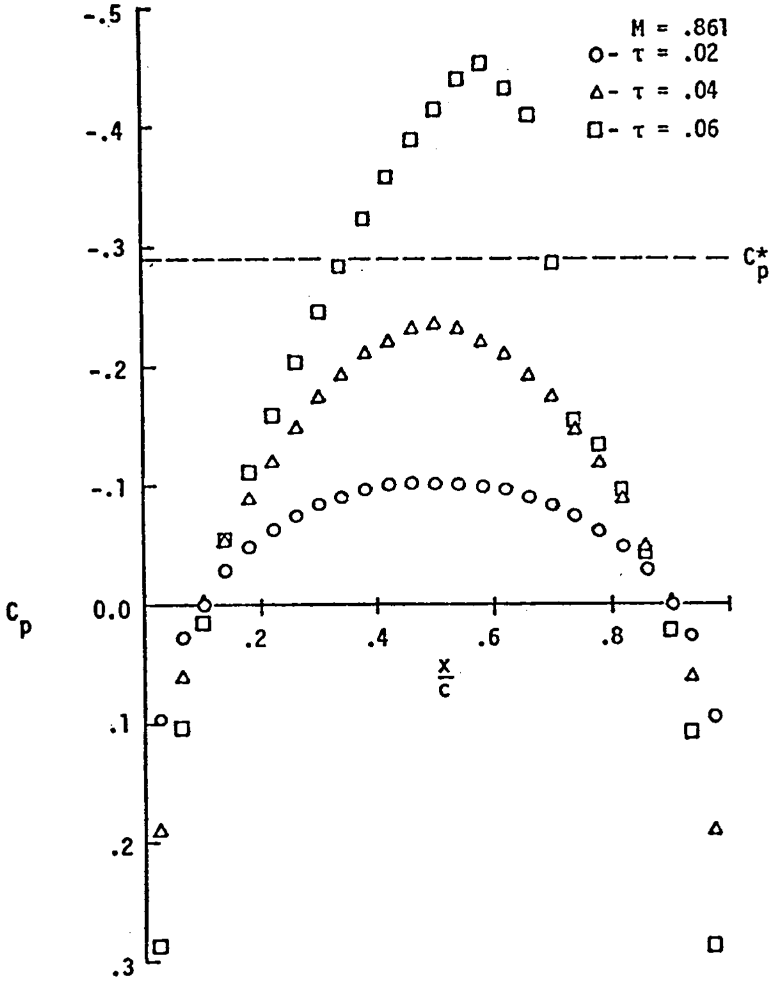


Figure 1 (Above). Nonlifting pressure distributions on biconvex airfoils.

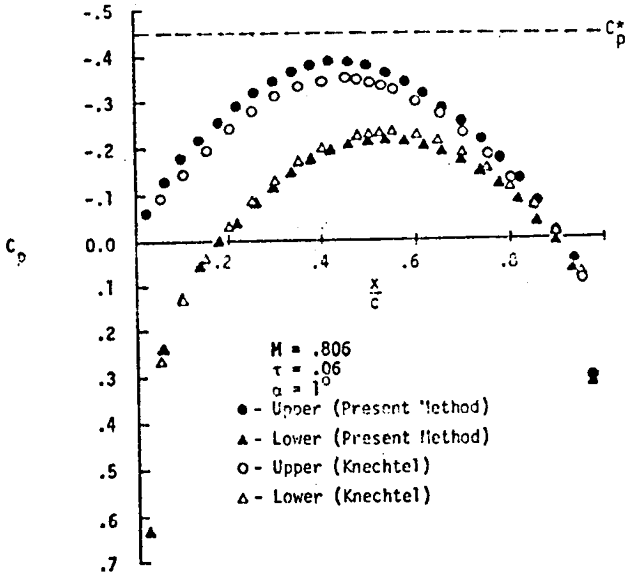


Figure 2 (Right). Lifting pressure distributions on a six percent thick biconvex airfoil at one degree angle of attack.

ANALYSIS OF ITERATIVE METHODS FOR THE TRANSONIC POTENTIAL EQUATION

John C. Strikwerda*

Several mathematical aspects of iterative methods for solving the transonic potential equation are being examined. At present, the study is concentrating on a one-dimensional model problem with a simple iteration method so that the effect of various iteration parameters on the behavior of the convergence process can be identified. The research will be expanded in the near future to include two-dimensional problems and commonly used iterative methods. This study should provide a better understanding of how to choose appropriate values of the iteration parameters for several methods currently in use.

---

*ICASE, 505-31-83-01, 804-827-2513

DESIGN OF SHOCKLESS TRANSONIC AIRFOILS BY THE METHOD OF COMPLEX CHARACTERISTICS

José Sanz*

The method of complex characteristics developed by Bauer, Garabedian and Korn has proved to be an excellent tool for the design of shockless airfoils. The last version of their code (Ref. 1) includes a fundamental improvement in the design method which consists in mapping the hodograph domain onto a circle thus, allowing one to design airfoils with a given input pressure distribution. In collaboration with NASA-Lewis and using a simple modification of the above code, compressor blades were able to be designed with a gap to chord ratio of .89 and a turning angle of 41 degrees, as shown in Figure 1.

However, an important problem in turbomachinery technology is the ability to design shockless cascades with gap to chord ratios of order one half, i.e., solidity of up to 2. To get this capability a new code is in an advanced state of development. The main new concept included is a conformal mapping of the hodograph domain onto an ellipse rather than onto a circle. This implies a new method of laying paths of integration to solve the finite difference scheme which approximates the differential equations modeling the flow. These new paths of integration are more in accord with the topology that the new mapping introduces and are producing a substantial reduction in computing time.

The new code is being developed with support by the NASA-Lewis Research Center. We expect that the new code will also be useful in the design of thin isolated airfoils.

References:
1. Bauer, Garabedian, and Korn "Supercritical Wing Sections III" Springer-Verlag, 1977.

---

*ICASE, 505-31-83-01, 804-827-2513

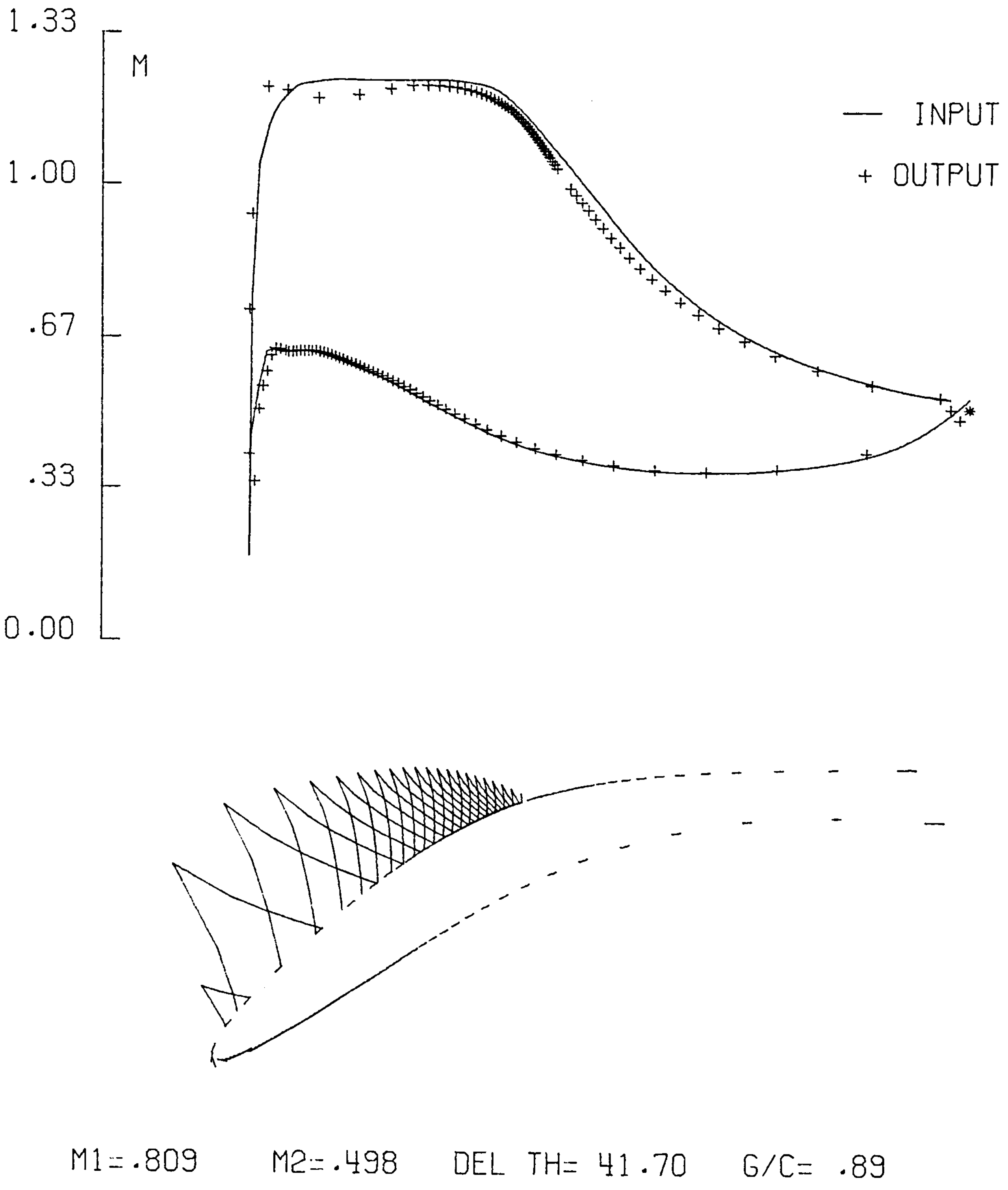


Figure 1. Compressor Blade Design Using B.G.K. code

APPLICATION AND EVALUATION OF TRANSONIC THEORIES FOR DESIGNING SUPERCRITICAL FIGHTER WINGS FOR TRANSONIC MANEUVER

Michael J. Mann*

The recent development of three-dimensional transonic computational methods has opened the possibility of new design procedures for aircraft operating at transonic conditions. The purpose of the present study has been to develop a theoretical procedure for the design of transonic wings. This procedure (ref. 1) uses the three-dimensional transonic analysis codes in an iterative fashion to achieve a desired pressure distribution. The method applies to conditions involving a large region of supersonic flow on the wing. Basic principles of supersonic flow are utilized to determine the necessary change in airfoil geometry to produce a desired change in pressure.

The design procedure has been used to design both a swept-back wing and a comparable swept-forward wing for a fighter operating at transonic maneuvering conditions. The airfoils were designed to have a large region of supersonic flow which terminates with a weak shock wave. The fuselage effects were accounted for in both designs. The swept-forward wing configuration includes a canard and was designed with a transonic wing-body-canard code. The development of the design procedure, its application, and the associated experimental testing have all been accomplished at the NASA Langley Research Center.

The application of the design procedure to the swept-back wing, SMF-2, is shown in figures 1 and 2. The Mach number is .90 and the lift coefficient is 0.85. In figure 1, the Jameson-Caughey FLO 27 (ref. 2) three-dimensional transonic computer code is correlated with experimental data on an early version of the SMF-2 configuration. The fuselage is represented by an infinite cylinder and a strip boundary layer theory was used to estimate viscous effects. The design procedure was then used to modify the wing shape in order to remove the forward part of the lambda shock and reduce the strength of the trailing-edge shock. The improved pressure distribution is compared with the original one in figure 2. These results were computed with the FLO 27 code.

References:

1. Mann, Michael J.: The Design of Supercritical Wings by the Use of Three-Dimensional Transonic Theory. NASA TP 1400, 1979.

2. Jameson, Antony and Caughey, D. A.: A Finite Volume Method for Transonic Potential Flow Calculations. AIAA payer No. 77-635, June 1977.

---

*STAD, 505-43-23-06, 804-827-3711

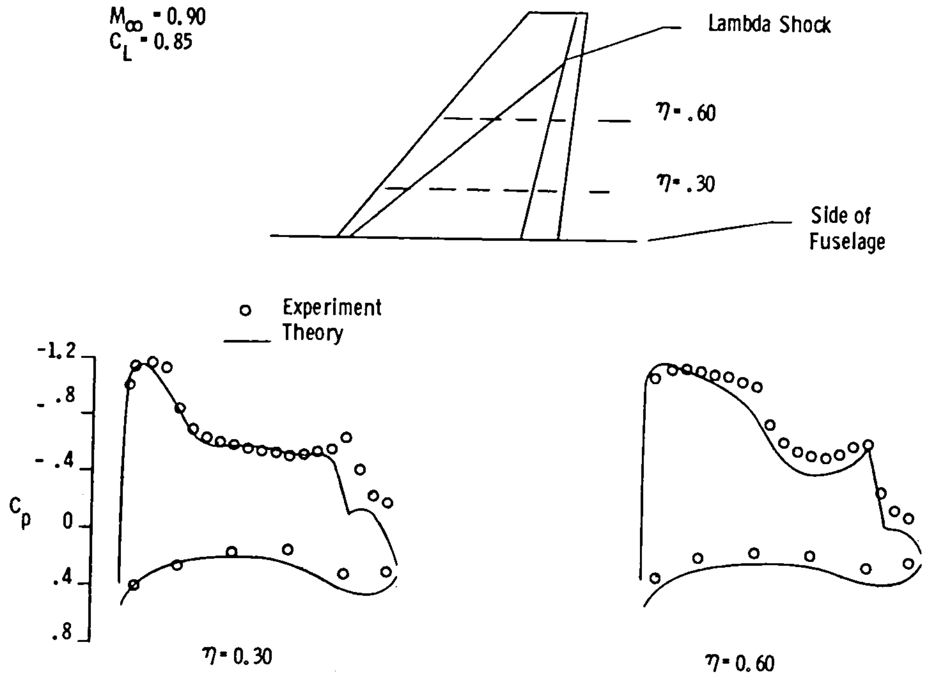


Figure 1.- Early Configuration of SMF-2 Wing.

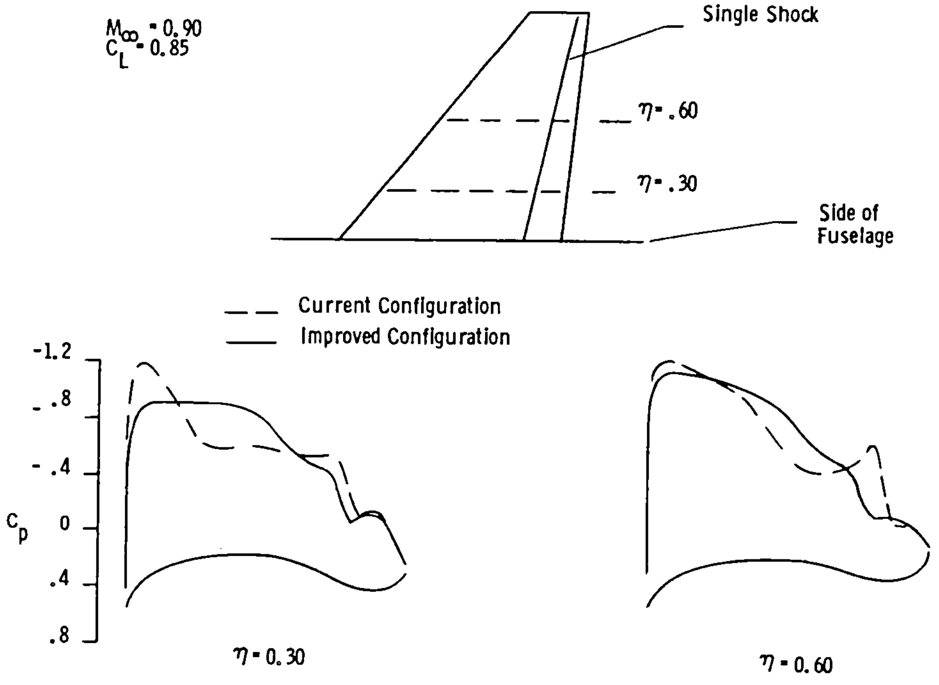


Figure 2.- Theoretical Modification to SMF-2 Wing.

IMPROVEMENTS IN THE ARTIFICIAL DENSITY METHOD FOR COMPUTATIONAL TRANSONICS

Jerry C. South, Jr.*

The artificial-density method (ADM) has been shown as an efficient method for including necessary dissipation terms in finite-difference and finite-element transonic-flow calculations (ref. 1). However, the method usually produced overshoots at mesh points just upstream of shocks; the overshoots were eliminated by increasing the numerical dissipation coefficient at the expense of more "smearing" of the shock jump. An improved procedure has been developed which greatly diminishes the shock overshoots without increasing the dissipation. It was found that the difficulty stemmed from the density calculation.

Space does not permit presentation of the details; however, the improved method uses a more "compact" difference operator. In the original method, even at subsonic flow points, the difference operator spread over five points in each direction; the latest method only requires three. Some comparisons of the two methods are shown in figures 1 and 2. The original method is identified as "node-point density" and the new method as "midsegment" or "midcell density." Figure 1 shows results for 1-D flow in a constant-area duct with a normal shock. The original method gives an overshoot upstream from the shock while the new "midsegment density" method gives practically no discernible overshoot. The method has been tested extensively in 2-D transonic flows, an example of which is shown in figure 2 for a nonlifting, NACA 0012 airfoil at $M_\infty = 0.85$. Here again we see that the original method gives an overshoot at the shock, while none is evident in the "midcell" density method. Versions of the new method are now being tested in 3-D flows using vectorized algorithms for the CYBER 203 computer.

References:

1. Hafez, M.; South, J. C., Jr.; and Murman, E. M. "Artificial Compressibility Methods for Numerical Solution of Transonic Full-Potential Equation" AIAA J., vol. 17, no. 8, pp. 838-844, Aug. 1979.

---

*STAD, 505-31-13, 804-827-2627

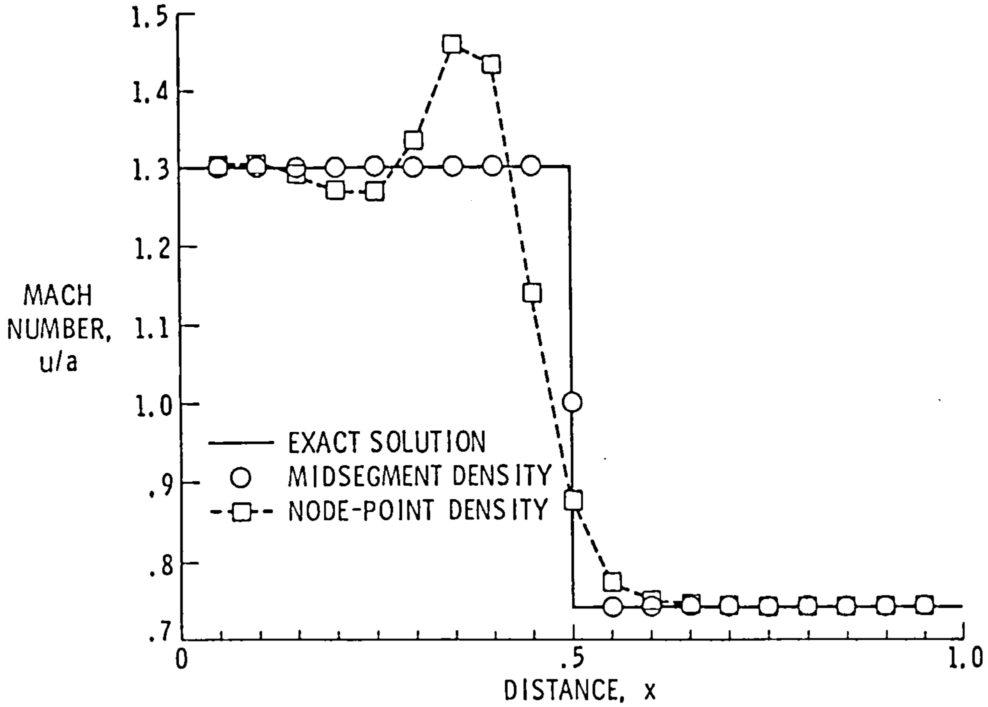


Figure 1.- Resolution of normal shock wave using two versions of artificial density method.

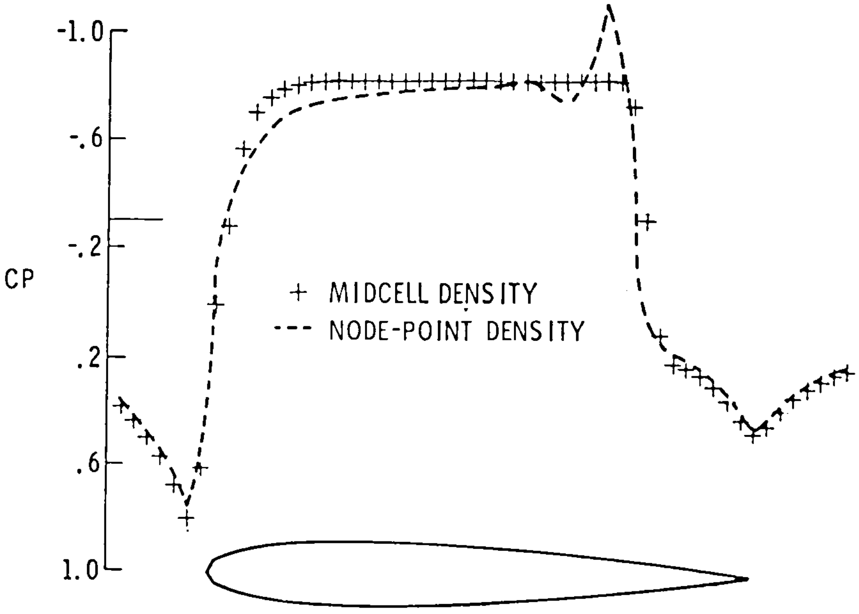


Figure 2.- Pressure distribution for NACA 0012 airfoil. (M_∞ = .85, α = 0, 121 × 25 mesh).

DEVELOPMENT OF A FULLY CONSERVATIVE, AXISYMMETRIC TRANSONIC FLOW CODE

Lawrence L. Green*

Technology has been developed for analysis of transonic, axisymmetric bodies by South and Jameson in reference 1. A nonconservative version of the computer code was developed and distributed by Keller and South in reference 2. The present project is to apply the artificial compressibility method presented by Hafez, Murman, and South, reference 3, to the RAXBOD code and develop a fully conservative transonic flow code for axisymmetric bodies.

Primary modifications have been successfully completed and preliminary results for a transonic sphere are shown in the accompanying figure. For comparison, the results for the same case are shown from the nonconservative RAXBOD code. The conservative results show increased shock strength and displacement of the shock somewhat downstream of the nonconservative version, as expected. Comparisons will be made in the future with the work of Salas, involving solutions to the Euler equations for similar problems, described elsewhere in this compendium.

References:
1. South, Jerry C., Jr.; and Jameson, Antony "Relaxation Solutions for Inviscid Axisymmetric Transonic Flow Over Blunt or Pointed Bodies" AIAA Proc. Comp. Fluid Dyns. Conf., Palm Springs, Calif., July 1973.
2. Keller, James D.; and South, Jerry C., Jr. "RAXBOD: A Fortran Program for Inviscid Transonic Flow Over Axisymmetric Bodies" NASA TM X-72331, Feb. 1976.
3. Hafez, M. M.; South, Jerry C., Jr.; and Murman, Earll M. "Artificial Compressibility Methods for Numerical Solution of Transonic Full Potential Equation" AIAA Paper No. 78-1148, July 1978.

---

*STAD, 505-31-13, 827-2627

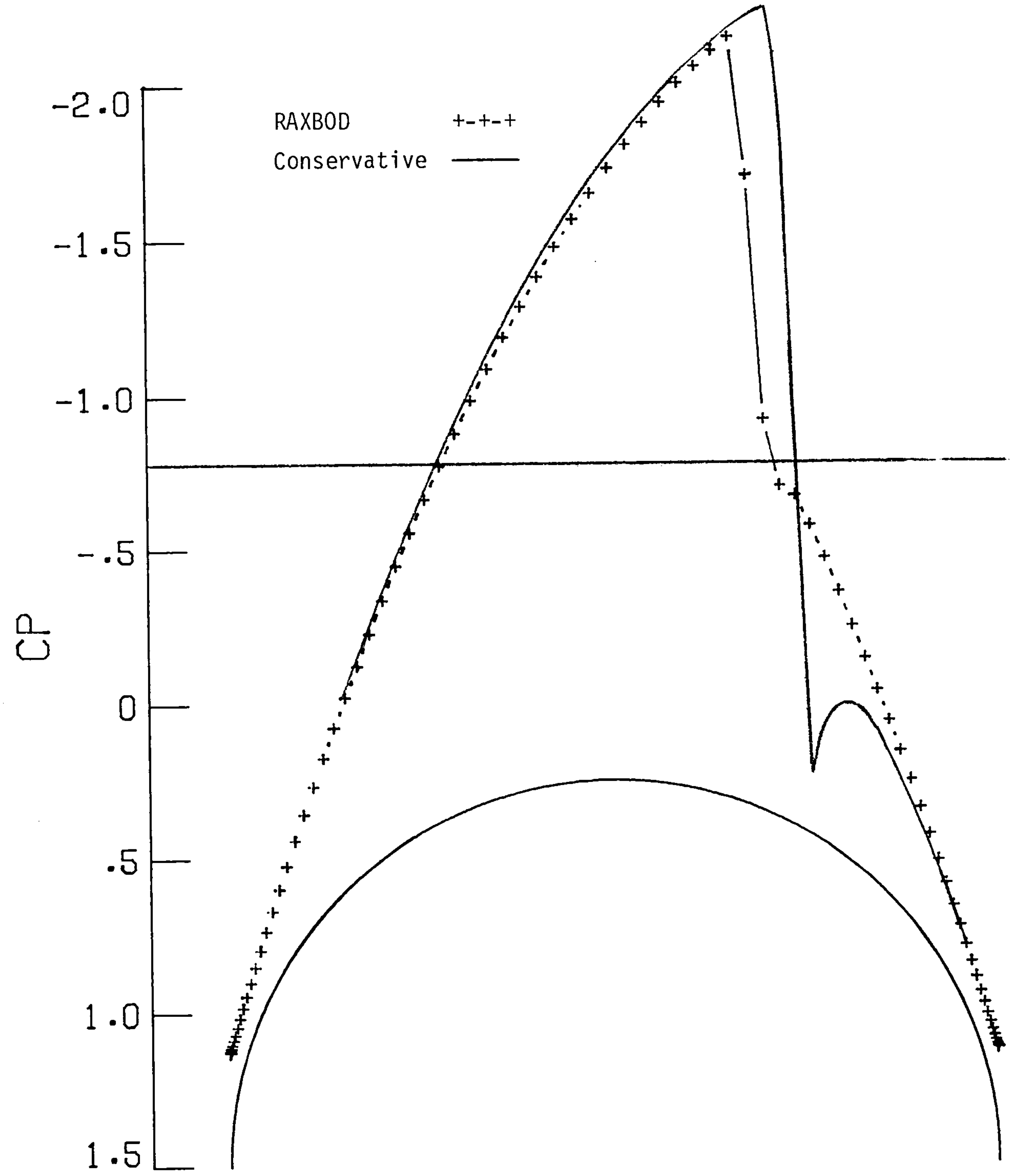


Figure.- Pressure coefficient for transonic sphere, $M_\infty = 0.7$.

DEVELOPMENT OF VECTOR ALGORITHMS FOR COMPUTATIONAL TRANSONICS

James D. Keller*

The state of the art in transonic flow computations has advanced to the point where two-dimensional flows, including boundary-layer interaction, can be computed efficiently on current computers. Three-dimensional calculations, however, are very time consuming and expensive on conventional computers such as the Control Data CYBER-175. If we can make proper use of the new "super" computers such as the CYBER-203, we should be able to reduce the cost of three-dimensional transonic calculations. Unfortuntely, the algorithm which has been used extensively in the past (successive line over-relaxation) is not very well-suited to the new vector processor type of computers. This was particularly true of the STAR-100, which had a rather slow speed on scalar operations.

Reference 1 gives the details of a number of different algorithms which are not only vectorizable, but which also have a good rate of convergence. The one which appears most attractive so far is called ZEBRA II. It is relatively easy to vectorize and has a convergence rate which is the same as SLOR. The ZEBRA II scheme involves very few scalar operations, the shortest vector operations have a vector length equal to half the number of points in a cross plane, and the scheme has no storage problems on a virtual memory computer. The ZEBRA II scheme has been implemented in a small pilot code for flow over a swept wing using the artificial density method (ref. 2). Computer times for this pilot code on various computers are given in Table 1. This pilot code can handle only the simplest swept-wing geometry and uses Cartesian coordinates and planar boundary conditions. Efforts are now underway to implement the ZEBRA II algorithm into production-type three-dimensional codes. An in-house effort is underway to put the ZEBRA II algorithm into the FLO-22 code which was developed by Antony Jameson at New York University. This code solves the full-potential equation for a swept wing using a parabolic coordinate system. Computer Sciences Corporation has been given a task to implement the ZEBRA II algorithm in the WIBCO code developed by Charles Boppe at Grumman. This code solves the transonic small-disturbance potential equation for a swept wing using a grid embedding technique.

References:

1. South, Jerry C., Jr.; Keller, James D.; and Hafez, Mohamed M. "Vector Processor Algorithms for Transonic Flow Calculations" AIAA Paper No. 79-1456, July 1979.

2. Hafez, Mohamed M.; South, Jerry C., Jr.; and Murman, Earll M. "Artificial Compressibility Methods for Numerical Solution of Transonic Full Potential Equation" AIAA Paper No. 78-1148, July 1978.

---

*STAD, 505-31-13, 827-2627

Table 1.- Timing comparison for ZEBRA II algorithm on a 3-D transonic flow code. Approximately 30,000 grid points, 400 iteration cycles, density updated every 4th cycle.

MACHINE	CPU TIME (SEC.)
CYBER-175	456
CDC 7600	282
CRAY-1 without vectorization	144
CRAY-1 with partial vectorization	69
STAR-100 vectorized code	58
CYBER-203 vectorized code	53

BASIC RESEARCH IN CFD AT NYU

Jerry C. South, Jr.*

Under NASA Grant NSG 1579, the NYU Research Team has been developing numerical methods for transonic flows. The team presently consists of Prof. Paul Garabedian (director of the Fluid Dynamics Division, Courant Institute of Mathematical Sciences), Prof. Antony Jameson and Drs. Frances Bauer, Octavio Betancourt, and Geoffrey McFadden. This effort has been a continuing one since 1970, and the past accomplishments are well known in the field of computational transonics: The complex-characteristics hodograph design method for shockless supercritical airfoils; the inviscid, circle-plane, mixed finite-difference airfoil analysis code; theory for iterative schemes for mixed flows; the 3-D yawed and swept wing codes used throughout industry and Government; the viscous 2-D airfoil "BGK" code; the conservative full-potential equation 2-D code; and the 3-D finite-volume fully conservative wing-body code. In 1976, the team was given the NASA Group Public Service Award.

Most recently the NYU team has focused on two problems: The transonic wing design problem and the multigrid relaxation method for transonic flows. The 3-D wing design method is based partly on the 2-D design method developed by Dr. McFadden in his dissertation (ref. 1). For a given input pressure or speed distribution, the ordinates of the wing sections are adjusted iteratively in such a way as to move the pressures calculated from the NYU FLO-22 swept-wing analysis code (ref. 2) toward the desired input values of pressure. Different formulas are used at subsonic and supersonic points to adjust the surface ordinates. Preliminary results have been obtained on coarse meshes.

The multigrid research has led to the most efficient 2-D circle-plane transonic airfoil code in existence. Jameson's "MAD" scheme (ref. 3) (for Multigrid with Alternating-Direction relaxation) achieves convergence for transonic airfoils in 30 to 80 cycles, with a spectral radius of about 0.6 to 0.8 on fine meshes. This research is being pursued further for 3-D wing applications.

References:
1. McFadden, Geoffrey B. "An Artificial Viscosity Method for the Design of Supercritical Airfoils" Courant Math. and Computing Lab. Rep. COO-3077-158, July 1979.
2. Jameson, Antony; and Caughey, D. A. "Numerical Calculation of the Transonic Flow Past a Swept Wing" Courant Institute of Math. and Sciences, ERDA Math. and and Computing Lab. Rep. COO-3077-140, June 1977.
3. Jameson, Antony "Acceleration of Transonic Potential Flow Calculations on Arbitrary Meshes by the Multiple Grid Method" Proceedings of AIAA Computational Fluid Dynamics Conference, Paper No. 79-1458, July 1979.

---

*STAD, 505-31-13, 804-827-2627

COMPUTATION OF ROTATIONAL TRANSONIC FLOW USING THE STREAM FUNCTION

Jerry C. South, Jr.*

Dr. Mohamed Hafez has recently joined the GWU faculty at LaRC and will specialize in computational fluid dynamics. Recently, he developed a technique for solving the nonlinear stream function equation for transonic flow. The great advantage of using the stream function, ψ, rather than the potential function, ϕ, is that rotational flows can be analyzed with ψ, whereas rotation is excluded by the potential. The conservative equation satisfied by the stream function for rotational flow is:

$$\left(\frac{\psi_x}{\rho}\right)_x + \left(\frac{\psi_y}{\rho}\right)_y = \omega \qquad (1)$$

where $\rho u = \psi_y$, $\rho v = - \psi_x$ (2)

$$u_y - v_x = \omega \qquad (3)$$

$$\rho = \rho_\infty e^{-\frac{S - S_\infty}{R}} \left\{1 - \frac{\gamma - 1}{2} M_\infty^2 (u^2 + v^2)\right\}^{\frac{1}{\gamma - 1}} \qquad (4)$$

and the entropy S and the rotation ω are functions of ψ alone. Some of the earliest attempts at computational transonics used the stream function formulation, but they generally failed because of difficulties in extracting the density ρ as a function of the derivatives ψ_x and ψ_y. The density is a double-valued function of ψ, whereas it is single-valued in terms of the speed, as in equation (4). In the old formulations, the speed components were calculated by $u = \psi_y/\rho$, $v = - \psi_x/\rho$, making equation (4) a nonlinear equation for ρ with a branch at the sonic speed. Hafez' idea is to obtain the speed components by integrating equation (3) after each iteration on ψ; the approach has worked nicely for irrotational flow, $\omega \equiv 0$ everywhere, agreeing well with potential calculations. Work will proceed during summer 1980 to carry out calculations for rotational transonic flow past cylinders and airfoils, comparing with Euler equation solutions of Salas (see this compendium) and potential calculations.

---

*STAD, 505-31-13, 804-827-2627

DEVELOPMENT OF A SMALL-DISTURBANCE EMBEDDED-GRID CODE FOR TRANSONIC WING/BODY/NACELLE/PYLON CONFIGURATIONS

Perry A. Newman*

Extension of transonic-flow computational techniques from 2-D airfoil sections to more realistic 3-D configurations is hindered by: (a) The computational resources required for code development and validation as well as (b) the need for a suitable coordinate system. A computational grid embedding technique (ref. 1) solves, to some extent, both of these problems. (See fig. 1.) This technique was initiated by Charles W. Boppe while an NASA-Grumman Research Associate at Langley; continuing development has been done at Grumman under contract NAS1-14732 (refs. 2-4). A wing-body code has been delivered and documented (ref. 5) while the wing/body/pod/pylon/winglet code (WBPPW) is in the process of being delivered to NASA.

Comparisons of initial calculations from the WBPPW code with experimental results for a number of configurations are given in references 3 and 4; the attached figure 2 is an example. Throughout the development of this technique for complex configurations, simplicity has been stressed. Coordinate systems are essentially rectangular in character, boundary conditions are planar, linear interpolations are used for multiple grid interactions, a simple two-dimensional strip boundary-layer analysis provides viscous effects, and, finally a fast, easy to use fuselage modeling system yields arbitrary body surface normals. When compared to other computational methodology, this approach lacks sophistication. However, this simplicity plays an important role in providing the flexibility required to treat complex configurations.

References:

1. Boppe, C. W. "Calculation of Transonic Wing Flows by Grid Embedding" AIAA Paper No. 77-207,. Jan. 1977.
2. Boppe, C. W. "Computational Transonic Flow About Realistic Aircraft Configurations" AIAA Paper 78-104, Jan. 1978.
3. Boppe, C. W.; and Stern, M. A. "Simulated Transonic Flows for Aircraft with Nacelles, Pylons, and Winglets" AIAA Paper No. 80-130, Jan. 1980.
4. Boppe, C. W.; and Aidala, P. V. "Complex Configuration Analysis at Transonic Speeds" AGARD Subsonic/Transonic Configuration Aerodynamics Symposium, May 1980.
5. Boppe, C. W. "Transonic Flow Field Analysis for Wing-Fuselage Configurations" NASA CR-3243, Feb. 1980.

---

*STAD, 534-02-13, 804-827-2627

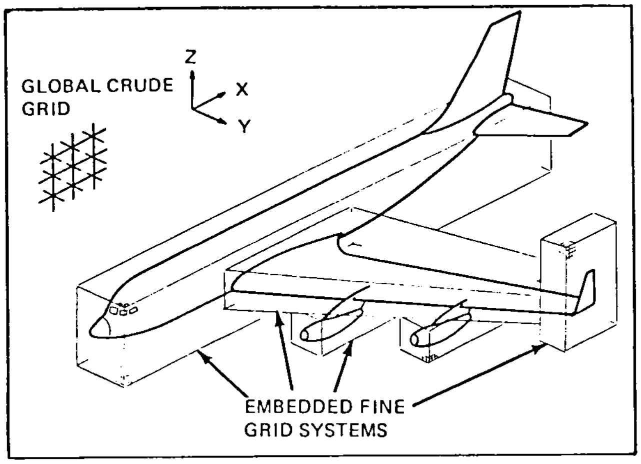

GLOBAL CRUDE GRID
Z
X
Y
EMBEDDED FINE GRID SYSTEMS

Fig. 1 Multiple grid system approach for complex aircraft.

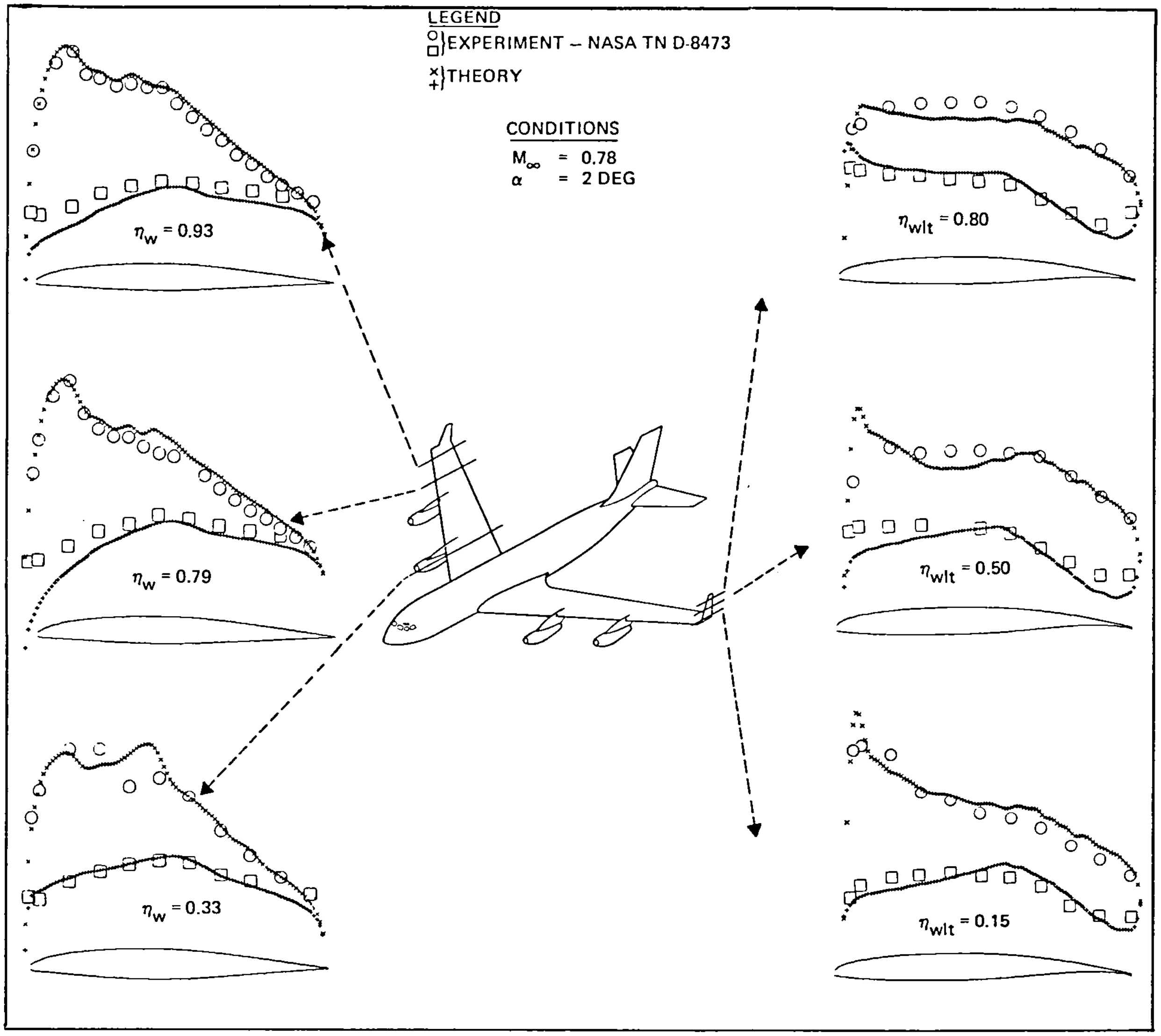


Fig. 2 Boeing KC-135 wing and winglet pressure distribution correlation.

VALIDATION OF A SMALL-DISTURBANCE EMBEDDED-GRID CODE FOR TRANSONIC WING/BODY/NACELLE/PYLON CONFIGURATIONS

Perry A. Newman*

Validation of the Boppe/Grumman 3-D transonic analyses codes (ref. 1) is being done using the experimental data base for the NASA Supercritical-Wing EET model. This work was initiated by E. G. Waggoner while a NASA-Vought Research Associate at Langley. Some results obtained from the Wing/Body code (WIBCO) studies are presented in reference 2. Validation studies using the Wing/Body/Pod/Pylon/Winglet code (WBPPW) are being continued by Waggoner under contract NAS1-16055.

These validations consist of comparing the results predicted by the codes with the carefully documented data base generated as the NASA supercritical-wing EET model evolved through design changes made and tested in the Langley 8-Foot Transonic tunnel. These include addition of nacelles, pylons, and winglets to the clean wing-body model. The attached figure (from ref. 2) shows wing planform and inboard airfoil section changes made to the baseline wing-body configuration. The table presents a comparison of the resulting experimental and calculated force and moment increments. These comparisons show that the computational results are sensitive to subtle design modifications for the wing-body transport configurations indicating that the code can be used as an effective guide in the design process. Validation studies have been initiated on the Wing/Body/Pod/Pylon/Winglet configurations using the newly developed WBPPW code.

References:

1. See preceding compendium summary entitled "Development of a Small-Disturbance Embedded-Grid Code for Transonic Wing/Body/Nacelle/Pylon Configurations."

2. Waggoner, E. G. "Computational Transonic Analysis for a Supercritical Transport Wing-Body Configuration", AIAA Paper 80-129, Jan. 1980.

---

*STAD, 534-02-13, 804-827-2627

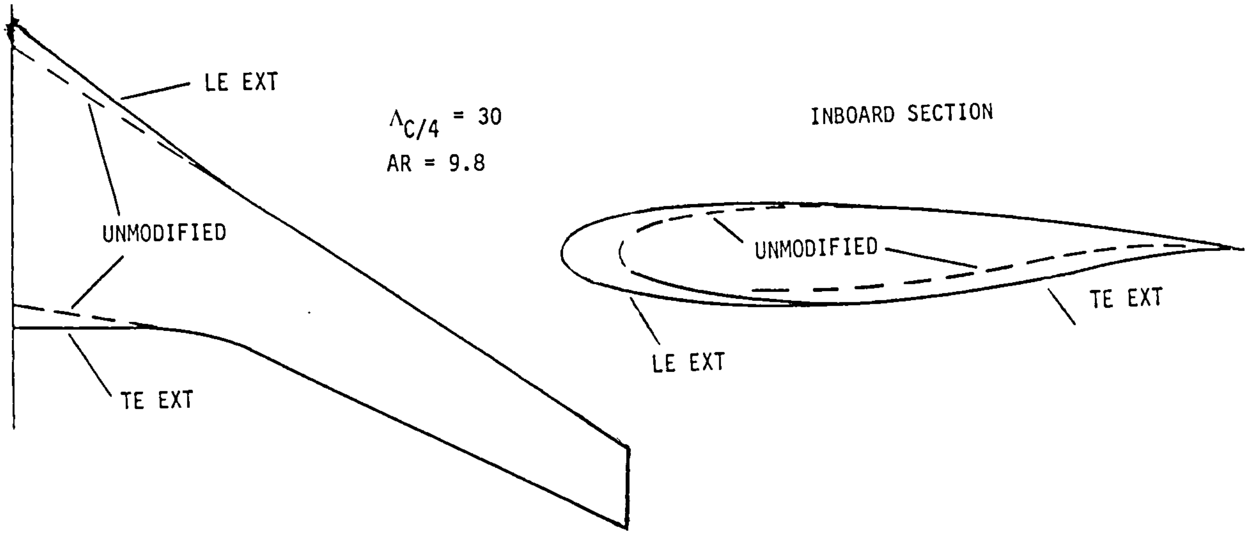
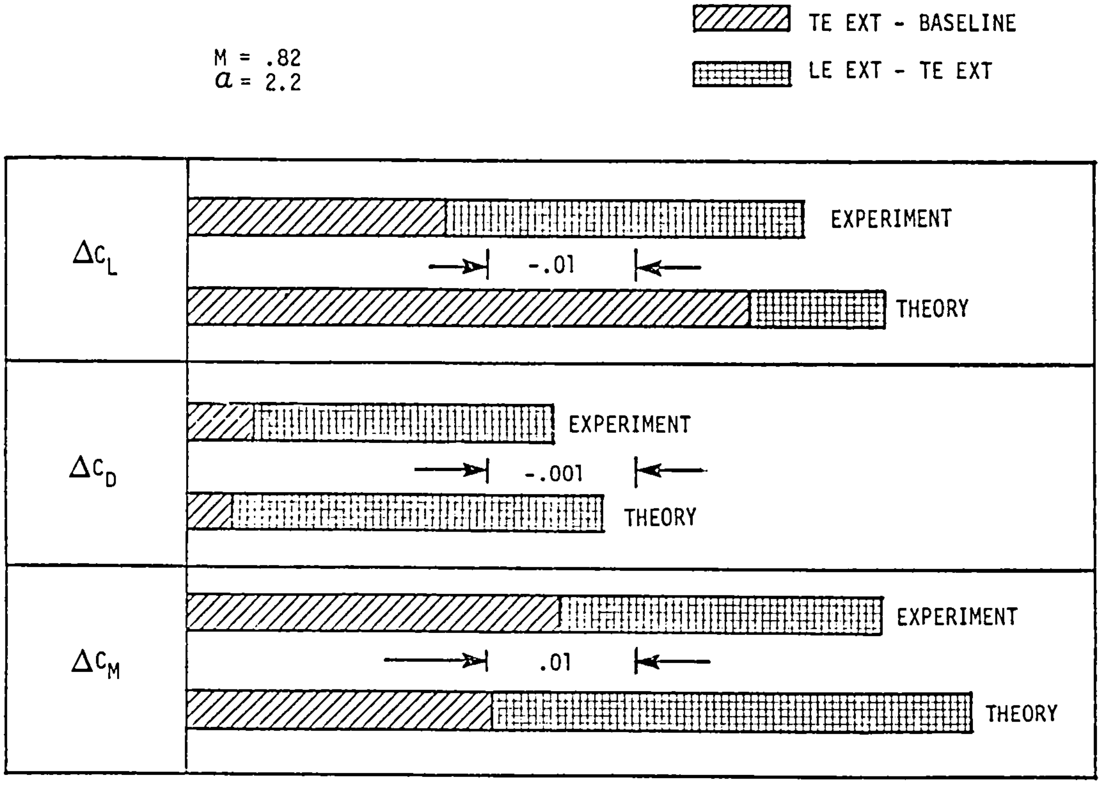

LE EXT
UNMODIFIED
TE EXT
$\Lambda_{C/4}$ = 30
AR = 9.8
INBOARD SECTION
UNMODIFIED
TE EXT
LE EXT

Figure Root Section and Planform Modifications to the Selected Wing

Table Comparison of Incremental Forces and Moments Between the Three Study Configurations



METHOD FOR NUMERICAL DESIGN OF A CONTOURED WIND-TUNNEL LINER FOR TEST OF A LAMINAR-FLOW-CONTROL SYSTEM FOR A YAWED SUPERCRITICAL AIRFOIL MODEL

Perry A. Newman*, and
E. Clay Anderson*

The present numerical design procedure was developed in order to determine the shape of a contoured, nonporous, wind-tunnel liner for use in the Langley 8-Foot Transonic tunnel test of a large-chord, laminar flow control (LFC), swept-wing panel which has a supercritical airfoil section. The procedure is based upon a simple idea and several existing computer codes which make it feasible. Basically, one determines bounding streamlines in the desired unbounded flow, makes all required blockage corrections, fairs into the existing tunnel, and finally builds the resulting contoured liner. The procedure has a general utility which is restricted by our current ability to calculate the viscous transonic flow field about arbitrary configurations. A brief description and an application to the 2-D streamlined tunnel problem were given in reference 1. Aerodynamic lines for the design point test condition have been generated using the present method; data files of them are now being processed into tapes for numerically controlled milling of the liner blocks. The ultimate test of the propriety of the design procedure is the LFC experiment, currently scheduled to begin in mid CY 1981.

The attached figure (taken from ref. 1) shows the agreement between calculated and experimentally determined wall shape for a 2-D streamlined tunnel test conducted by R. W. Barnwell and J. L. Everhart of NASA,(STAD) LRC (data unpublished). The symbols represent their experimentally determined wall shape; the curves were obtained from the present numerical procedure.

Reference:

1. Newman, P. A., and Anderson, E. C. "Analytical Design of a Contoured Wind-Tunnel Liner for Supercritical Testing", NASA Conference Publication 2045, Advanced Technology Airfoil Research, Vol. 1, pp. 499-509, 1979.

---

*STAD, 534-01-13, 804-827-2627

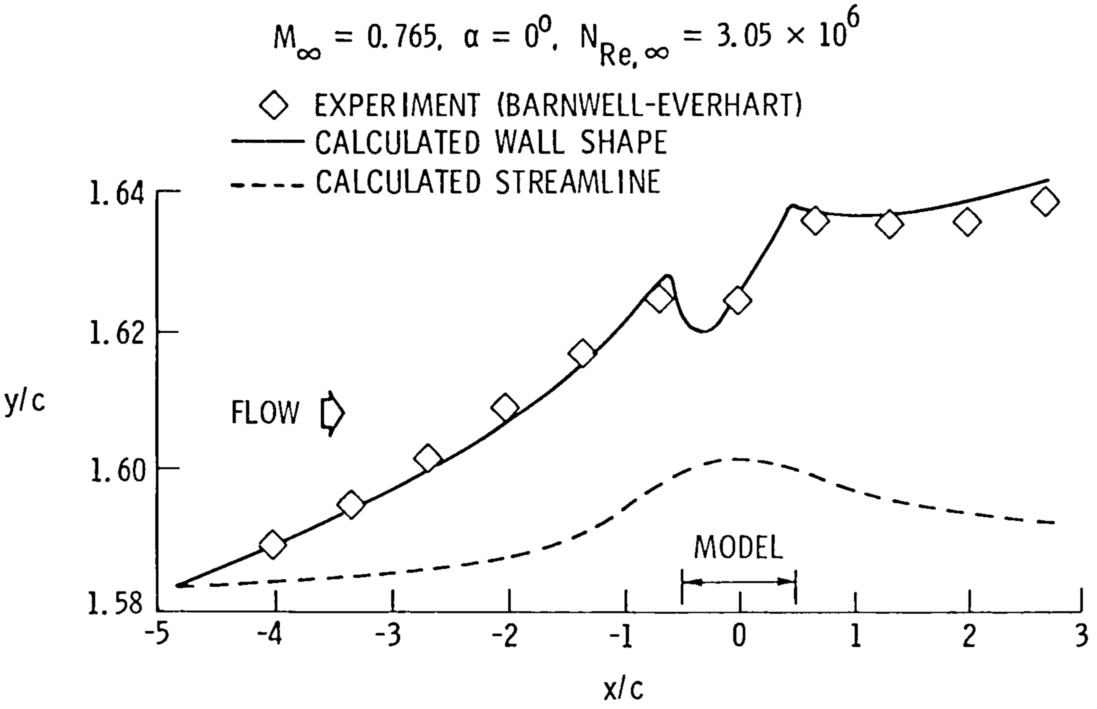


Figure - Comparison of flexible nonporous wall shapes for an NACA 0012 airfoil test at zero lift in the Langley 6- by 19-inch transonic tunnel.

DEVELOPMENT OF A COUPLED FINITE-VOLUME/PANEL CODE FOR ANALYSIS OF LFC SWEPT WING TESTS

Perry A. Newman*

Design of the wind-tunnel liner for the LFC supercritical wing experiment in the 8-Foot transonic tunnel (ref. 1) is for a single-test point condition. In order to assess and/or correct for the influence of the liner at off-design transonic-flow conditions, a 3-D analysis code is required. This WINGTEST code is being developed by Flow Research Company under contract NAS1-16170.

The test configuration to be modeled by the WINGTEST code is a constant-chord, variable airfoil section wing which spans a contoured-wall, transonic wind tunnel at an arbitrary yaw angle. This will be accomplished by modifying the newly developed FLO 29 full potential code to treat the test section, adapting a 3-D panel method code to treat the contraction section, and developing an iterative procedure to couple the codes to obtain an interactive solution. This is a first-cut at arbitrary 3-D wind-tunnel simulation; if successful, many generalizations will open for investigation.

Reference:

1. See preceding compendium summary entitled "Method for Numerical Design of a Contoured Wind-Tunnel Liner for Test of a Laminar-Flow-Control System for a Yawed Supercritical Airfoil Model."

---

*STAD, 534-01-13, 804-827-2627

MULTIPLE LEVEL TECHNIQUES FOR COMPUTATIONAL FLUID DYNAMICS

D. R. McCarthy* and R. C. Swanson**

In recent times significant progress has been made in the application of the multi-grid technique in the solution of elliptic partial differential equations. This method attempts to eliminate relaxation stall by solving an equation on a sequence of course and fine grids. It also contains the structure necessary to relate selective local mesh refinements to the relaxation process on coarser global grids. In the present work the objective is to demonstrate the effectiveness of the multi-grid process in transonic flow analysis.

The Multiple Level Adaptive Technique (MLAT) or multi-grid procedure has been incorporated into the serial 3-D transonic potential flow code of Reyhner (Ref. 1). This modified program has been applied to the solution of the flow about the axisymmetric NASA TM X-2937 inlet. Through this application the many necessary criteria for operation of MLAT are being investigated. The code provides four levels of mesh, the finest of which is 161 (axial) by 81 (radial) by 5 (angular). After about one hour of CPU time, encompassing work equivalent to about 475 sweeps of the field, the unmodified program still retains residual (average change in solution at each field point) on the order of 10^{-5}, and is effectively stalled. A recent MLAT run achieved convergence to within a residual of 10^{-7}, with relaxation work equivalent to 40 sweeps. This is a substantial performance gain for the code. Additional cases with MLAT are now being performed. Also, the MLAT is being programmed into the recently completed vector 3-D transonic potential flow code of Reyhner.

Reference:
Reyhner, T. G., "Transonic Potential Flow Around Axisymmetric Inlets and Bodies at Angle of Attack," AIAA Paper 77-145, January 1977.

---

*Indiana/Purdue University
**HSAD, 505-31-43, 804-827-2673

ALGORITHM FOR SOLVING THE FULL TRANSONIC POTENTIAL FLOW EQUATIONS FOR A NACELLE/WING/PYLON GEOMETRY USING THE CDC CYBER-203 COMPUTER

T. A. Reyhner* and D. E. Reubush**

During the past decade there has been tremendous progress in the field of transonic potential-flow computation. This work which started with small disturbance theory and simple two-dimensional configurations has been extended to the full potential equation and several classes of three-dimensional geometries. Most extensions to three-dimensions have been based on some mapping from the physical coordinates to a more convenient computational coordinate system. If such mappings can be found, the computational problem exclusive of the generation of the mapping, can be greatly simplified. The difficulty with this approach is that as the geometry of objects to be considered becomes more complex, it becomes more difficult to find convenient mappings that will provide reasonable coordinates for the computation. A further problem is that with a complex mapping it can be very difficult to have any physical understanding of the solution process.

A project has been in progress for several years to obtain solutions of the transonic potential flow equations in simple physical coordinates. This type of solution method does not require the finding of a new mapping for each new family of geometries. It also has the advantage that the entire solution procedure is in terms of physical coordinates and variables, thus any failures are much easier to interpret. The first work on this concept was for axisymmetric flow (1). Later the initial work was extended to axisymmetric geometries at angle of attack (2). The current work has extended the concept to general three-dimensional inlet, body and duct geometries and has vectorized the code for use on the CDC Cyber 200 series of computers (STAR). Future work will incorporate wing and pylon geometries in addition to the nacelle.

The work has been primarily done by T. A. Reyhner and associates of the Boeing Commercial Airplane Company with computer time and consultation provided by NASA-Langley.

A comparison between theory and experiment has been made for an asymmetric inlet tested in the NASA-Ames 40-by 80-foot tunnel. The model had a fan face diameter of 1.397 meters and was used with a high-bypass turbofan engine. Figure 1 presents a graphical display of the inlet and spinner geometry. The geometry description used in the analysis is smooth and the entire inlet was specified. Only half of the inlet is shown in figure 1 and curves are represented using straight line segments due to limitations of the plotting equipment. The comparison between theory and experiment for a run at 60^o angle of attack and 38.9 m/s forward velocity is shown in figure 2. Agreement between theory and experiment is excellent.

References:

1. Reyhner, T. A., "Axisymmetric Transonic Potential Flow Around Inlets," AIAA Journal, Vol. 15, May 1977, pp. 624-631.
2. Reyhner, T. A., "Transonic Potential Flow Around Axisymmetric Inlets and Bodies at Angle of Attack," AIAA Journal, Vol. 15, September 1977, pp. 1299-1306.

---

*Boeing Commercial Airplane Company

**HSAD, 505-31-43, 804-827-2673

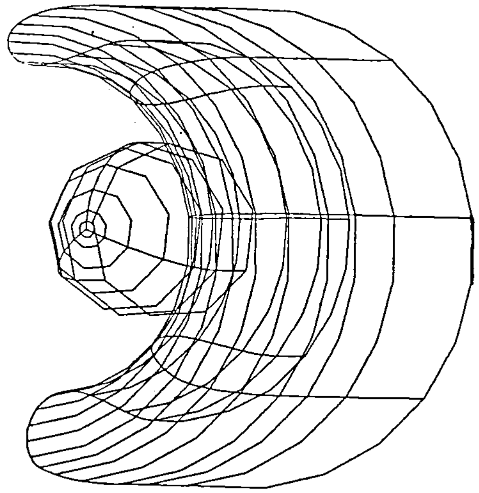


Figure 1.- GRAPHICAL DISPLAY OF INLET GEOMETRY

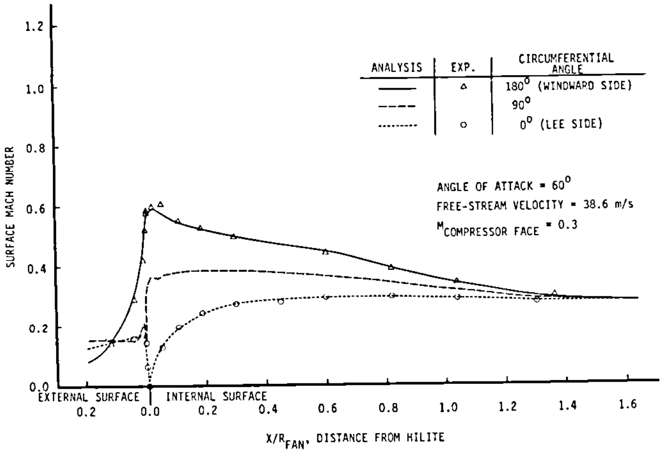


Figure 2.- INLET COWL SURFACE MACH NUMBER DISTRIBUTION

A SMALL DISTURBANCE TECHNIQUE FOR PREDICTING THE EFFECTS OF JET EXHAUST FLOW ON THE CHARACTERISTICS OF TRANSPORT AIRPLANES

Chen Sun* and William K. Abeyounis**

One of the most complex and demanding problems during the design of transport aircraft is the integration of the propulsion system. This problem is particularly difficult at transonic speeds where jet interference and entrainment effects can be significant. Current methods for predicting this complex flow for three-dimensional transport configurations have not been adequate. The purpose of this work is to extend a transonic small disturbance technique (Ref. 1) to more accurately account for jet interference and entrainment effects on transport configurations.

The first step of this small disturbance technique is to "calibrate" the small disturbance code. Pressure distributions on the wing are calculated for a wing-body configuration using an exact transonic code. The wing-body case is then computed using the small disturbance code, modifying the wing geometry until the pressure distributions match those of the exact transonic code. The small-disturbance approximation is compensated for by tailoring the leading edge slopes and by adding correction ramps on the aft portion of the wing.

The next step of the small disturbance technique is to make a more detailed nacelle or nacelle-pylon calculation. The present work will upgrade this calculation by incorporating the effects of a powered jet in a quasi-cylindrical fashion. The effects of jet entrainment will also be incorporated using a simple, isobaric mixing model. This calculation provides streamline slopes on a control surface surrounding the nacelle.

The final step of the small disturbance technique is to incorporate the effects of the powered jet on the total configuration. This is accomplished by again computing the wing-body case using the small disturbance code. In this wing-body calculation the nacelle effects are included by imposing the streamline slopes from the detailed nacelle calculation on the control surface surrounding the nacelle.

Reference:
Gould, D.; Sun, C.; and Yoshihara, H: AFTI-III - Experiments and Calculations at Transonic Conditions: Symposium on Transonic Aircraft Technology, Aug. 15-17, 1978. (Lancaster, Calif.)

---

*The Boeing Company
**HSAD, 505-31-43, 804-827-2675

Unsteady Flow

Time-accurate methods designed primarily for the unsteady problem.

DEVELOPMENT OF FREQUENCY PLANE PERTURBATION METHOD FOR TRANSONIC UNSTEADY FLOWS

Robert M. Bennett*

One of the important and currently unsolved problems in aeronautics is accurate calculation of the reduction or dip in flutter speed that occurs at transonic Mach numbers. As one approach to addressing this problem, the Boeing Airplane Company has been working for several years under contract to the Langley Research Center to develop a transonic perturbation method for calculating oscillatory aerodynamics for use in conventional flutter analysis and design programs. The incremental forces for an oscillating wing or airfoil are calculated by a finite difference method as a perturbation from a calculated nonlinear flow field containing shock waves and other transonic effects. In the past, this method has failed to give meaningful results except at very low frequencies of oscillation. These frequency limits were a result of the numerical solution procedure and precluded results for normal flutter frequencies of oscillation. Recently the method was modified (under contract NAS1-15128) and good results have been obtained at frequencies of up to ten times the previously encountered limits and including the flutter range. The modifications include: (1) important revisions to the finite difference grid network, (2) a specially-developed, out-of-core, sparse matrix solver, and (3) revised treatment of boundary conditions.

An oscillating flat plate is used as a test case for the finite difference method as exact solutions are available. A sample result for the flat plate oscillating at a reduced frequency, $k = \omega c/2v$, of 0.6 is shown in the figure. The real (or in-phase) and imaginary (or out-of-phase) components of lifting pressure are shown plotted versus the airfoil chord. There is good agreement between the numerical solution (symbols) and the exact solution (lines). Previously, this example could not be calculated satisfactorily as the reduced frequency of 0.6 is about ten times larger than the previous frequency limit. Although cases with supercritical flows at subsonic speeds have not yet been treated with the revised method, examples for a circular arc airfoil at low supersonic Mach numbers with a detached bow shock wave have been satisfactorily calculated. The work is continuing in an effort to exploit this development by extending the method to treat three dimensional wings with thickness and by applying it to the calculation of transonic flutter boundaries for two and three dimensional wings.

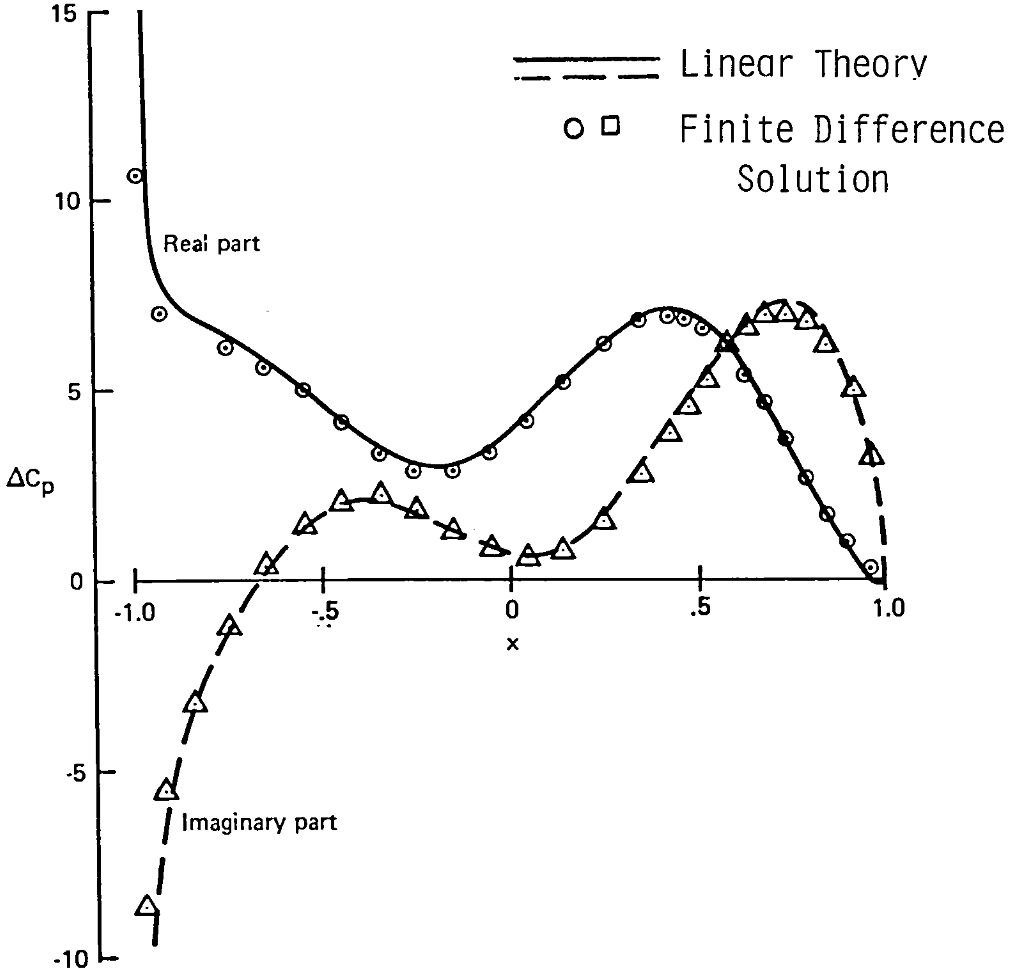
---

*Aeroelasticity Branch, 505-02-23, 804-827-2661

FINITE DIFFERENCE SOLUTION FOR FLAT PLATE OSCILLATING IN PITCH

$M = 0.9 \qquad k = 0.6$

Frequency Plane Perturbation Method (Boeing)



DEVELOPMENT OF A FINITE ELEMENT METHOD FOR TRANSONIC UNSTEADY POTENTIAL FLOW

Kenneth R. Kimble*

The finite element method, developed originally for structural problems has many features which recommend its use for computing fluid flow for geometrically complex configurations. As illustrated in figure 1, the finite element mesh can be tailored to resolve flow singularities easily for simple problems. The generation of a corresponding grid for a complete aircraft configuration is relatively simple since the topology is almost unlimited in contrast to the rectangle-like mesh requirements of finite difference methods.

In order to utilize finite elements in transonic flow an analog of upstream and mixed differencing had to be developed. This was done recently, and a comparison case is illustrated in figure 2 where the TSFOIL solution due to Murman is compared to our mixed flow finite element code.

The unsteady transonic flow equation requires that the mesh be fine enough to resolve at least one wavelength in order to obtain accurate solutions. In order to perform the solution of the resulting equations (for which current iterative methods have been unsuccessful) an out-of-core equation solver (called a frontal elimination algorithm) is being adapted for use with the transonic finite element code. An example of an unsteady flow calculation for an airfoil with oscillating flap is given in figure 3.

Current work is proceeding on the solution of the full potential inviscid steady and unsteady transonic equations using a finite element analog of the artificial compressibility method (ref. 1).

References:
1. Hafez, M. M., Murman, E. M., and South, J. C., "Artificial Compressibility Methods for Numerical Solution of Transonic Full Potential Equation," Paper 78-1148, AIAA 11th Fluid and Plasma Dynamics Conference, Seattle, Wash., July 10-12, 1978.

---

*The University of Tennessee Space Institute, Tullahoma, TN 37388, 615-455-0631 work supported by NASA Grant NSG 1224.

Fig. 1 Mesh for airfoil between $-1 \leq x \leq 1$

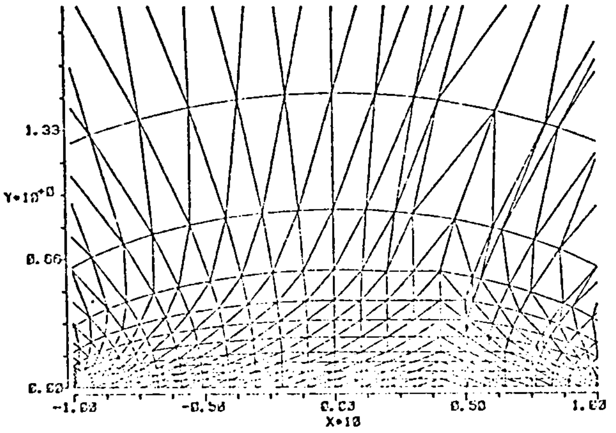
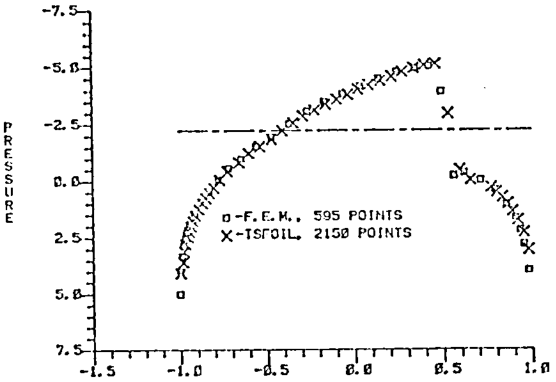

1.33
Y*10**0
0.66
0.00
-1.00
-0.50
0.00
0.50
1.00
X*10

Fig. 2 Steady flow at $M_\infty = .85$ on 6% parabolic, airfoil



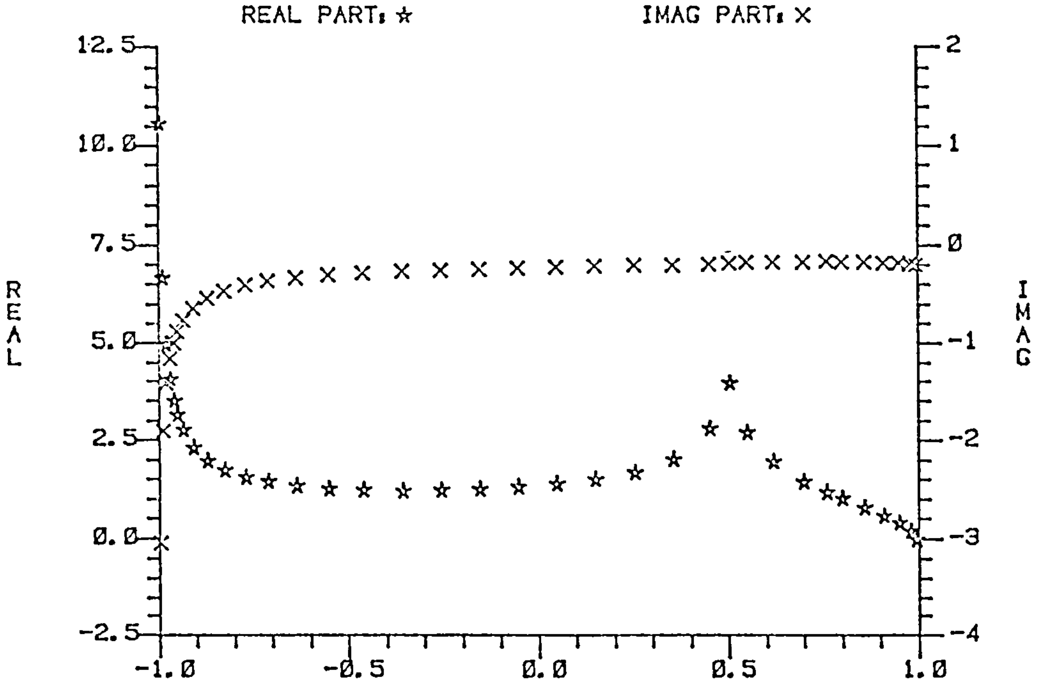


Fig. 3 Unsteady flow for oscillating flap on flat plate ($M_\infty = .8$, k = .6

AN EVALUATION OF TWO-DIMENSIONAL SHOCK COMPATIBILITY CONDITIONS FOR LINEAR UNSTEADY TRANSONIC FLOW PERTURBATIONS

Woodrow Whitlow, Jr.*

An important assumption in treating unsteady transonic flow problems as linear perturbations about a nonlinear steady state is that the aerodynamic loads vary linearly with the amplitude of the unsteady motion. Inherent in that assumption is the requirement that the shock excursion amplitude, an example of which is shown in Figure 1, remain sufficiently small such that aerodynamic loads show the necessary behavior with motion amplitude. Various factors, such as airfoil geometry and the amplitude and frequency of its motions, affect the unsteady shock movement. However, the flow conditions across shock waves and their movements may be determined by imposing a compatibility condition at the steady state shock locations.

The early perturbation method of Weatherill et al.[1] neglected the unsteady shock conditions but is now being modified to include a normal shock condition. Cunningham[2] has also derived a normal shock compatibility condition, and Williams[3] has presented a condition that includes the effects of local shock inclination. Part of the current effort is also aimed at determining an alternate shock condition that includes the effects of local shock slope.

In this study, to be conducted at Langley Research Center, the various compatibility conditions are to be incorporated into finite difference solutions of the perturbation equations. Where possible, comparisons with experimental data will be made. It is hoped that the results will determine the importance of including the effects of shock inclination in the compatibility conditions. Also, the aerodynamic loads will be monitored for various types of unsteady motions, reduced frequencies, and amplitudes to determine the conditions for which the assumption of linear unsteady perturbations is no longer valid.

References:
1. Weatherill, W. H.; Sebastian, J. D.; and Ehlers, F. E.: The Practical Application of a Finite Difference Method for Analyzing Transonic Flow over Oscillating Airfoils and Wings. NASA CR-2933, 1978.

2. Cunningham, A. M., Jr.: A Steady and Oscillatory Kernel Function Method for Interfering Surfaces in Subsonic, Transonic, and Supersonic Flow. NASA CR-144895, 1976.

3. Williams, M. H.: Linearization of Unsteady Transonic Flows Containing Shocks. AIAA Journal, vol. 17, pp. 394-397, 1979.

---

*SDD, 505-02-23, 804-827-2661

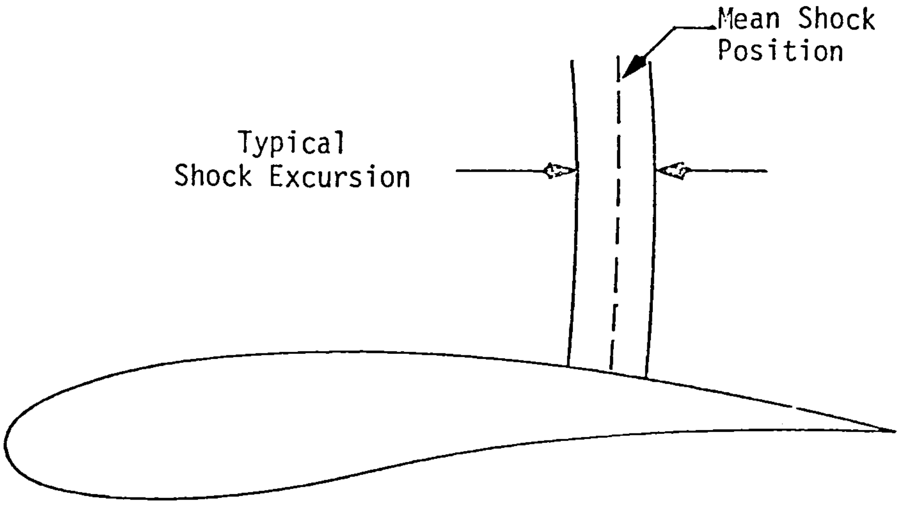

Mean Shock Position
Typical Shock Excursion

Figure 1. Typical unsteady shock motion.

A FINITE ELEMENT METHOD FOR THE PERIODICALLY OSCILLATING THIN AIRFOIL

G. Fix, M. D. Gunzburger, R. A. Nicolaides and C. Cox*

This investigation deals with the application of a least squares finite element method to the oscillating thin airfoil problem. The governing equation is the (small disturbance) unsteady potential transonic flow equation. The dependence on time is removed by assuming periodic solutions with frequency ω, which is the frequency of oscillation of the airfoil, and the equations are linearized by assuming that the oscillations are small. This equation is recast into a first order system by introducing the velocity vector $\underline{u} = (u,v)$ which is, of course, $\nabla\phi$, ϕ being the potential. All boundary conditions are also cast in terms of the variables $\underline{u}$, $\underline{v}$, and ϕ. In particular, on the wing we specify v and across the wake behind the trailing edge we require that v and the pressure, i.e. $(u + i\omega\phi)$ be continuous. The domain in which our governing equations are valid is exterior to the airfoil. In our computational scheme we must, of course, choose a finite computational region. On the outer boundary of our computational region, we replace the Sommerfeld radiation conditions by high order approximations to the condition [1]. Finally, we also insure that the Kutta condition at the trailing edge is satisfied. The finite element method employed consists of minimizing a suitable quadratic functional $J(u,v,\phi)$ over a finite element space, which we choose to be a space of piecewise linear functions on selected grids. With proper choices of grids (see [2]), the method is second order accurate. Due to either the first order formulation or the use of a least squares principle, the method has the following advantages:

1. The associated matrix problem is Hermitian and positive definite regardless of the size of ω or whether the basic flow is or is not transonic.

2. The achievement of second order accuracy is also independent of the above considerations.

A computer code implementing the method has been developed and is presently in the final stages of testing.

References:
1. Bayliss, Gunzburger, Turkel, ICASE Report 80-1.

2. Fix, Gunzburger, Nicolaides "Finite Element Methods of the Least Squares Type" Comp. and Math. with Appl., 5, pp 87-98.

---

*ICASE, 505-31-83-01, 804-827-2513

IMPROVED SONIC-BOX COMPUTER PROGRAMS FOR CALCULATING TRANSONIC AERODYNAMIC LOADS ON OSCILLATING WINGS WITH THICKNESS

Song Y. Ruo*
E. Carson Yates, Jr.**
Jerome G. Theisen*

The potential equation for steady transonic flow is essentially nonlinear. In contrast, the corresponding small-perturbation equation for unsteady flow can be reduced to a linear equation with constant coefficients if the frequency of oscillation is high enough. Unfortunately, the linearization requires frequencies above the range of usual interest for lifting-surface flutter. Moreover, an important consequence of the linearization is the suppression of deviations are important in the propagation of disturbances over the lifting surface, improvement in the theory may be accomplished by retaining at least the effect of finite wing thickness on variations in mean local Mach number. This is achieved by treating steady-flow properties as local parameters in the unsteady flow. This assumption, equivalent to local linearization, permits the nonlinear differential equation for the velocity potential to be reduced to a linear equation with variable coefficients containing the local Mach number. After a nonuniform coordinate transformation, the equation becomes identical to the linearized transonic unsteady-flow equation with constant coefficients. Hence the problem can be solved in the transformed space by any method that is suitable for the linear equation, such as the sonic kernal function of the sonic-box method.

For exploratory use in flutter analysis, the present method has been implemented as a modification to the sonic-box program. To illustrate its application, the accompanying figure shows the detrimental effect on flutter speed of increasing the thickness of a delta wing with elliptical cross section in the lateral direction. Increasing thickness from 0 to 4 percent reduces flutter speed by 15 percent.

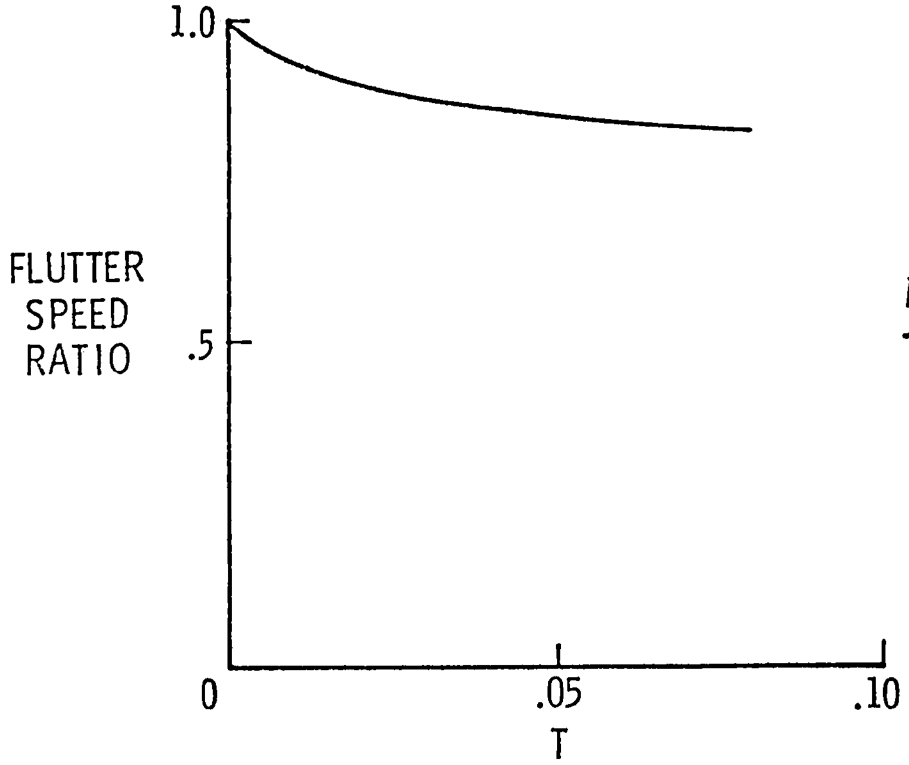
An alternate modification of the sonic-box program is also available. For this modification, the coordinate transformation is not used. Instead, the velocity potential is formulated directly in terms of local Mach number.

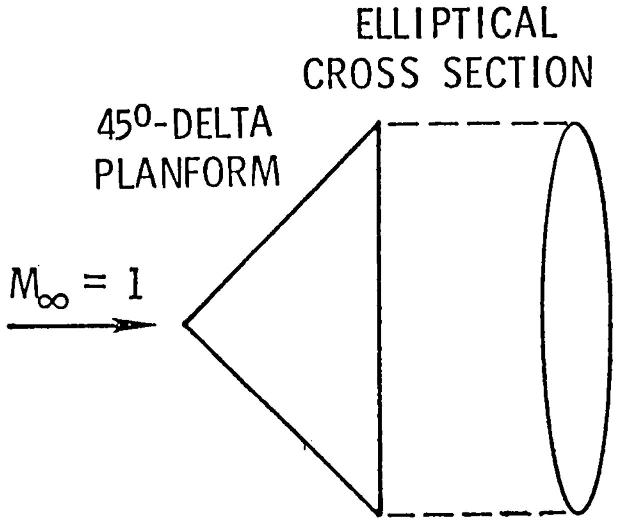
---

*Lockheed-Georgia Company

**Aeroelasticity Branch, 505-33-53, 804-827-2661

EFFECT OF THICKNESS ON TRANSONIC FLUTTER SPEED




ELLIPTICAL
CROSS SECTION
45^0-DELTA
PLANFORM
$M_\infty = 1$

5. Inviscid Euler Equations

Methods for solving the equations of inviscid, rotational flow in primitive-variable (p, ρ, u, v, w) form. Validation of codes included.

UPSTREAM DIFFERENCING FOR COMPRESSIBLE FLOW

Bram van Leer*

The one-dimensional Euler equations of compressible flow are usually written in conservation form when shock-wave propagation is considered and in diagonal or characteristic form when sound-wave propagation is considered. The physical content of the characteristic equations is best preserved if upstream differencing is used. Shock waves are properly accounted for if the scheme is cast in the conservation form. Van Leer has shown how to combine upstream differencing and conservative differencing in explicit nodal-point schemes [1] and control-volume schemes [2], [3].

In the latter scheme the characteristic equations are used to evaluate the fluxes of conserved quantities at cell interfaces while the conservation laws are used to update the cell integrals of conserved quantities. In each mesh a set of monotone basis functions is used to approximate the initial-value distributions, thus eliminating numerical oscillations and providing sub-grid resolution.

The present work aims at the following extensions of these schemes:

a) Switching to an implicit mode wherever the local Courant number exceeds unity (see Fig. 1).

b) Using discontinuous basis functions for achieving the equivalent of shock fitting (see Fig. 2).

c) Developing multi-dimensional forms, factorized or not, that account for the underlying bi-characteristic equations.

d) Deriving efficient nodal-point versions, using the technique of flux-vector splitting developed by Steger, Beam and Warming [4].

References:

1. van Leer, B. "Towards the Ultimate Conservative Difference Scheme III. Upstream Centered Finite-Difference Schemes for Ideal Compressible Flow," J. Computational Phys. 23 (1977), p. 263.

2. Ibidem, "IV. A New Approach to Numerical Convection," J. Computational Phys. 23 (1979), p. 276.

3. Ibidem, "V. A Second-Order Sequel to Godunov's Method," J. Computational Phys. 32 (1979), p. 101.

4. Steger, J. S. and Warming, R. F., "Flux Vector Splitting of the Inviscid Gasdynamic Equations with Application to Finite Difference Methods," NASA Technical Memorandum 78605.

---

*ICASE, 505-31-83-01, 804-827-2513

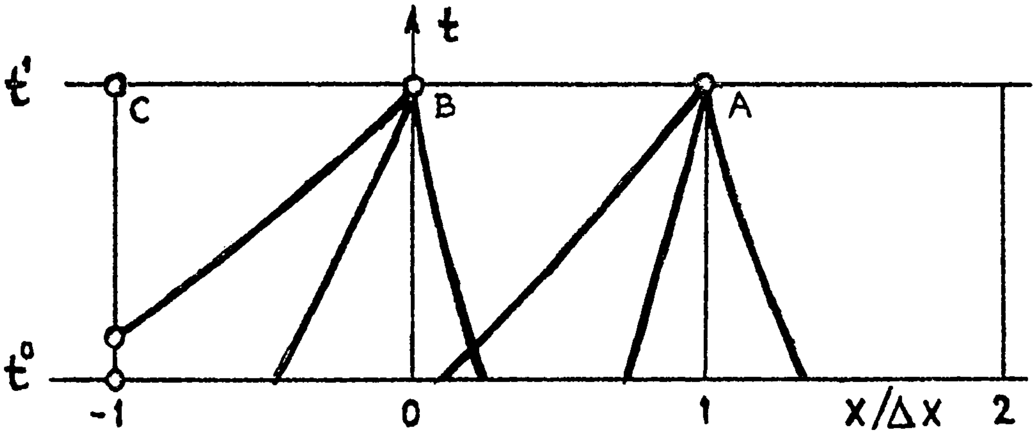


Figure 1: Time-space diagram for control-volume scheme. Characteristics are drawn backward in time from cell boundaries. The domain of dependence of point A at t^0 is included in the adjacent meshes. For B the domain extends beyond the left-hand mesh (Courant number > 1); boundary data at $x/\Delta x = -1$, $t > t^0$ may be introduced. This causes a dependence on C and makes the scheme implicit.

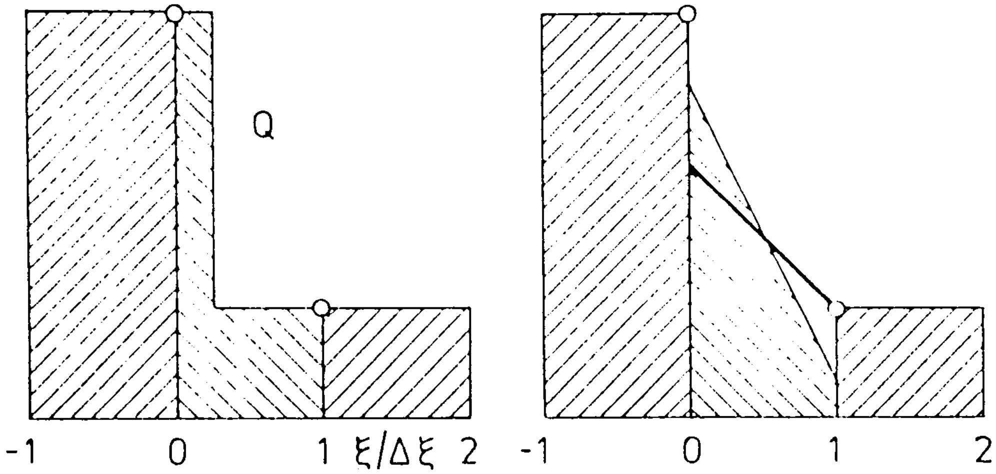


Figure 2: Initial-value representation. Left: Q is discontinuous in mesh (0,1). Right: approximation by linear basis functions. Thin line: least-squares approximation. Thick line: nearest monotone representation with same mesh integral.

If discontinuous basis functions were included, the original initial-value distribution could be retrieved from mesh integral and mesh-boundary data.

FINITE DIFFERENCE SOLUTION OF A GALACTIC FLOW PROBLEM

B. van Leer*, T. Zang, M. Y. Hussaini*
G. D. van Albada, and W. W. Roberts

Cosmic gas flows are distinguished from terrestrial and laboratory flows in deriving their main features not from shaped, solid boundaries but rather from gravitational forces and radiative processes. A cosmic flow problem, when solved numerically, may bring out lesser known properties of the finite-difference scheme employed. One simple, yet representative, cosmic flow problem has been selected for testing a number of schemes currently used in aerodynamics and astrophysics.

The problem considered models, in one dimension, the periodic flow of isothermal galactic gas across the spiral arms of a spiral galaxy, driven by the gravitational field of the underlying mass distribution in the system [1]. The rise and fall of the gravitational potential has the same effect as the widening and narrowing of the channel in channel flow. Flow-dependent Coriolis forces add to the complexity of the flow.

The problem has a stationary solution that may or may not include a shock, depending on the amplitude of the potential oscillation. The following schemes have been or will be tested on their ability to approximate this stationary solution.

- The "beam scheme" [2] used only in astrophysics
- Godunov's method [3] in several forms
- Lax-Wendroff scheme with higher-order filter
- MacCormack scheme with higher-order filter (see Fig. 1)
- Various forms of SHASTA-based Flux Corrected Transport [4], no convergence to asymptotic solution
- MUSCL (monotone higher-order upsteam-biased differencing)[5]
- Pseudo-spectral method (see T. Zang and M. Y. Hussaini, "Mixed Spectral Finite Difference Methods for Compressible Navier-Stokes Equations," in this Compendium)

References:
1. Woodward, P. R. Astrophys. J. 195 (1975), 61.
2. Sanders, P. H. and Prendergast, K. H. Astrophys. J. 188 (1974), 489.
3. Godunov, S. K., Zabrodyn, A. W. and Prokopov, G. P. Z. Vycisl. Mat. i Mat. Fiz. 1 (1961), 1020.
4. Book, D. L., Boris, J. P., and Hain, K. J. Computational Phys. 18 (1975), 248.
5. van Leer, B. J. Computational Phys. 32 (1979), 101.

---

*ICASE, 505-31-83--01, 804-827-2513

COMPUTATION OF THE SHAPE OF SLENDER STREAMS OF FLUID

James F. Geer and John C. Strikwerda*

The shape of a slender jet of fluid falling steadily under the force of gravity is studied. The problem is formulated as a nonlinear free boundary value problem for the potential. The use of perturbation expansions results in a system of equations that can be solved efficiently by integrating them in the stream-wise direction. Computations have been made for jets issuing from orifices of various shapes including an ellipse, a rectangle, and an equilateral triangle. Figures 1 and 2 show the cross-sectional shapes for vertically falling jets issuing from orifices in the shape of a triangle and a rectangle. In each case the jet forms sheets of fluid which propagate outward in directions perpendicular to the sides of the original shape. Surface tension effects have been ignored in these examples.

Work has been completed for vertically falling jets without surface tension (Ref. 1 and 2). Current research is on nonvertical jets and jets with surface tension.

The method is stable and second-order accurate (Ref. 3). This method appears to be new and may be applicable to other nonlinear free boundary value problems.

References:
1. Geer, J. F. and Strikwerda, J. C. "Vertical Slender Jets" Journal of Fluid Mechanics, to appear.

2. Geer, J. F. and Strikwerda, J. C. "Vertical Slender Jets" ICASE Report 79-23.

3. Strikwerda, J. C. and Geer, J. F. "A Numerical Method for Computing the Shape of a Vertical Slender Jet" ICASE Report 80-7.

---

*ICASE, 505-31-83-01, 804-827-2513

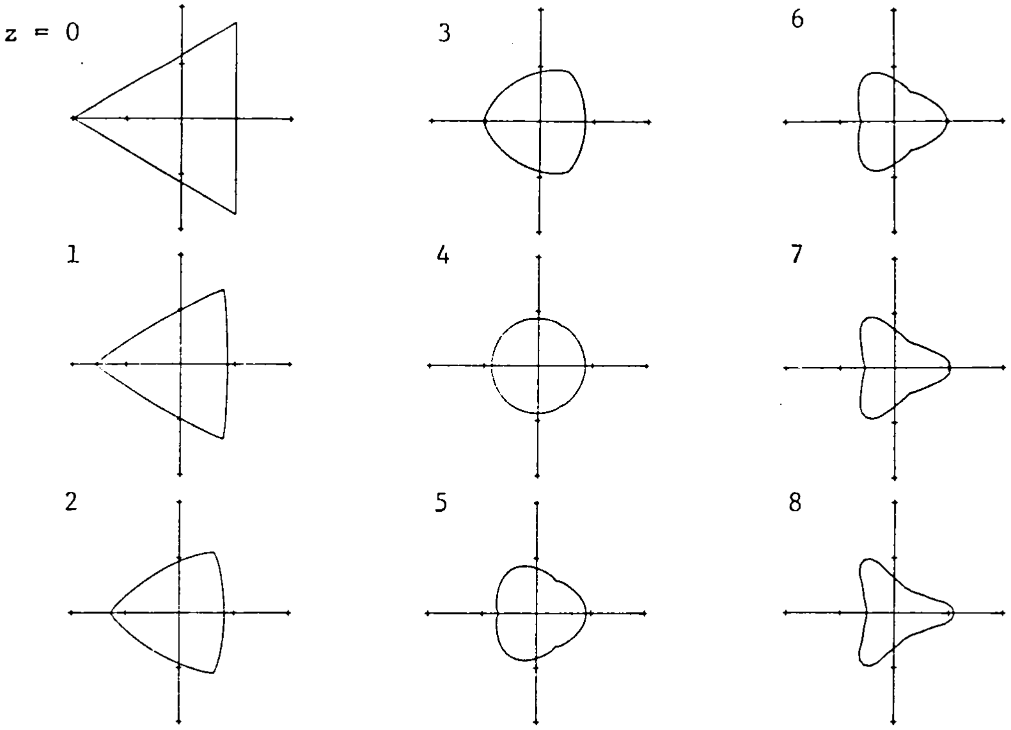


Figure 1.- Cross-sectional shapes at several values of z for a jet with the initial shape of an equilateral triangle, with side of length 3.

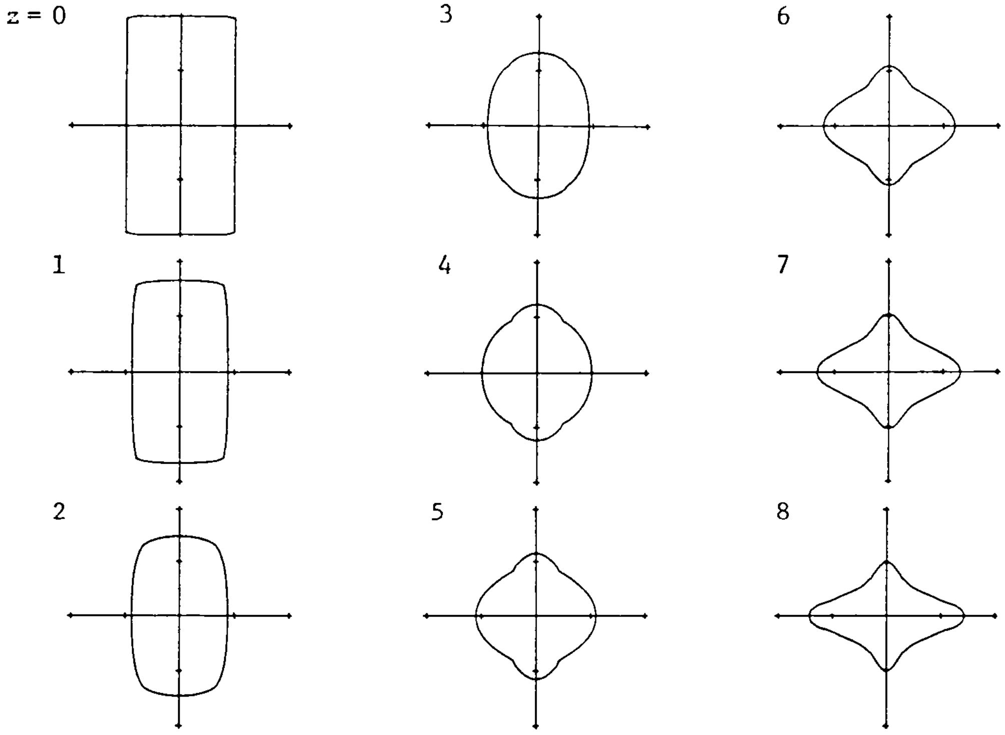


Figure 2.- Cross-sectional shapes for a jet with the initial shape of a rectangle with sides of length 2 and 4.

STUDY OF TRANSONIC FLOW OVER A CIRCULAR CYLINDER USING THE EULER EQUATIONS

Manual D. Salas*

During the last decade, the transonic flow over two-dimensional airfoils has been investigated in great detail through numerical solutions of the full-potential equation. It is well known that the accuracy of these solutions can deteriorate if the flows are strongly rotational. The purpose of this work is to develop benchmark solutions using the Euler equations in order to better assess the importance of rotational effects and the range of validity of the potential solutions. As a preliminary effort on the development of a code for general two-dimensional airfoils, a study is made of the flow over a circular cylinder.

To eliminate ambiguities in imposing the free-stream boundary conditions, the infinite domain outside the unit circle is mapped into the finite domain inside the unit circle by means of an inverse radial transformation. The time-dependent Euler equations are written in their λ-form (ref. 1). The Euler equations are then integrated in time using the Gabutti finite difference scheme until a steady state is reached. The Gabutti scheme has the advantage of being stable up to Courant number 2.

A comparison of the surface Mach number calculated using the Euler equations and the potential equation (in nonconservative) form is shown in figure 1 for a free-stream Mach number of 0.45. There is good agreement between the two solutions in the forward section of the circle, with some disagreement in the rear of the circle. At this point in the study, it is premature to account for the discrepancies. The Euler solution shows a strong isentropic recompressing ahead of the shock which is being treated as a fitted discontinuity in the numerical analysis.

The isobars for the Euler solution for the same case are shown in figure 2. The pressure levels shown are nondimensionalized with respect to the free-stream pressure.

References:
1. Moretti, G. "The λ Scheme" Computers and Fluids, Vol. 7, pp 191-205, 1979.

---

*STAD, 505-31-13, 804-827-2627

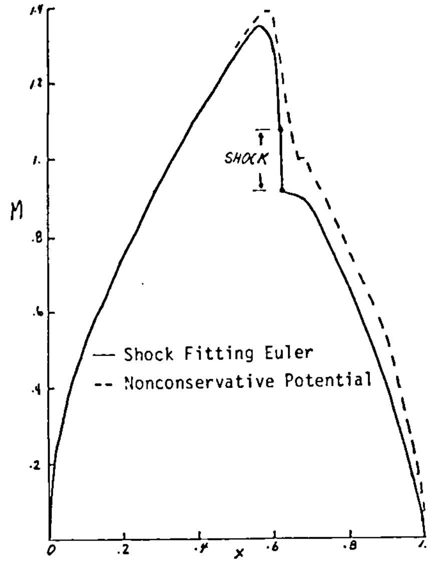


Figure 1. Calculated surface Mach number for a circle at $M_\infty = 0.45$

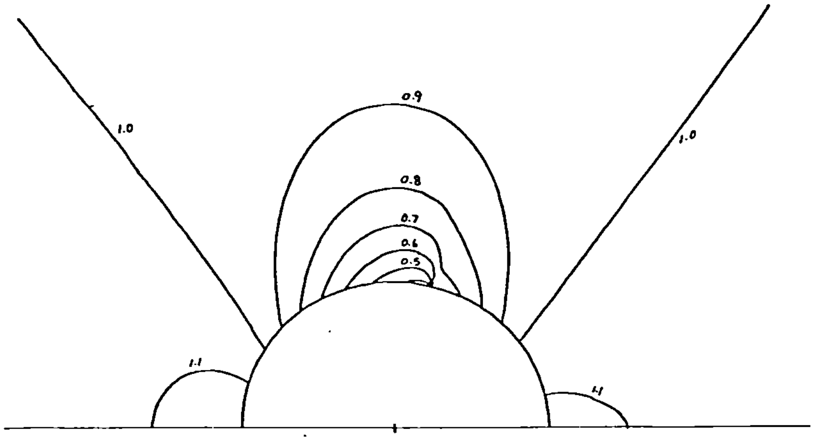


Figure 2. Isobars for the Euler solution of a circle at $M_\infty = 0.45$

STUDY OF FLOWFIELDS ABOUT ASYMMETRIC EXTERNAL CORNERS

Manuel D. Salas*

A numerical study of flowfields about asymmetrical conical corners formed by the juncture of swept compressive wedges, which are typical of delta wings and rectangular inlets, see figure 1, is being conducted.

The region of interest is the region bounded by the outer bow shock, the two crossflow sonic lines, and the surface of the body. This region is mapped into a rectangular computational plane where grid points are evenly distributed. The resulting mesh in the physical plane is shown on figure 2.

In this region the time-dependent conical Euler equations are integrated in time using MacCormack's finite difference scheme, with the bow shock fitted as a discontinuity, until a steady state is reached.

The main thrust of the study is an investigation of how the crossflow transitions from a symmetrical configuration, where the corner is a cross-flow stagnation point, to an asymmetric configuration, where the cross flow spills over the corner as shown in figure 3.

The details of the investigation will appear in reference 1.

References:

1. Salas, M. D. "A Careful Numerical Study of Flowfields About Asymmetric External Conical Corners" AIAA Paper No. 80-1329, AIAA 13th Fluid and Plasma Dynamics Conference, July 1980.

---

[*] STAD, 505-31-13, 804-827-2627

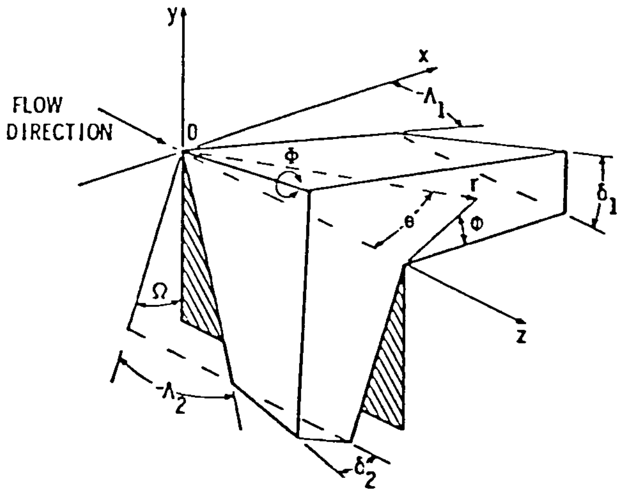

y
x
FLOW
DIRECTION
0
$-\Lambda_1$
Φ
r
θ
Φ
δ_1
Ω
z
$-\Lambda_2$
δ_2

Figure 1.- Corner configuration.

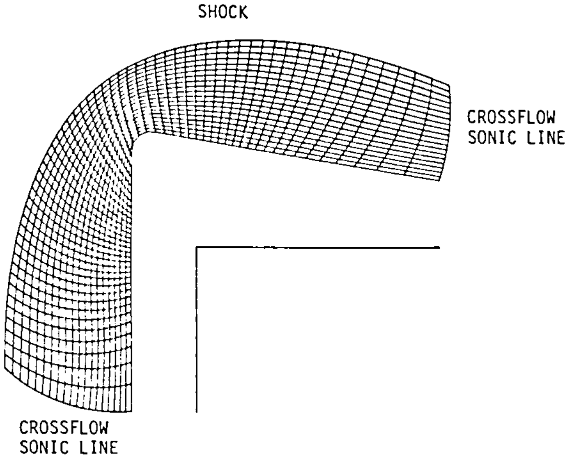

SHOCK
CROSSFLOW
SONIC LINE
CROSSFLOW
SONIC LINE

Figure 2.- Mesh-point distribution on the physical plane.

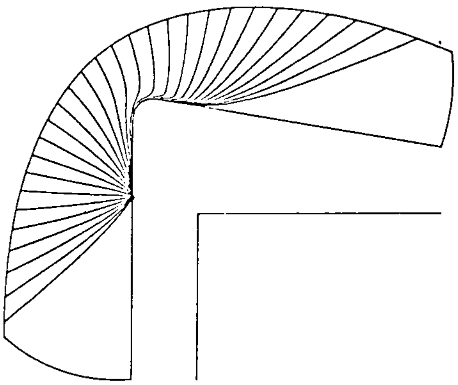


Figure 3.- Computed crossflow streamlines showing flow spilling over the corner.

INTERACTION OF A TWO-DIMENSIONAL SHOCK WAVE WITH A VORTEX

Manuel D. Salas* and Sui-Kwong P. Pao**

This investigation considers the time-dependent interaction of a vortex field and a shock wave inside a shock tube. The numerical investigation attempts to duplicate the experimental study reported in reference 1. The primary interest in this phenomenon concerns the noise generated by the shock-vortex interaction. The phenomenon can also be important in learning how to handle numerically the shock-vortex interaction that occurs in high-speed fighter wings.

In the numerical study, the two-dimensional Euler equations are integrated in time using the MacCormack finite difference scheme to simulate the transient motion of a shock wave over a given vortex field. The region of interest, see figure 1, is the region bounded by the shock-tube walls, the shock wave on the right, and the given supersonic inflow conditions at $x = 0$ on the left.

As the initially straight shock moves through the vortex field, the shock profile deforms into an S shape. As the interaction continues a kink develops in the shock profile, where a secondary shock wave forms. See figure 2. These numerical results are in agreement with the experimental observations of reference 1.

The appearance of the secondary shock wave complicates the numerical calculation of the shock making the MacCormack scheme inappropriate for this problem. We presently plan to replace the MacCormack scheme with the λ-scheme, see reference 2, to alleviate this problem.

References:
1. Naumann, A.; and Hermanns, E. "On the Interaction Between a Shock Wave and a Vortex Field" AGARD-CP-131.
2. Moretti, G. "The λ Scheme" Computers and Fluids, Vol. 7, pp. 191-205, 1979.

---

*STAD, 505-31-13, 804-827-2627

**ANRD, 505-31-13, 804-827-2645

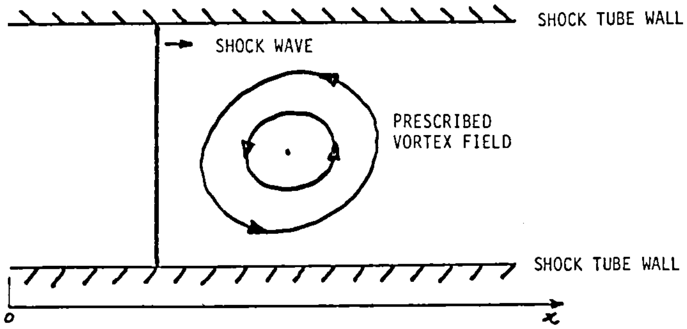

SHOCK TUBE WALL
SHOCK WAVE
PRESCRIBED
VORTEX FIELD
SHOCK TUBE WALL
0
x

Figure 1.- Sketch of configuration being considered.

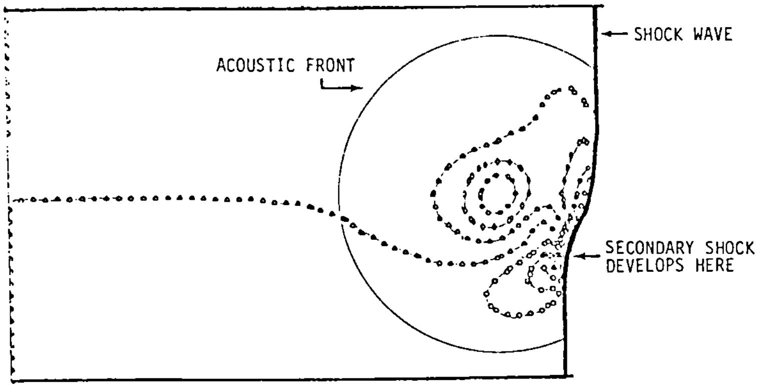


Figure 2.- Isobars for shock-vortex interaction.

CALCULATION OF SUPERSONIC FLOW PAST AIRCRAFT WITH SHOCKS

Manuel D. Salas* and James C. Townsend**

Under a grant to the Polytechnic Institute of New York, a computer code is being developed to solve the three-dimensional, supersonic, steady Euler equations for the flow field about an arrow wing aircraft. The hyperbolic character of the equations allows integration of the equations from one axial station to the next. At each axial station a series of conformal mappings is used to transform the region between the bow shock and the fuselage of the aircraft into a rectangular computational region. The Euler equations are then integrated using the λ-scheme.[1]

A comparison between experimental and calculated surface pressures is shown in figure 1 at two stations. These results are at $M = 2.36$ and zero angle of attack.

Presently, the code is being extended to include the angle-of-attack capability.

References:

1. Moretti, G. "The λ Scheme" Computers and Fluids, Vol. 7, pp. 191-205, 1979.

---

*STAD, 505-31-13, 804-827-2627

**HSAD, 505-31-13, 804-827-3181

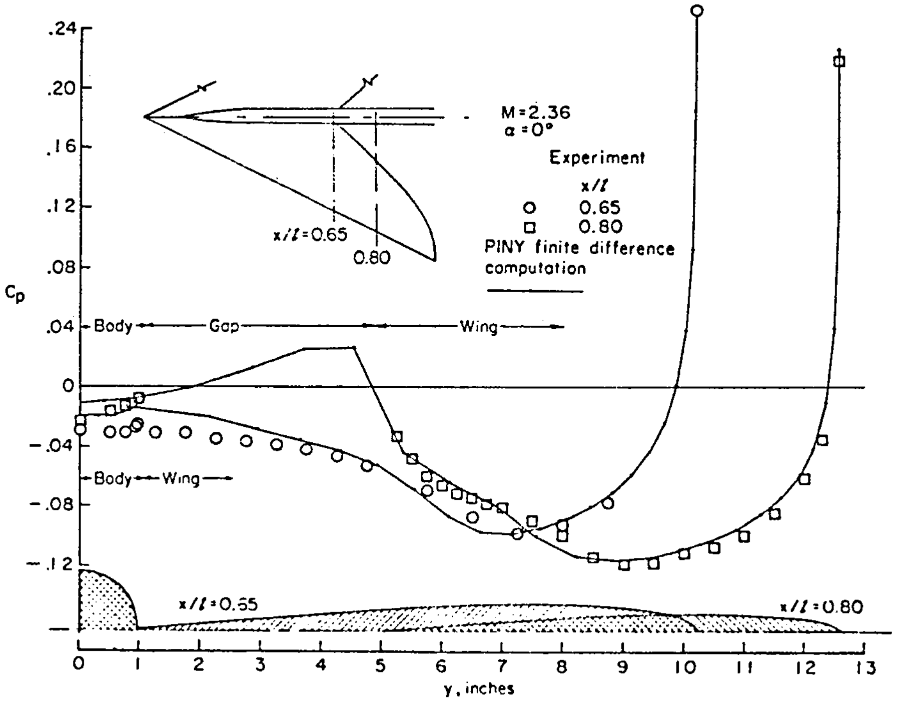


Figure 1.- Comparison between experimental and numerical results.

DEVELOPMENT OF A MULTI-GRID METHOD FOR THE EULER EQUATIONS

Stephen F. Wornom*

Time accurate and relaxation procedures for the compressible inviscid fluid conservation-law-equations (Euler equations) are frequently less efficient than procedures developed for potential flow. This is so because with the Euler equations one has to deal with a system of nonlinear partial differential equations as opposed to a single nonlinear partial differential equation in potential flow. The consequence of a system of equations is that a spectrum of associated eigenvalues must be considered so the equations are usually stiffer. Of more import, with a system of equations, the spectrum of associated eigenvalues can appear (as in subsonic flow) with both positive and negative real parts. This last restriction severely limits the choice of allowable difference operators and iterative or time-dependent procedures.

The purpose of this study is to develop and thoroughly test a point relaxation algorithm for the Euler equations which is made possible by use of flux vector similarity splittings. In this approach, the flux vectors are split such that their associated Jacobian matrices have either all positive or all negative real parts. The split vectors can then be spacially differenced using dissipative forward or backward operators. Simple point iterative methods can thus be used as a solution procedure. A major part of the study will be devoted to improving the iterative convergence rate using the multi-grid relaxation approach.

This research is being conducted by Dr. J. L. Steger of Flow Simulations, Inc., under contract NAS1-14517.

---

*STAD, 505-31-13, 804-827-2627

SOLUTIONS TO THE EULER EQUATIONS USING A STREAMLINE-CHARACTERISTICS COORDINATE SYSTEM

Lawrence Sirovich* and James C. Townsend**

Although current finite-difference methods offer a reasonable approach to computational fluid mechanics for supersonic flows and can produce good results, they have a variety of difficulties which make it worthwhile to investigate new approaches. The present work, which is being done at Brown University under grant NSG 1617, uses flow field physics, analytical methods and known approximations in developing a rapid and accurate numerical code. The method has been applied to unsteady one-dimensional flows and to steady two-dimensional flows with attached shockwaves.

Figure 1 shows such a flow over a two-dimensional airfoil. The two-dimensional, inviscid steady flow equations are written in characteristic form with entropy, flow angle and Mach angle as independent variables in terms of coordinates along the streamlines and principle (C^+) characteristics. An approximate solution, found by assuming that the flow angle Θ is constant along the C^+ characteristics, is used to map the entire upper half plane into a finite region in the computational plane (fig. 2). The body surface and the streamlines become straight horizontal lines in the mapped plane, and the C^+ characteristics become vertical lines. The curved lines in figure 2 are the leading-edge and trailing-edge shock waves, and the point where they meet ($\alpha = 1$) corresponds to conditions at infinity, where these shocks have weakened to Mach lines.

With the leading-edge shock location fixed in the computational plane, a simple iteration is applied to satisfy the relations along the C^- characteristics (neglected in the approximate solution). Figures 3 and 4 compare the approximate solution for a typical case with the results after just three iterations, where the solution is converged to within 1 percent throughout the flow field. The converged solution (which is the solution of the exact equations) includes the transformation locating the shocks, streamlines and characteristics in the physical (x,y) plane. The usual physical flow variables (pressure, velocity, etc.) are recovered from the computational variables through simple relations.

Work is continuing with the eventual goal of producing a code for fully three-dimensional supersonic flows.

---

* Brown University, 401-863-2114
**HSAD, 505-31-43, 804-827-3181

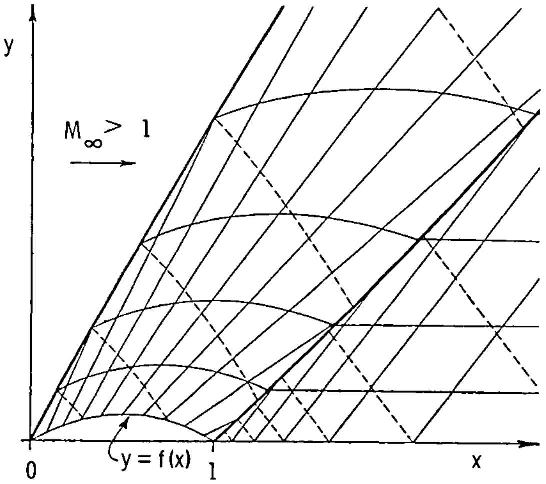


Figure 1.- Supersonic flow past a two-dimensional airfoil, showing shock waves (heavy lines), streamlines and C^+ characteristics (solid lines), and C^- characteristics (dashed lines).

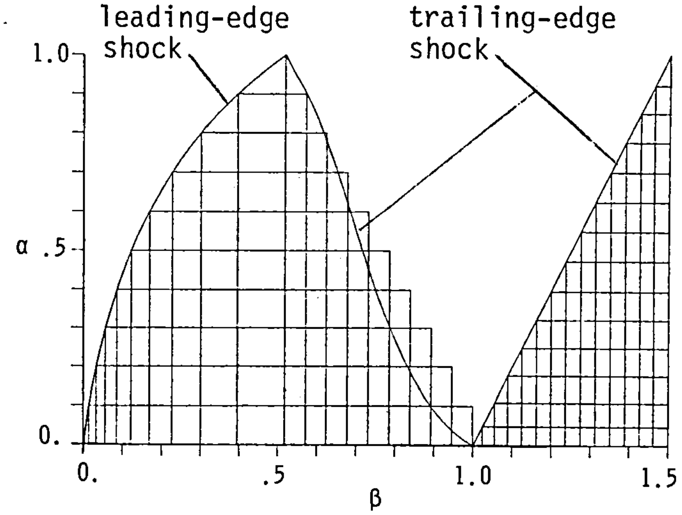

leading-edge shock
trailing-edge shock
1.0
α .5
0.
0.
.5
1.0
1.5
β

Figure 2.- Approximate solution in computational plane for Mach number 2.5. Streamlines are constant α ; C^+ characteristics are constant β.

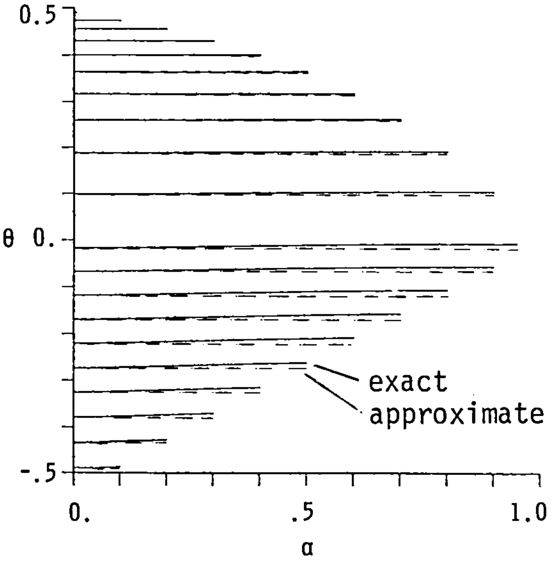


Figure 3.- Approximate and converged solutions for flow angle Θ on each C^+ characteristic of figure 2.

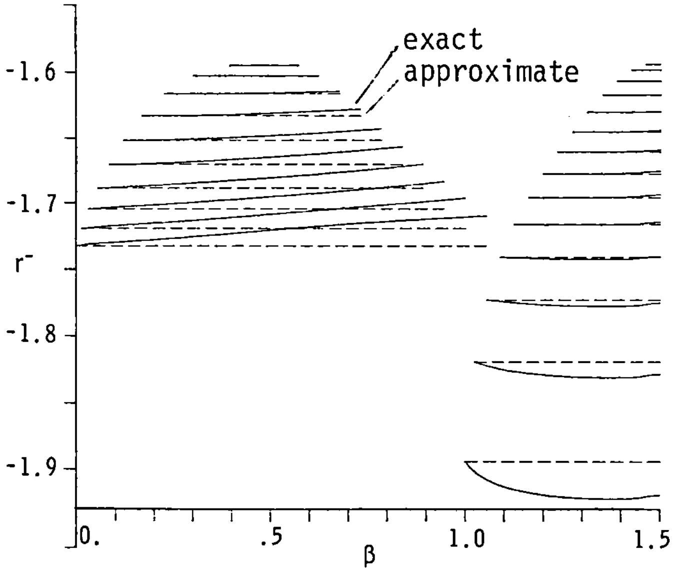


Figure 4.- Approximate and converged solutions for Riemann invariant on each streamline of figure 2.

CALCULATION OF TWO-DIMENSIONAL INLET FLOW FIELDS IN A SUPERSONIC FREE STREAM

Wallace C. Sawyer*

It is of great interest to the designers of supersonic airbreathing missile configurations to understand the complexities of the flow field about the configuration forebodies and the inlet. Analysis methods are of considerable importance to produce efficient designs, e.g., to maximize total pressure recovery, and to minimize cowl drag due to spillage at off-design conditions.

Nielsen Engineering and Research, Inc., under contract to NASA/LaRC (ref. 1-4) has developed a method to calculate two-dimensional inlet flow fields in a supersonic free stream by an implicit, shock-capturing, finite-difference method. The Euler equations are subjected to a general curvilinear transformation and a body-fitted coordinate system is employed. The method is used to solve super-critical, critical, and subcritical flow fields which are simulated by prescribing appropriate conditions at the inlet outflow boundary.

Typical results are shown in figure 1 for an external compression inlet composed of a 10^{o} ramp and a 20^{o} cowl. These Mach 2 results are in excellent agreement with both experimental data and results of an exact solution.

Extensions of the procedure to the case of a nonuniform-free stream is straight forward and is being pursued. Work is also in progress for the inclusion of laminar and turbulent viscous effects. A two-dimensional inlet model is under construction, the test results will be compared with theory for validation of the computational techniques.

References:

1. Biringen, S. H.; and McMillan, O. J.: Calculation of Two-Dimensional Inlet Flow Fields in a Supersonic Free Stream - Program Documentation and Test Cases. NASA CR-3221, March 1980.

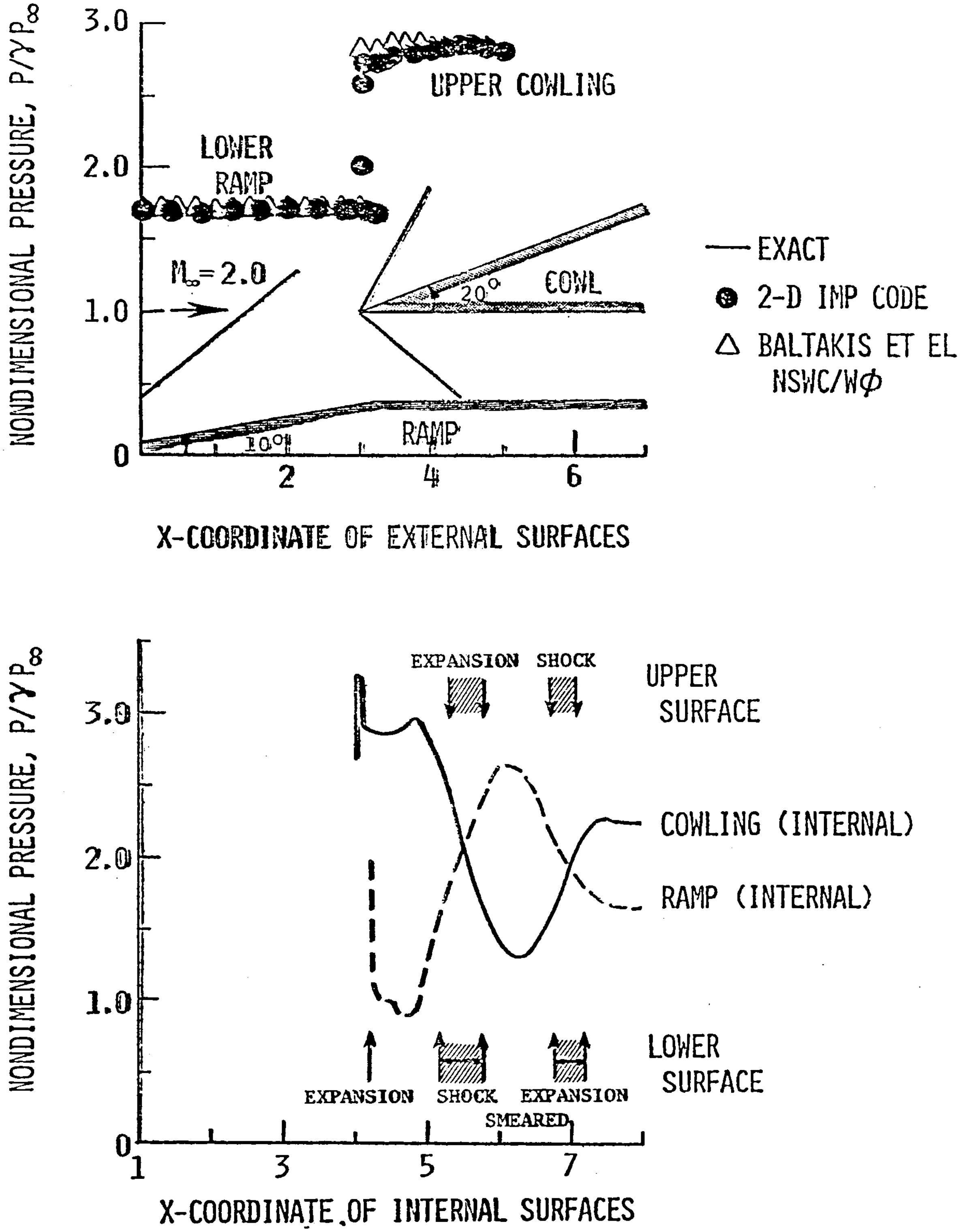
2. Biringen, S. H.; and McMillan, O. J.: Calculation of Two-Dimensional Inlet Flow Fields in a Supersonic Free Stream by an Implicit Marching Code With Nonorthogonal Mesh Generation - User's Manual. NASA CR-3222, January 1980.

3. Chaussee, D. S.; and McMillan, O. J.: A Supersonic, Three-Dimensional Code for Flow Over Blunt Bodies - User's Manual. NASA CR-3223, January 1980.

4. Chaussee, D. S.; and McMillan, O. J.: A Supersonic Three-Dimensional Code for Flow Over Blunt Bodies - Program Documentation and Test Cases. NASA CR-3224, February 1980.

---

*HSAD, 505-43-23-02, 804-827-3181



TWO-DIMENSIONAL SINGLE WEDGE INLET SUPERSONIC FLOW $M_\infty = 2.0$

THE FLOW RESULTING FROM THE INTERSECTION OF AN OBLIQUE SHOCK WAVE WITH A CYLINDER ALIGNED WITH A STEADY SUPERSONIC FLOW

James C. Townsend*

This study addresses the general store-carriage and separation problem. Current analysis methods use linearized supersonic flow theory which does not account for the complex shock wave interactions occurring when a store (such as a missile) is carried in close proximity to an aircraft. A program has been started to examine the details of the flow for this situation, both numerically and experimentally, by first breaking the general problem down into several simple unit type problems.

This particular unit problem is the first reflection of a wing shock from a store. The wing shock is represented as a planar oblique shock wave, and the store it intersects is represented as an infinite cylinder aligned with the supersonic free-stream flow. The free-stream Mach number and the incident shock angle determine the flow conditions between the incident and reflected shock waves, so only the reflected shock conditions and the flow behind it need to be computed. As shown in figure 1, the incident shock is simply reflected where it first intersects the cylinder, but at some point as it wraps around the cylinder the simple reflection can no longer be sustained and the reflection changes to the more complex lambda configuration.

For computation the region in each cross-section plane between the reflected shock and the cylinder is mapped into a rectangular region (fig. 2). The steady-state Euler equations are integrated using a finite-difference method to march in the free-stream direction, that is, along the cylinder axis. The problem is in the initial formulation stage with no computed results available yet.

A complementary experimental study is also planned in support of this work.

---

*HSAD, 505-31-43, 804-827-3181

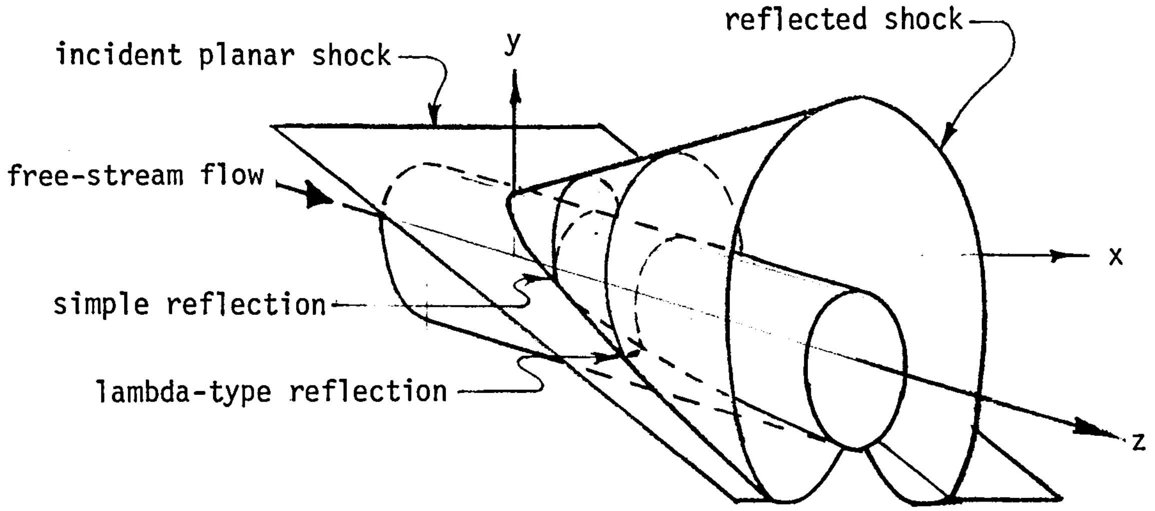

incident planar shock
y
reflected shock
free-stream flow
x
simple reflection
lambda-type reflection
z

Figure 1. - General configuration of shock reflection.

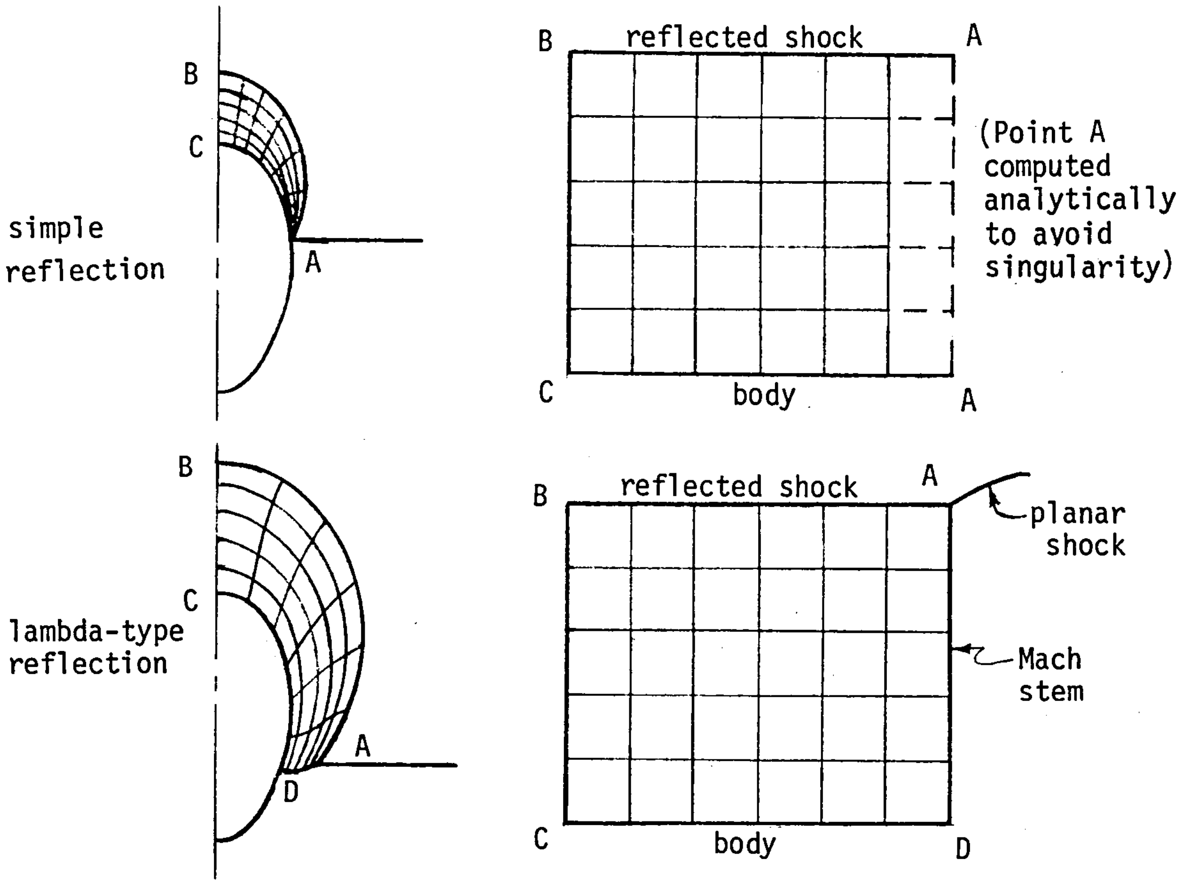

B
C
simple reflection
A
B
reflected shock
A
(Point A computed analytically to avoid singularity)
C
body
A
B
C
lambda-type reflection
A
D
B
reflected shock
A
planar shock
Mach stem
C
body
D

Figure 2. - Mappings from physical to computational plane.

COMPARISON OF RESULTS FROM A FINITE DIFFERENCE CODE WITH RESULTS FROM EXPERIMENTS AND OTHER METHODS

Emma Jean Landrum*

A number of computer codes have been developed or are being developed which show promise of significantly improving the estimation of aerodynamic characteristics for arbitrarily-shaped bodies at supersonic speeds.

One of these codes is a shock-fitting finite difference method (ref. 1) based on general numerical scheme for solving the Euler equations for supersonic flow about complex configurations. A finite difference marching scheme of second order accuracy is used to compute the complete flow field between the body and the bow shock. A conformal mapping of the region between the bow shock and the body into a rectangular region produces the computational grid. When embedded shock waves occur, the mesh is adjusted so that the mesh lines coincide with the shocks and the Rankine-Hugonoit relations are satisfied explicitly across each of the shock waves; this is the floating, shock-fitting method.

Results for an analytically-developed forebody (which resembles the cockpit region of a fighter aircraft) are compared with experiment and four linearized theory methods in figure 1 for Mach numbers of 1.7 and 4.5 at an angle of attack of 5°. The finite difference method provides excellent agreement with experiment at all Mach numbers. In addition, computer solution time for the shock-fitting-finite-difference method is competitive with other codes (ref. 2).

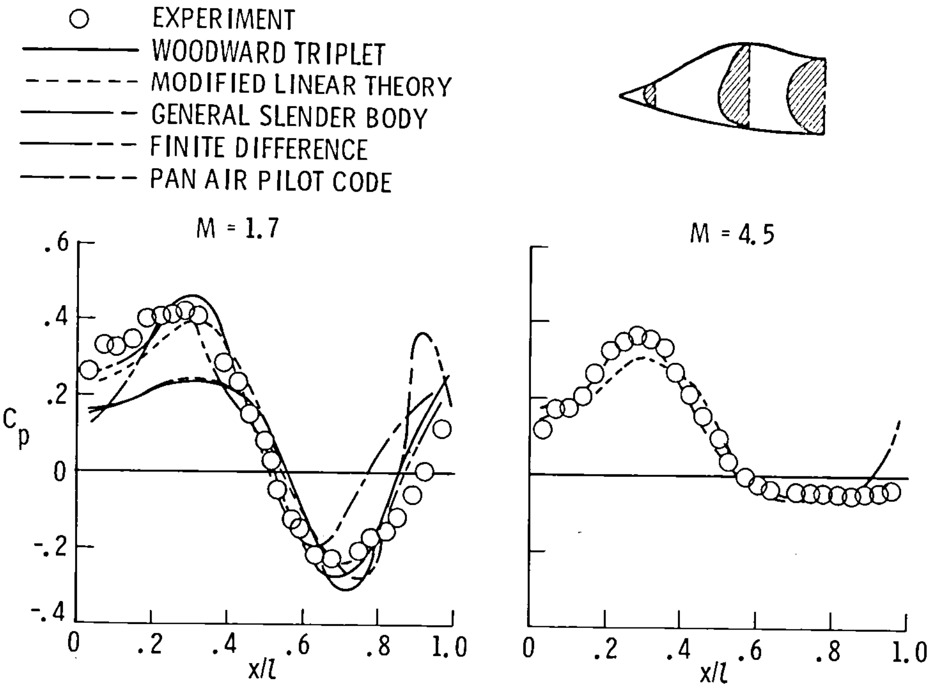
References:

1. Marconi, F.; Salas, M.; and Yaeger, L.: Development of a Computer Code for Calculating the Steady Super/Hypersonic Inviscid Flow Around Real Configurations. NASA CR-2575, April 1976.

2. Landrum, E. J.; and Miller, D. S.: Assessment of Analytic Methods for the Prediction of Aerodynamic Characteristics of Arbitrary Bodies at Supersonic Speeds. Presented at AIAA 18th Aerospace Sciences Meeting, Pasadena California, January 14-16, 1980 (AIAA Paper No. 80-0071, 1980).

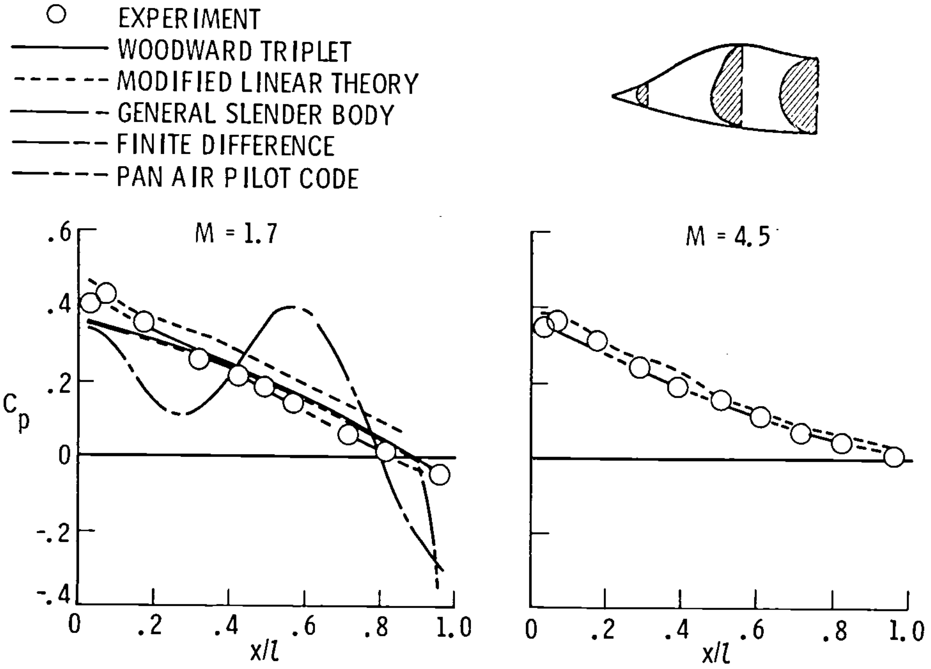
---

*HSAD, 505-31-43-01, 804-827-3181



(a) $\theta = 0^{\circ}$.

Fig. 1.- Analytically developed forebody, $\alpha = 5^{\circ}$.



(b) $\theta = 180^{\circ}$.

Fig. 1.- Concluded.

APPLICATION OF A SOLUTION ON THE EULER EQUATIONS TO THE PREDICTION OF STATIC PERFORMANCE OF 2-D C-D NOZZLES

Mary L. Mason*

Advanced computational models are currently being developed for the solution of the two-dimensional and three-dimensional Euler equations applied to internal nozzle flow and exhaust jet flow. These algorithms must be evaluated by comparisons of the theoretical results with experimental data. Comparisons of theory and data are essential in determining the accuracy of the theoretical results and in defining a valid range of application of the theory.

An experiment has been conducted in the 16-foot transonic tunnel static test facility to define internal performance characteristics for five two-dimensional C-D nozzles. Internal static-pressure data ($p/p_{t,j}$) along the flaps and sidewalls and internal thrust ratio data (F/Fi) were taken at nozzle pressure ratios ranging from 2.0 to approximatley 9.0. The internal performance data may be applied to the evaluation of computational models for prediction of nozzle internal flow behavior.

Since the internal flow data exhibit predominantly two-dimensional characteristics, the force and pressure measurements were compared with the results of the two-dimensional inviscid model of Cline (ref. 1). This algorithm solves the two-dimensional Euler equations for nozzle internal flow and exhaust jet flow, with shock effects incorporated using an artificial viscosity. The results of the data-theory comparison are given in fig. 1 for one of the experimental 2-D C-D nozzle configurations. The x coordinate used in the figure represents the distance from the nozzle throat, positive downstream. The ℓ coordinate used to nondimensionalize x is the distance from the nozzle throat to the nozzle exit. Agreement of data with theoretical results is good for both thrust ratio and static-pressure data. The theory is limited in application to near-design conditions or underexpanded conditions. Since the algorithm is inviscid, it is inadequate for modeling severely overexpanded nozzle flow where internal separation may occur. However, at design nozzle pressure ratios and above, the good agreement between data and theory indicates the effective application of two-dimensional inviscid theory to preliminary design-performance prediction for 2-D C-D nozzles.

Reference:

Cline, Michael C.: NAP: A Computer Program for the Computation of Two-Dimensional, Time-Dependent Inviscid Nozzle Flow. LA-5984 (Contract W-74 05-ENG-36), Los Alamos Scientific Lab., Jan. 1977.

---

*HSAD, 505-32-13, 804-827-2675

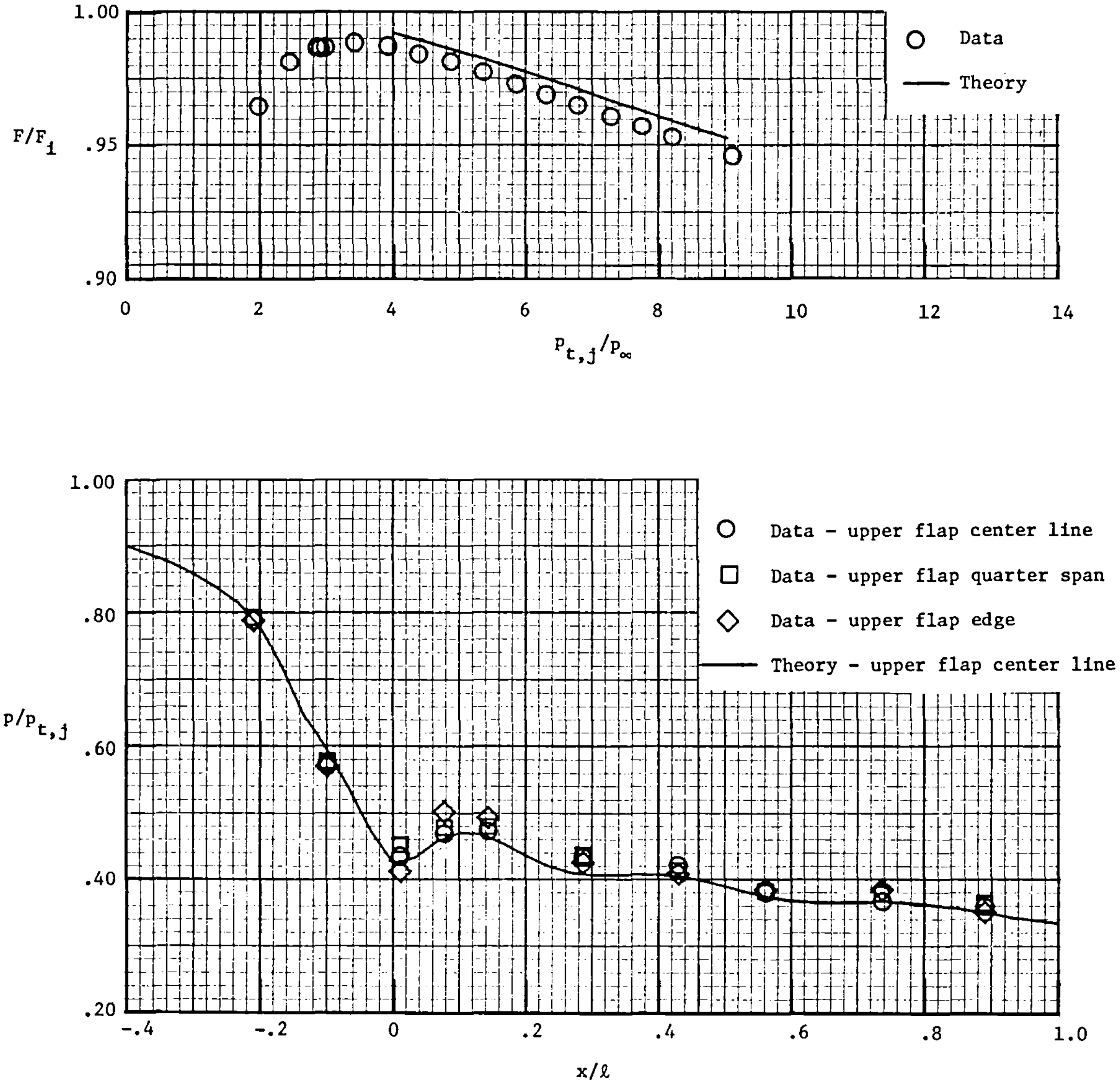


Figure 1. Comparison of theoretical and experimental static pressure distributions and internal thrust ratios.

A SOLUTION OF THE EULER EQUATIONS FOR THE FLOW IN THE ENGINE NACELLE-PYLON-WING REGION OF TRANSPORT AIRPLANES

William B. Compton III*

The integration of a propulsion system with the airframe of a transport airplane must be given close attention if an aerodynamically efficient configuration is desired. The problem becomes even more critical at transonic speeds where strong compressibility effects occur and shock waves are possible. Currently, calculations of the flow over the relatively complex engine nacelle-wing-pylon geometry can only be made with theories on the order of fully linear potential flow coupled with a compressibility correction. Calculations using the Navier-Stokes equations are limited to the flow over less complex configurations such as the afterbody of a nacelle.

This project was initiated to develop a more exact numerical solution of the flow in the region of an airplane's engine nacelle and wing. The Euler equations will be solved using an implicit, time-dependent, finite-difference approach. The Beam and Warming factored ADI scheme was chosen as the numerical algorithm. The solution will be good for highly compressible flows, and will allow for streams having different entropy levels.

Figure 1 shows the computational domain. Within this region, a boundary-fitted coordinate system is generated by solving Laplace's equation, the Thompson-Thames-Mastin method of generating grids, at cross-sectional planes. The curvilinear coordinate system produced in the physical space is then transformed into a cylindrical coordinate system for the computations. Figure 2 shows a cross-section of the coordinate grid in physical space.

As a first step in getting the program operational, the grid shown in figure 2 is being used to compute the flow over a cylinder. If this computation establishes the suitability of the coordinate system, the program will be extended to 3 dimensions with an engine nacelle-pylon-wing geometry. Initially, the airplane geometry will be simplified and only configurations with pointed leading edges and a vertical plane of symmetry will be considered. Future extensions of the program are planned to include the capability of handling blunt leading edges and swept wings.

---

*HSAD, 505-31-43, 804-827-2673

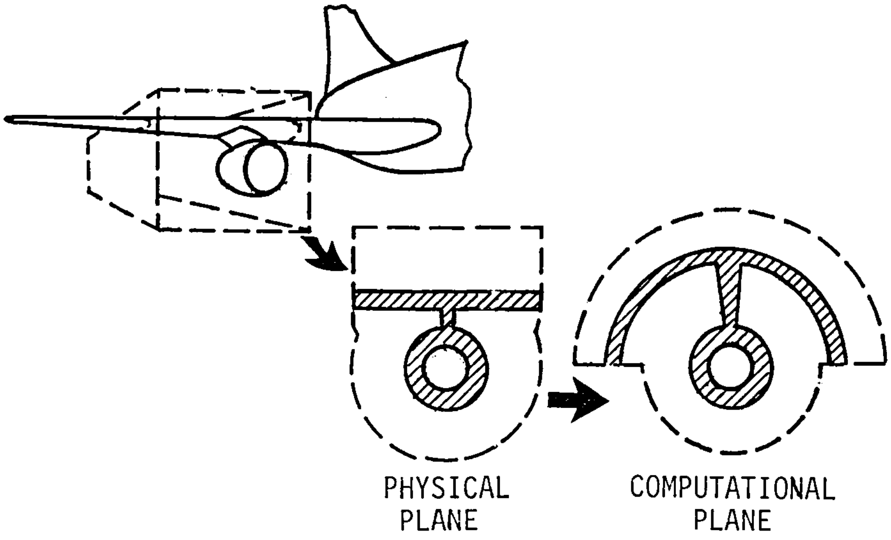

PHYSICAL
PLANE
COMPUTATIONAL
PLANE

Figure 1. - Computational domain.

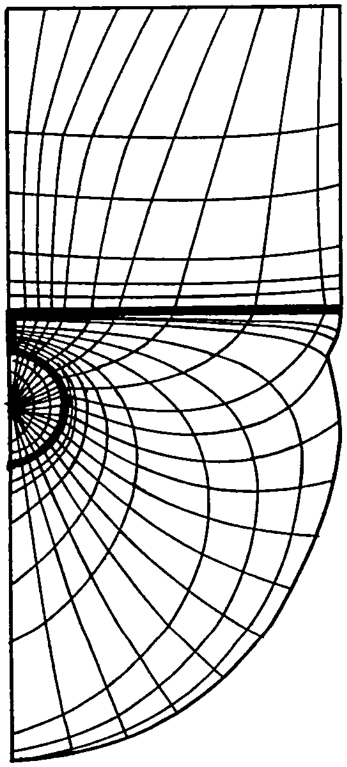


Figure 2. - Cross-section of the coordinate grid in physical space.

6. Navier-Stokes Equations

Methods for solving the Navier-Stokes equations.

Incompressible Flows

Methods designed primarily for solution of incompressible flow.

BOUNDARY-LAYER RESOLVING DIFFERENCE EQUATIONS FOR STEADY, INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Thomas B. Gatski*, Chester Grosch** and Milton E. Rose**

A particularly natural (spline-type) approximation basis for solving a boundary value problem for a linear ordinary differential equation is a patch-basis (ref. 1); under certain circumstances there results a set of difference equations whose solution exactly solves the differential equation at mesh points. This idea has been independently developed and applied to singular perturbation problems in order to demonstrate that accurate difference schemes can be developed which help avoid the need for mesh refinement in boundary layers.

This research investigates an extention of this method to the steady-state vorticity-stream function formulation of the Navier-Stokes equations in two dimensions. A central part of the study aims at comparing classes of computational boundary conditions which have been proposed for these equations. A Newton iteration scheme using a multigrid technique is being explored in order to enable experiments to be conducted economically.

References:
1. Rose, Milton E. "Finite Difference Schemes for Differential Equations" Math. of Comp., Vol. XVIII, No. 86, April, 1964

---

*ANRD, 505-32-03-05, 804-827-2617
**ICASE, 505-31-83-01, 804-827-2513

SPLIT-VELOCITY NAVIER-STOKES TECHNIQUES FOR HIGH REYNOLDS NUMBER FLOWS

Douglas L. Dwoyer*

The split-velocity Navier-Stokes solution procedure was originally developed by Dodge (ref. 1) for the solution of internal flow problems. The method consists of splitting the velocity vector into its rotational and irrotational parts in such a way that the irrotational part of the velocity vector is identified with pressure. This split allows one to solve a modified form of the potential equation for the pressure and transport equations for the rotational part of the velocity. The procedure should be well-suited to high Reynolds number flows since the equations for the rotational part of the velocity need only be solved in the viscous dominated parts of the flowfield.

In the present work, the extension of the split-velocity method to external flows is investigated. Incompressible laminar flow over a flat plate (semi-infinite and finite) was chosen as a model problem. In the laminar case the calculated results agreed very well with asymptotic theory solutions for both the semi-infinite and finite cases at high Reynolds number. Further, computation time was found to be virtually independent of Reynolds number, a typical case requiring less than 1 minute of CYBER-175 time on a 40 x 60 grid.

A comparison of the calculated skin friction along a turbulent semi-infinite flat plate with a curve fit of experimental data is shown in the figure at a Reynolds number per foot of 10^6. There is good agreement between the present results and the data with the calculation requiring less than 2 minutes of CYBER-175 time, which is about 30% more computer time than a laminar case under the same conditions and on the same grid.

The split velocity methods appear to be quite accurate and efficient for calculating external high Reynolds number flows. Based on these results, the method is being applied to more complex flows and a compressible extension is being developed.

References:

1. Dodge, P. R. "Numerical Method for 2D and 3D Viscous Flows" AIAA J., Vol. 15, No. 7, July 1977, pp. 961-965.

---

*STAD, 505-31-13, 804-827-2627

- METHOD VERIFIED FOR HIGH RN TURBULENT FLOW.
- SAME GLOBAL CONVERGENCE HISTORY AS LAMINAR CASE.
- 30% MORE COMPUTER TIME THAN LAMINAR CASE. ADDITIONAL TIME DUE TO EVALUATION OF TURBULENCE MODEL.
- PLANS
  - Turbulent Incompressible Airfoil Calculation
  - Vector Processor Algorithms
  - Compressible Extension

INCOMPRESSIBLE HIGH RN FLAT PLATE FLOW

41 × 60 GRID < 2 MINS. CYBER 175 TIME

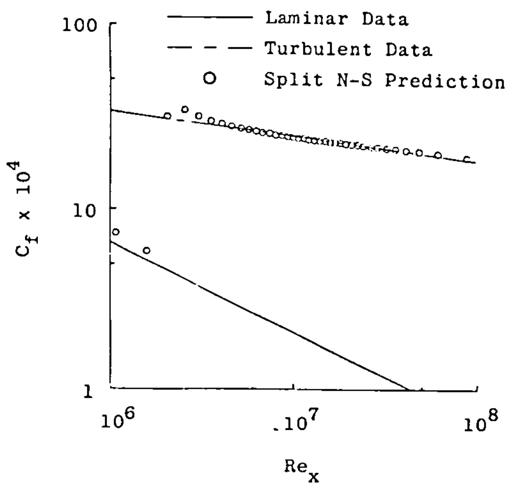


Figure 1.- Velocity split procedure for solving Navier-Stokes equations.

FULLY VISCOUS MULTI-ELEMENT AIRFOIL SOLUTIONS

Frank C. Thames*

In order to obtain more accurate solutions to multi-element airfoil problems, an approach is being developed which is based on the full Navier-Stokes equations. Key features of the method include:

- Use of boundary fitted coordinates (ref. 1) as illustrated in figure 1.
- Simple, algebraic turbulence models (ref. 2).
- Velocity-split fluid equations (ref. 3).
- Multi-grid solution algorithm (ref. 4).

Boundary-fitted coordinates are essential in viscous flow solutions to rationalize the nearly singular behavior of the Navier-Stokes equations in the neighborhood of solid boundaries. For the present simple, algebraic turbulence models will be used. More complex ones may be added later. The velocity-split Navier-Stokes equation formulation allows the flow problem to be split into its constituent potential and viscous parts and each part can be solved on separate grids whose spacing is appropriate to the length scale of the particular type of flow. The choice of the multi-grid algorithm was based primarily on the method's outstanding convergence rate.

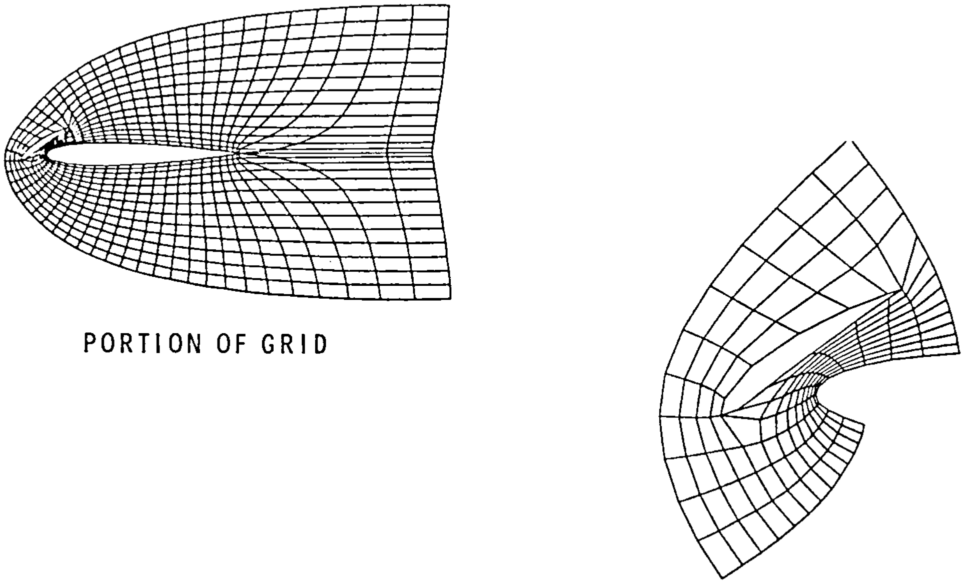
The work to date has been focused on the grid generation task and the evaluation of relaxation schemes to drive the multi-grid procedure. A sample convergence history is given in figure 2. Current results indicate that the multi-directional schemes work best.

References:

1. Thompson, J. F.; Thames, F. C.; and Mastin, C. W. "Boundary-Fitted Curvilinear Coordinate Systems for Solution of Partial Differential Equations on Fields Containing Any Number of Arbitrary Two-Dimensional Bodies" NASA CR-2729, July 1977.
2. Baldwin, R. S.; and Lomax, H. "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows" AIAA Paper 78-257, Jan. 1978.
3. Dwoyer, D. L. "Application of a Velocity-Split Navier-Stokes Solution Technique to External Flow Problems" AIAA Paper 79-1449, July 1979.
4. Brandt, A. "Multi-Level Adaptive Solutions to Boundary Value Problems" Mathematics of Computation, vol. 31, no. 138, April 1977, pp. 333-390.

---

*STAD, 505-31-13, 804-827-2627



PORTION OF GRID

SLOT REGION

Figure 1.- Multi-element airfoil computational grid.

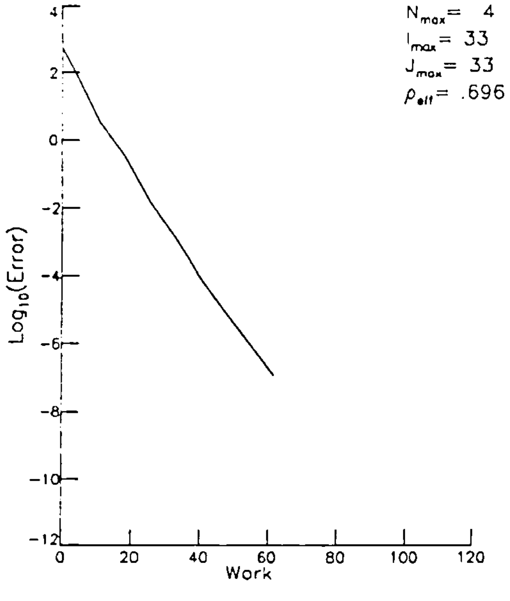


Figure 2.- Convergence history for multi-grid applied to Laplace's equation with an approximate factorization relaxation driver.

Compressible Flows

Methods designed primarily for solution of compressible flow.

NAVIER STOKES SOLUTION FOR THE DYNAMIC STALL OF HELICOPTER AIRFOILS

Warren H. Young, Jr.*

The limitations on helicopter flight envelopes caused by vibratory loads must be predicted in order to assess helicopter agility, blade and control system stresses, and ride quality. The objective of this analytical work is the prediction of unsteady airloads from dynamic stall. The airloads are necessary to predict blade vibratory response in stall flutter. The analysis will be used to analytically evaluate new airfoil design on the basis of susceptibility to dynamic stall onset and point of recovery from stall.

The analysis is a solution of the Reynolds-averaged Navier Stokes equations for two-dimensional, time dependent, subsonic, compressible flow about an airfoil. The equations are formulated in non-conservative form with the energy and mass fluxes as the independent variables. Total temperature was assumed to be constant. The turbulent shear stress has been modeled by an algebraic length scale in the turbulent kinetic energy equation. Flow fields containing laminar, transitional, and turbulent flow are calculated without interruption of the solution scheme.

The finite-difference equations are written in a non-orthogonal, highly stretched coordinate system. The grid allows computation for airfoils of general shape, efficiency in minimizing the number of grid points and considerable control in the placement of grid lines. The finite-difference equations are solved by a linearized block implicit scheme based on the Douglas-Gunn ADI procedure (ref. 1). The calculation scheme allows large time steps and is also suitable for steady solutions (ref. 2).

This work is continuing at Scientific Research Associates, Inc., under contract NAS1-15214. Recent results (fig. 1) for a highly stalled airfoil at one million Reynolds number show the important flow features that were measured (fig. 2) by laser velocimetry. Based on these results, modifications will be made to better resolve the turbulent shear in the free shear layer above the airfoil. Possible future work may include weak shocks in the solution.

References:

1. Gibeling, H. J.; Shamroth, S. J.; and Eiseman, P. R.: Analysis of Strong-Interaction Dynamic Stall for Laminar Flow on Airfoils. NASA CR-2969, April 1978.
2. Shamroth, S. J.; and Gibeling, H. J.: The Prediction of the Turbulent Flow About an Isolated Airfoil. AIAA Paper 79-1543, July 23-25, 1979.

---

*SDD and Structures Laboratory, USARTL, 505-42-13-08, 804-827-2661

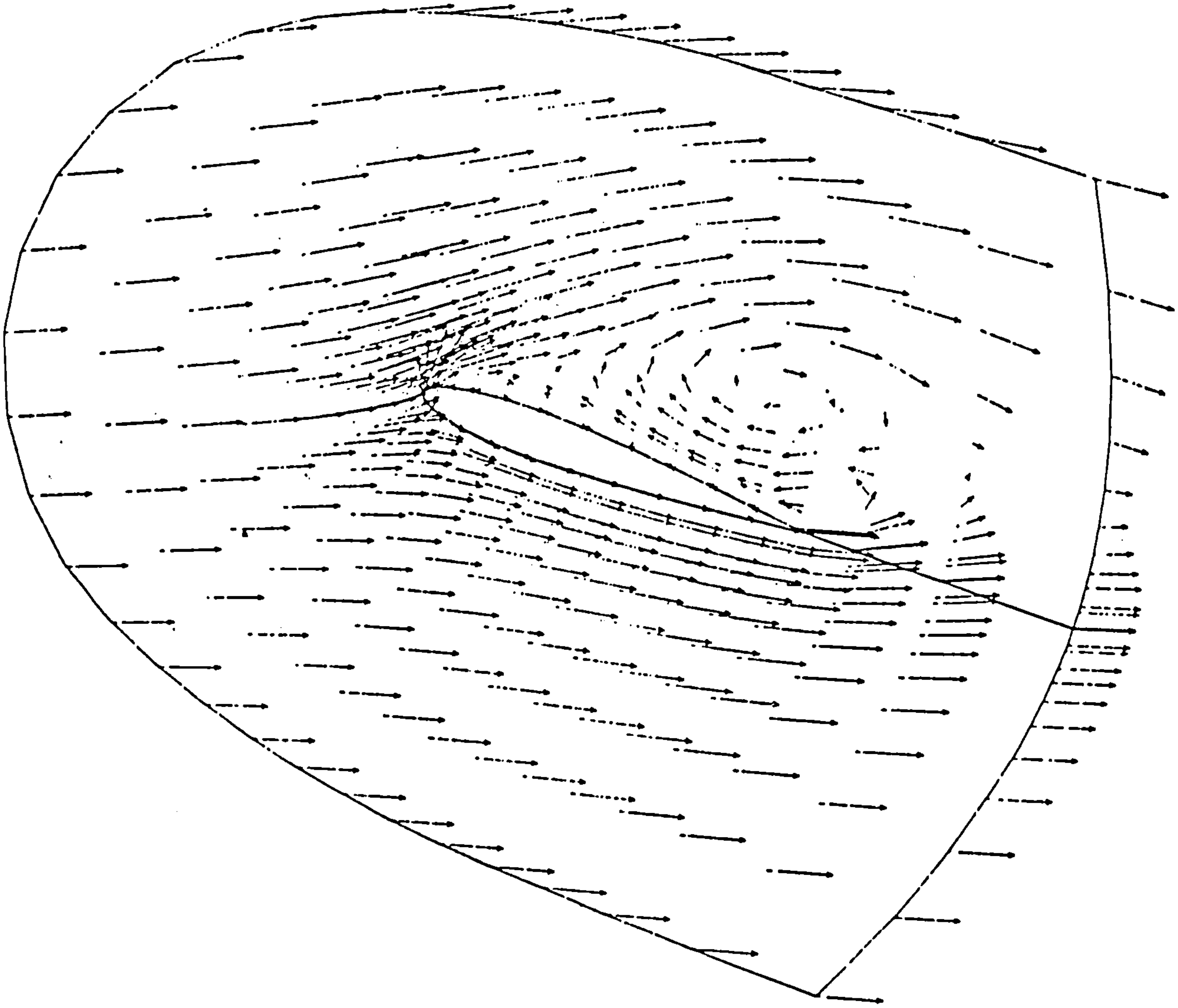


Figure 1.- Computed velocity vector field, at 19^0 angle of attack.

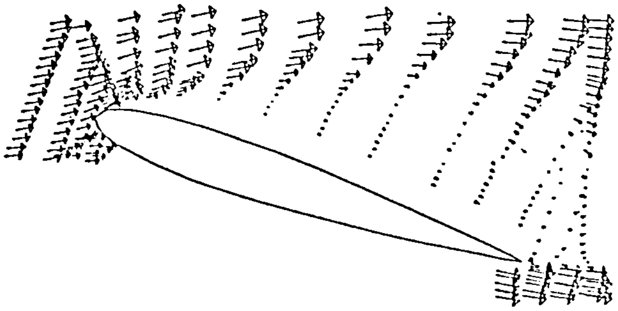


Figure 2.- Velocities measured by laser velocimetry.

Viscous Analysis of Tip Vortex Formation

Warren H. Young, Jr.*

Recent flight tests of the UH-1 helicopter have demonstrated benefits in noise reduction and performance optimization from modifications of tip vortex size. The purposes of the present program are (1) to identify the relation between blade tip planform and the viscous rollup of the tip vortex, (2) to establish benchmark calculations for simpler, more approximate models of the tip vortex formation, and (3) to compare the aerodynamic performance of proposed tip shapes.

A prototype turbulent solution has been completed under contract NAS1-14904. The initiation, formation, and viscous diffusion of a vortex over the upper surface of a thin tip has been calculated. The theory accounts for the vorticity in the lower surface boundary layer (Fig. 1) being convected around the tip, separating from the surface, and forming the tip vortex. The Reynolds-averaged Navier-Stokes equations, with surface boundary conditions and vortex stretching approximations, are solved in calculation planes perpendicular to the freestream direction. A percursor inviscid solution is required to generate the streamwise pressure gradient as input to the present analysis. The present scheme is parabolized in the freestream direction to allow marching through successive calculation planes. The elliptic nature of the Navier-Stokes equations is retained in the two directions that lie in the calculation plane. Solution in the plane is by an ADI scheme (Ref. 1).

Results of the prototype solution are shown in figure 2. The arrows are velocity vectors in a plane normal to the freestream direction. The tip vortex rollup, which was calculated to begin near 30% chord, has moved above and slightly inboard of the tip. The calculation has produced a realistic qualitative results and, upon refinement, will be suitable for a parametric comparison of helicopter rotor blade tip shapes. These results have been published in reference 1.

The development of the technique is continuing under contract to Scientific Research Associates. The current tasks in improving the solution include exact satisfaction of the no-slip condition, restoration of the vortex-stretching terms, and extension to planforms other than rectangular. Possible future tasks include development of a coordinate system to allow the thickness and curvature about the tip to influence the vortex formation. The results will be used to both verify and improve the more approximate, but simpler, percursor inviscid solution.

References:

1. Shamroth, S. J., and Briley, W. R.: A Viscous Flow Analysis for the Tip Vortex Generation Process. NASA CR-3184, Oct. 1979.

---

*SDD and Structures Laboratory, USARTL, 505-42-13-08, 804-827-2661

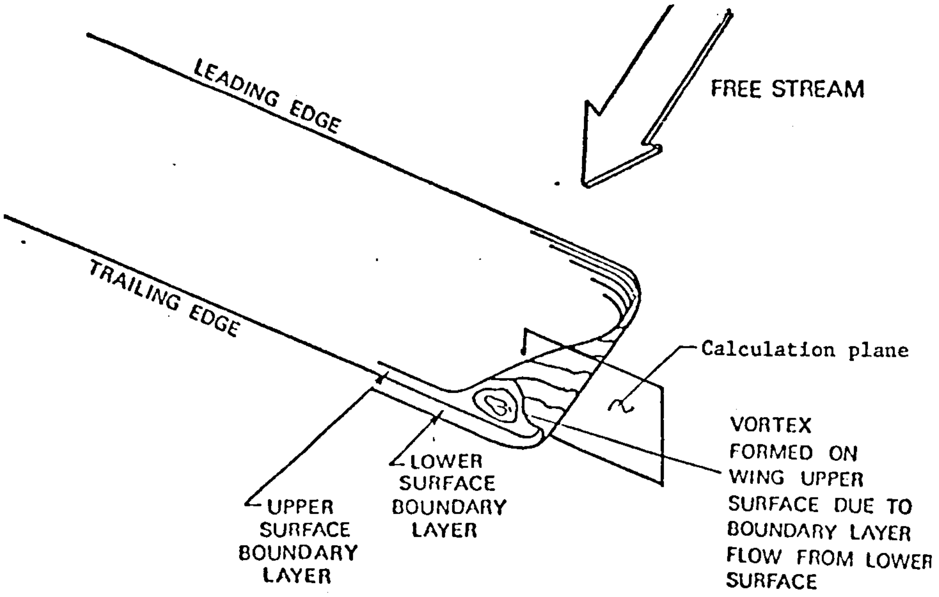

LEADING EDGE
FREE STREAM
TRAILING EDGE
Calculation plane
VORTEX FORMED ON WING UPPER SURFACE DUE TO BOUNDARY LAYER FLOW FROM LOWER SURFACE
LOWER SURFACE BOUNDARY LAYER
UPPER SURFACE BOUNDARY LAYER

Figure 1.- Concept of tip vortex formation process.

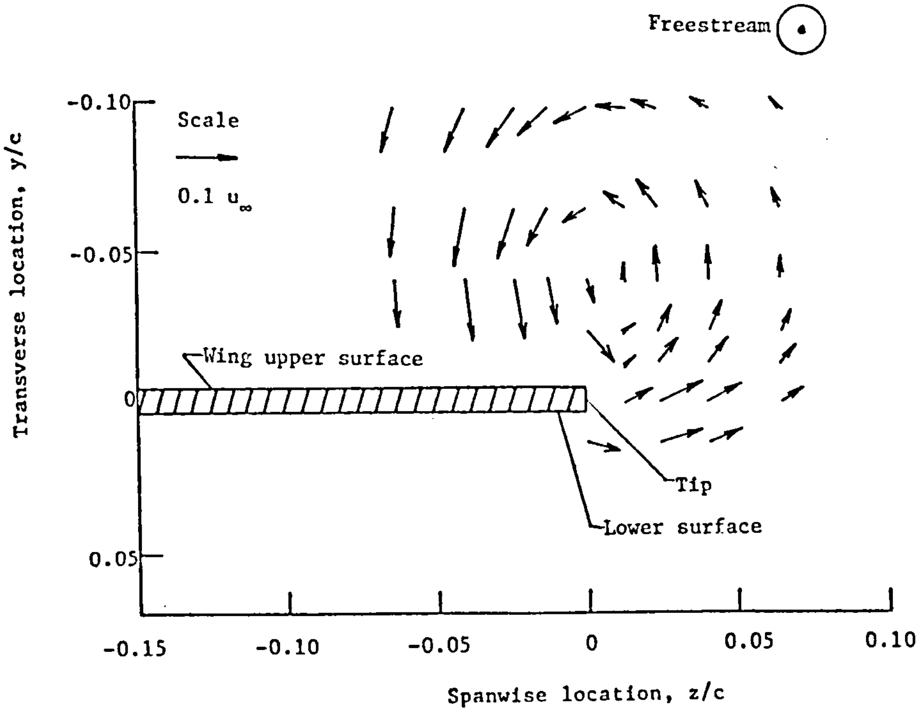


Figure 2.- Velocity vectors in the calculation plane at 75% chord.

COMPUTATION OF THE PERPENDICULAR INJECTOR FLOW FIELD IN A HYDROGEN FUELED SCRAMJET

J. Philip Drummond and Elizabeth H. Weidner*

Work is underway at the NASA Langley Research Center to develop a hydrogen fueled supersonic combustion ramjet (scramjet) capable of operating at hypersonic speeds in the atmosphere. Hydrogen fuel is introduced into the engine in both a parallel and perpendicular direction to the engine primary flow. This approach tailors the injection to an optimum heat release schedule in the engine over the flight Mach number range of interest. Efficient engine design requires a detailed understanding of the flow field near fuel injectors.

A computer program has been developed at Langley to begin studying the flow field near one or more perpendicular fuel injectors that are injecting hydrogen fuel at near sonic conditions into a ducted supersonic crossflow. The flow field is highly turbulent and contains regions of subsonic recirculating flow on either side of the injector (fig. 1). Therefore, solutions to the two-dimensional elliptic Navier-Stokes and one or more species equations are required to describe the flow. These governing equations are integrated by time relaxation using the time-split finite difference technique of MacCormack (ref. 1).

Flow field results for a 6 cm high by 10 cm long duct into which cold H_2 (242 K) is being injected are given in figures 2-6 (ref. 2). Note the strong bow shock ahead of the injector, the expansion of the primary flow over the injector, and the recirculating flow upstream and downstream of the injector. Significant penetration of the hydrogen jet into the primary flow, and convection/diffusion of H_2 upstream of the jet is also apparent in figure 6. These results agree qualitatively with experiment. A quantitative comparison of program results with a data case is given in reference 2.

Current work with the program includes addition of a chemistry model to study H_2-air reaction and improvement of the turbulence model. The analysis of a two-dimensional inlet/combustor with H_2 injection will then be attempted.

References:

1. MacCormack, R. W. and Baldwin, E. S.: "A Numerical Method for Solving the Navier-Stokes Equations with Application to Shock Boundary Layer Interactions," AIAA Paper No. 75-1, 1975.

2. Drummond, J. P.: "Numerical Investigation of the Perpendicular Injector Flow Field in a Hydrogen Fueled Scramjet," AIAA Paper No. 79-1482, 1979.

---

*HSAD, 505-32-93-01, 804-827-2803

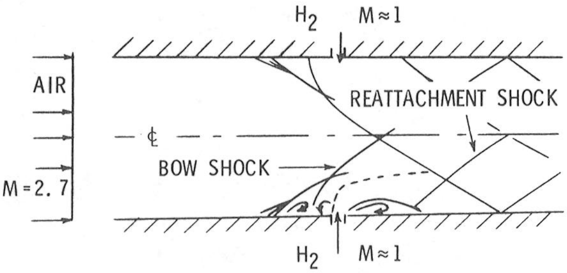

H_2
M ≈ 1
AIR
REATTACHMENT SHOCK
℄
BOW SHOCK
M = 2.7
H_2
M ≈ 1

Fig. 1 Model problem for flow between two struts.

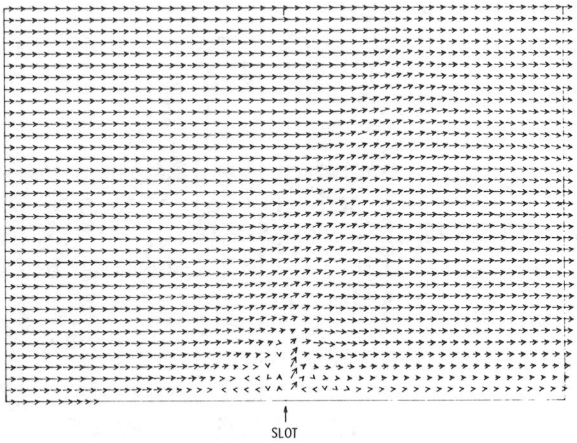


Fig. 2 Velocity vector field in lower half duct.

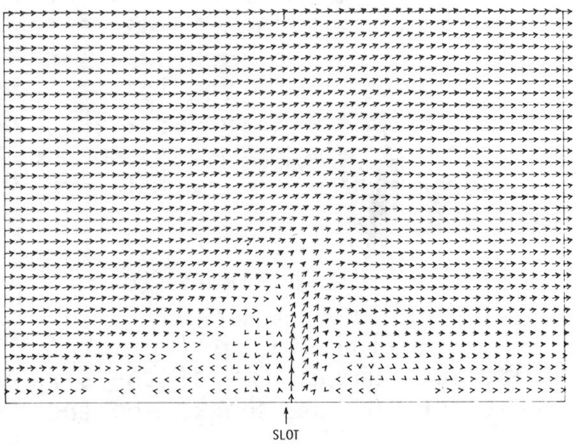


Fig. 3 Magnified velocity vector field about lower injector.

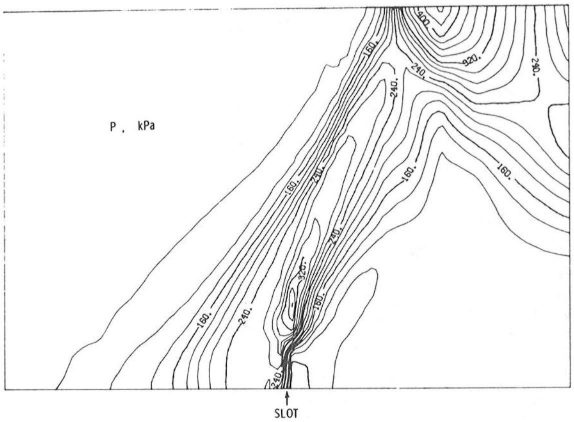


Fig. 4 Static pressure contours in lower half duct.

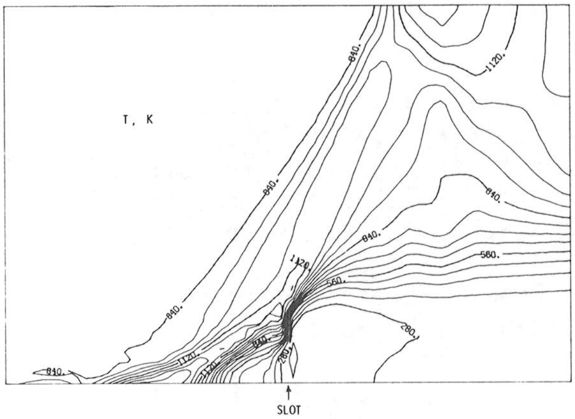


Fig. 5 Static temperature contours in lower half duct.

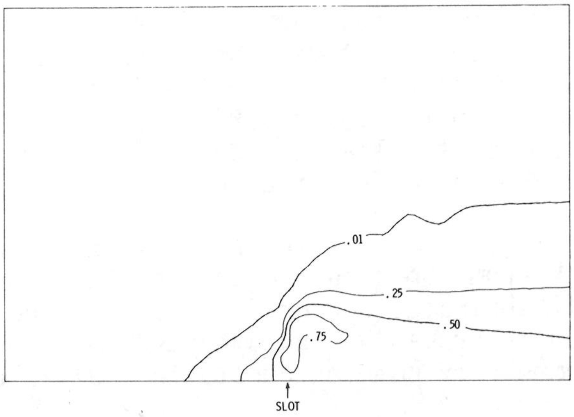


Fig. 6 Hydrogen mass fraction contours in lower half duct.

NUMERICAL ANALYSIS OF THE INLET FLOW OF A SCRAMJET ENGINE

Ajay Kumar*

NASA Langley Research Center is currently engaged in developing a hydrogen-fueled supersonic combustion ramjet (scramjet) engine for hypersonic speeds. It has a fixed-geometry inlet with wedge-shaped sidewalls. Sweep of these sidewalls, in combination with a recess in the cowl, allows spillage to occur efficiently with fixed geometry of the inlet. Inlet compression is completed by three wedge-shaped struts located at the minimum-area section.

The objective of this work is to develop a numerical code to analyze the scramjet inlet flow. In an attempt to understand some of the aspects of this flow, a two-dimensional model problem (fig. 1(a)) is formulated. In this problem, the top surface of the inlet produces a six-degree compression. The cowl plate is located in such a way that there is some space left between the cowl plate and the shock from the top surface to allow for flow spillage. A four-degree half-angle strut is placed in the flow as shown in the figure.

Two-dimensional N.S. equations are used in the conservative law form to describe the inlet flow field. Boundary-fitted curvilinear coordinates are used to transform the physical domain into a rectangular domain (fig. 1(b)) with uniform mesh spacing. The strut EFGH is transformed into a slit E'G' which is coincendent with one of the mesh lines. The transformed equations are solved by an explicit time-asymptotic, finite-difference method due to MacCormack.

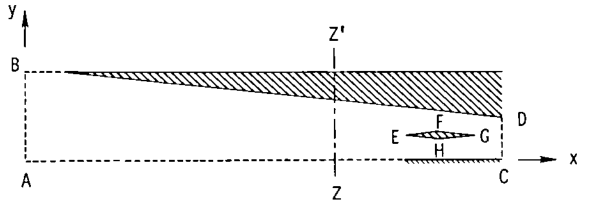
Results presented here are for the laminar flow of air with freestream conditions taken as: $M_\infty = 5$, $P_\infty = 1$ atm, and $T_\infty = 293$ K. Figures 2 and 3 show the plots in the region downstream of line zz' in figure 1(a). Upstream of this line, the shock from the top surface turns the flow downward by about six degrees. Due to this, some of the flow spills out of the inlet. Figure 2 shows the surface pressure distributions. It is seen that the pressure on the top surface remains constant downstream of the compression corner until the shock from the leading edge of the strut's upper surface hits the top surface and gets reflected. This produces a large pressure increase on the top surface. Downstream of this point, the expansion waves from the strut's shoulder interact with the reflected shock and decrease the pressure on the top surface. The pressure on the strut's upper surface shows an increase due to the leading edge shock and than a decrease due to the expansion waves at the shoulder. The pressure on the lower surface of the strut shows two expansion waves produced by the strut's leading edge and the shoulder. These expansion waves interact with the shock from the leading edge of the cowl.

Figure 3 shows the plot of the velocity vectors. It is seen that the shock from the strut separates the flow on the top surface of the inlet. A small separated region is also produced near the trailing edge of the strut which is probably caused by the cowl shock. The flow downstream of the strut becomes very complex due to the multiple interactions of the shocks and the expansion waves as seen in figure 4 which shows pressure variation across the inlet at x = .1089 m, just upstream of the inlet outflow boundary.

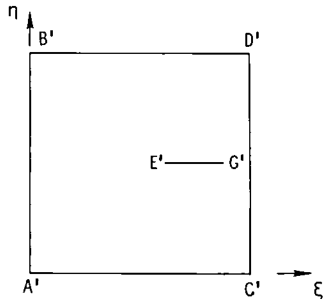
---

*Old Dominion University, Norfolk, VA, under Contract NAS1-15930, 804-827-2803

A mesh size of 51 x 51 is used in the above analysis. The converged solution is obtained in 2150 time-steps which required two minutes of computing time on CDC-CYBER-203 computer.



(a) Physical plane



(b) Transformed plane

Fig. 1 Configuration of model inlet problem

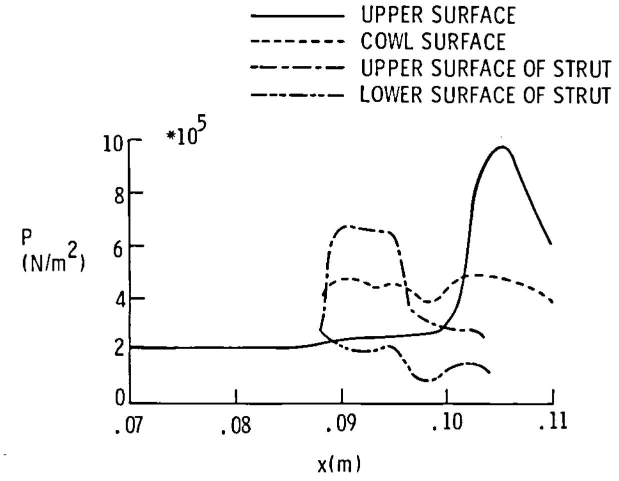


Fig. 2 Surface pressure distributions

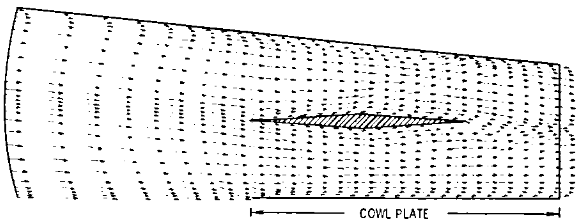


Fig. 3 Velocity vector field downstream of line zz'

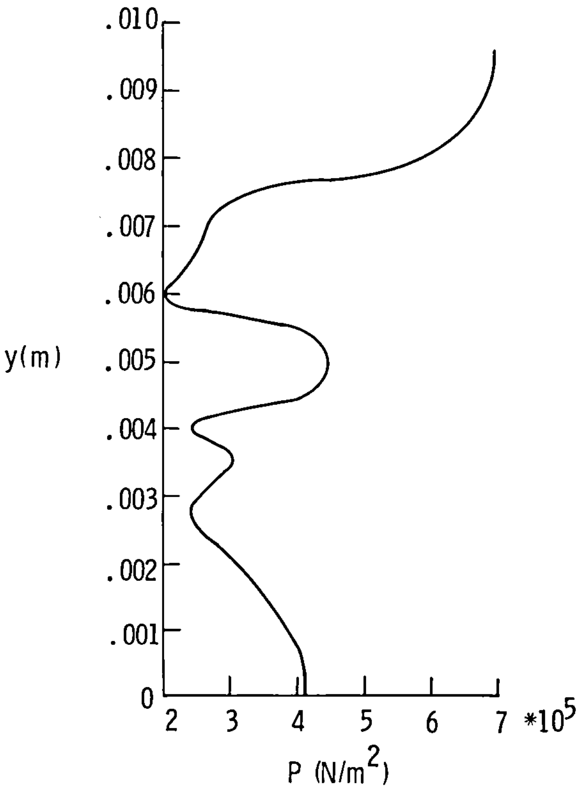


Fig. 4 Pressure variation across the inlet just upstream of outflow boundary

GIM/STAR: EXTERNAL AND INTERNAL FLOWS FOR HYPERSONIC VEHICLES

L. W. Spradley, J. J. Stalnaker, A. W. Ratliff,*
J. L. Hunt and J. P. Drummond**

The General Interpolants Method (GIM) is a hybrid procedure for generating numerical analogs of the three-dimensional Navier-Stokes equations. A code using GIM is now available on the Langley CDC/STAR computer (ref. 1). The GIM/STAR effort is an interrelated program, being pursued by both Langley's Hypersonic Aerodynamics Branch (HAB) and Hypersonic Propulsion Branch (HPB) through coordinated contracts with Lockheed/Huntsville.

The GIM/STAR code combines the finite element geometry point of departure with finite difference computational analysis. Complex 3-D flow domains can be modeled due to the allowance of regional numerical transformations. Grid planes and metric coefficients are generated in the transformed coordinates; computations are made in the physical plane after the application of inverse transformations. Steady-state solution of the governing equations is obtained with a transient procedure using the MacCormack unsplit finite difference numerical integration technique.

Two complex flow types, relevant to Langley's hypersonic program, that have been computed with GIM/STAR are shown in the figures. The first illustration is that of a Mach 5 flow into a two-dimensional inlet, configured to tax the code's capabilities. The external flow was treated inviscidly. The shock generated by the 25° ramp angle is shown as captured by the code; 34 percent of the flow is spilled at the cowl lip as predicted by theory. Inside the duct, the Navier-Stokes equations were solved with a no-slip condition on the upper wall and an assumed laminar boundary layer. The turning of the internal flow was accomplished by a strong expansion in the upper initial portion of the duct diffuser and a strong shock emanating from the cowl lip. The shock impinged the internal upper wall and resulted in separation of the boundary layer and formulation of a recirculation zone. The second illustration shows the pressure contours calculated in and about a three-dimensional supersonic exhaust plume emersed in a hypersonic external flow. The strong pressure gradient outboard the nozzle lip indicates the presence of a shock created by the underexpanded plume spilling over into the external expanding flow.

Work is currently underway to enhance the overall computational speed of the code in supersonic viscous and inviscid flow domains by incorporating a forward-marching scheme in combination with transient relaxation at each cross plane. This algorithm allows direct solution of a parabolic form of the governing equations without the necessity of a pressure iteration and is readily adaptable to a hyperbolic solution of the Euler equations as well. Turbulence and chemistry models are to be incorporated into the code during the next 12 months.

Reference:
1. Spradley, L. W. and M. L. Pearson, "GIM Code User's Manual for the STAR-100 Computer-Version SE-1," NASA CR-3157, November 1979.

---

*Lockheed-Huntsville, 205-837-1800
**HSAD, 505-31-73, 804-827-3294; 505-05-43, 804-827-2803

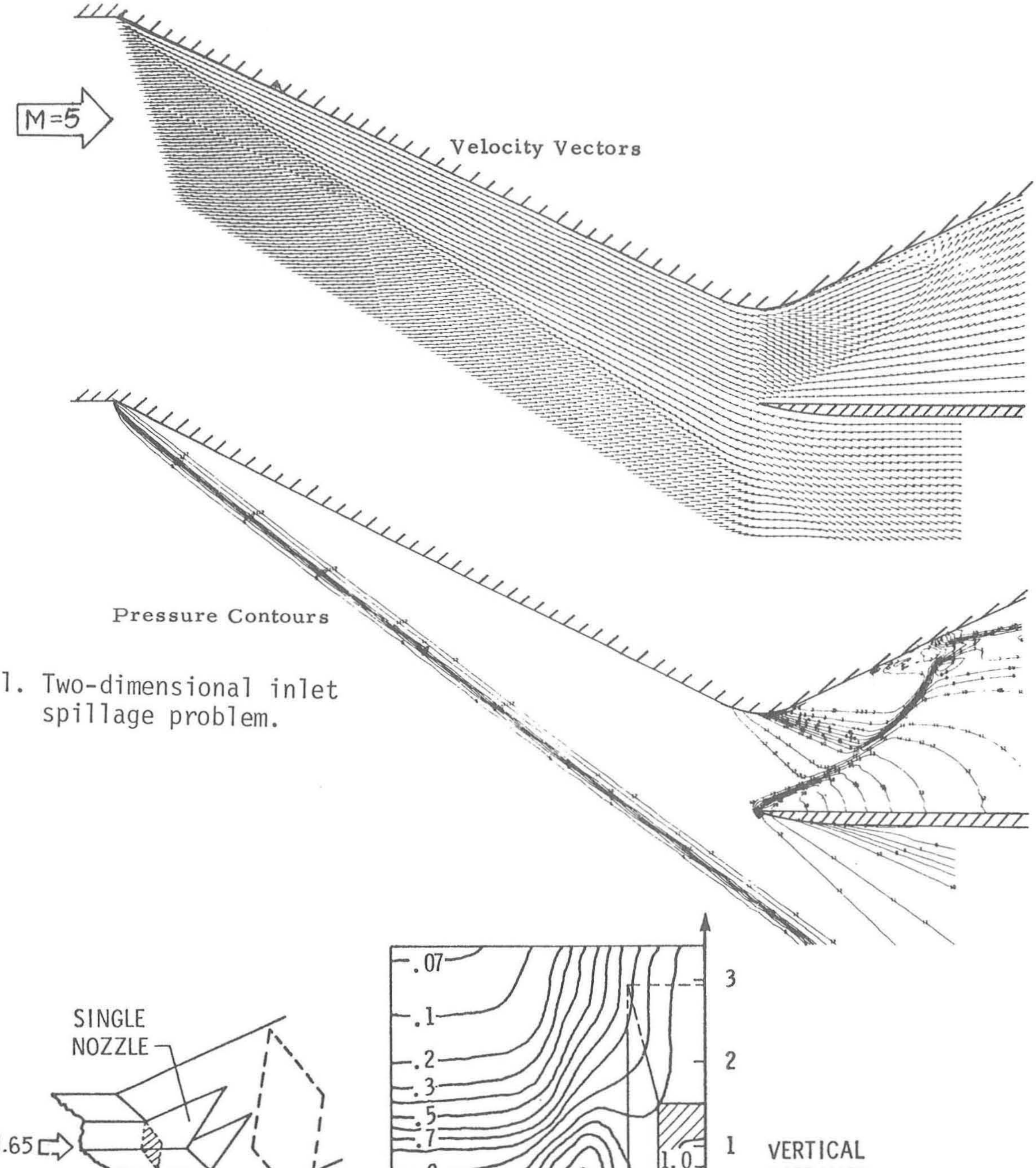

M=5
Velocity Vectors
Pressure Contours

Fig. 1. Two-dimensional inlet spillage problem.

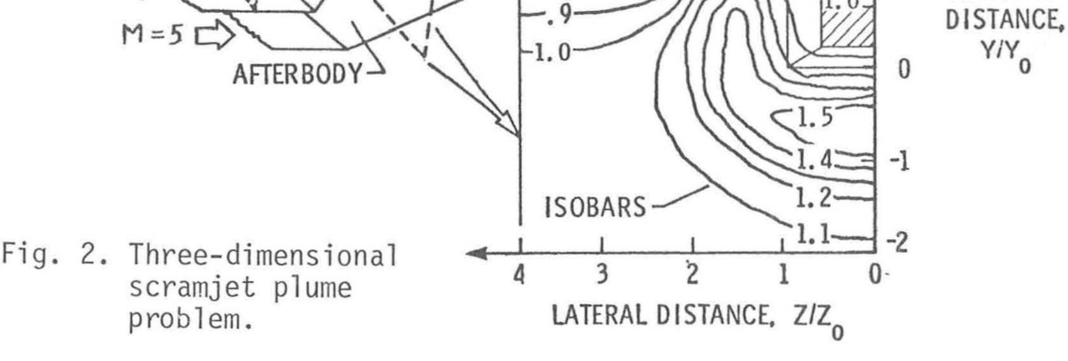


Fig. 2. Three-dimensional scramjet plume problem.

MIXED SPECTRAL/FINITE DIFFERENCE METHOD FOR COMPRESSIBLE NAVIER-STOKES EQUATIONS

Thomas A. Zang and M. Y. Hussaini*

In high Reynolds number flows the numerical dissipation resulting from the truncation error of the advection terms may overwhelm the physical dissipation from the viscous terms. A numerical scheme for compressible, mildly viscous flows which employs a standard finite difference approximation for the viscous term but resorts to a pseudo-spectral method for the advection terms is examined here. This method offers the prospect of greatly reducing the major source of numerical error without dramatically increasing the computational effort (unlike a fully pseudo-spectral method which is quite costly to apply to the complicated compressible viscous terms).

Several model problems have been considered. The first is the familiar Burger's equation with periodic boundary conditions. A spectral/finite-difference (SFD) method was compared with a second-order, fully finite difference (FFD) method. Explicit time-differencing was applied to the advection term and implicit time-differencing to the diffusion term. Figure 1 illustrates the results. The solid curve is the exact solution and the open circles are the numerical results. Not only does the SFD method produce high accuracy away from the shock, but it produces a fairly narrow shock as well.

The SFD method has also been applied to a set of viscous, compressible, quasi-one-dimensional euqations that approximate the flow in a nozzle of slowly varying cross section. Here, the pseudo-spectral portion uses Chebyshev polynomials. The steady-state numerical solution for one case is shown in Figure 2. Two Chebyshev grids have been patched together at the throat in order to improve the resolution at the sonic point. Some smoothing has been applied to the numerical results, as indeed is required for presentable results even for standard second-order finite difference methods.

A third application of the SFD method has been to a set of inviscid fluid equations with periodic boundary conditions that model some forced gas flows of astrophysical interest. The steady-state solutions exhibit strong shocks for which the advection steepening is balanced primarily by Coriolis forces rather than by pressure forces. The density obtained in one SFD calculation which included a small, explicit viscous term is shown in Figure 3.

---

*ICASE, 505-31-83-01, 804-827-2513

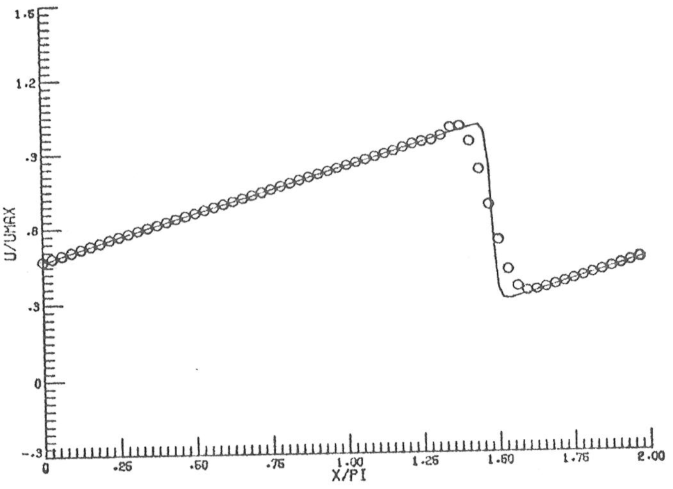


Fig. 1a: Fully finite difference solution to Burger's equation.

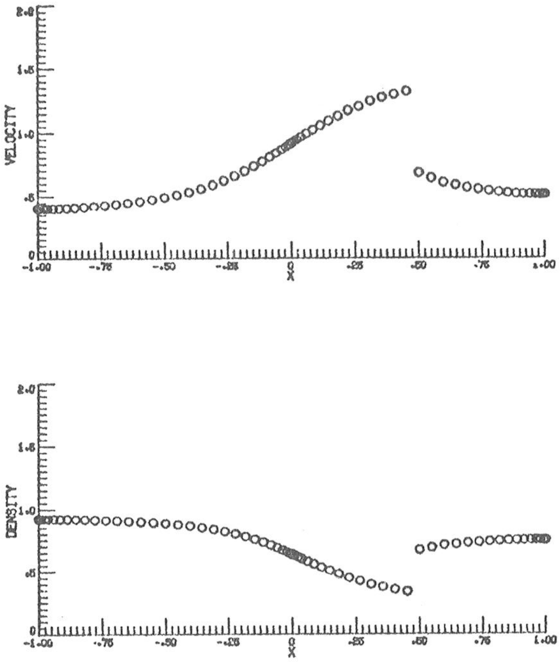


Fig. 2: Spectral/finite difference solution to quasi-one-dimensional nozzle flow.

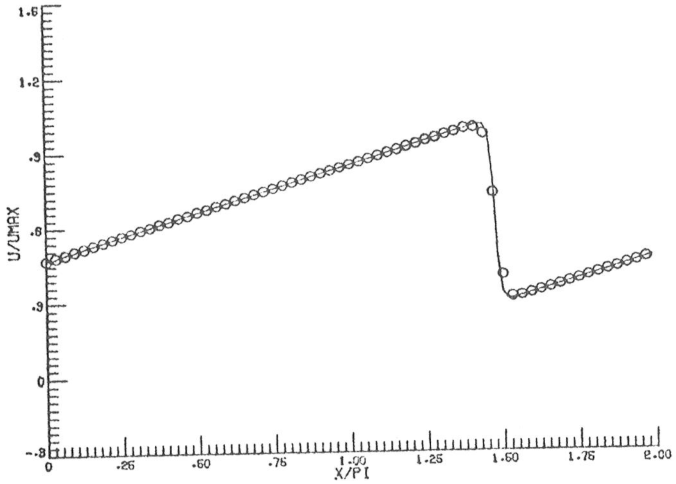


Fig. 1b: Spectral/finite difference solution to Burger's equation.

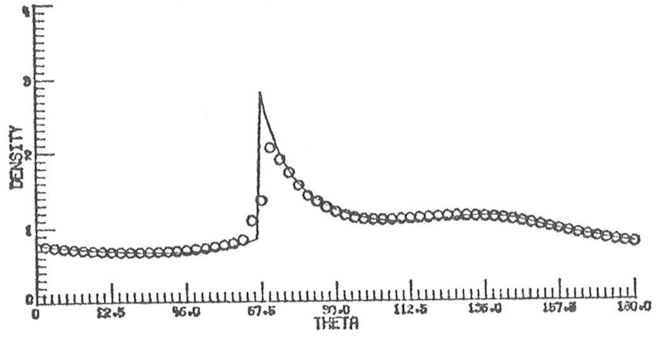


Fig. 3: Spectral/finite difference solution to astrophysical problem.

OUTFLOW BOUNDARY CONDITIONS FOR FLUID DYNAMICS

Alvin Bayliss* and Eli Turkel

In many problems in exterior aerodynamics one uses a time-dependent code to march the fluid dynamics equations to a steady state. Since these equations admit wave-like solutions, a steady state is achieved only because energy radiates out to infinity. In order to compute the solution in an exterior region the domain of integration must be artifically restricted. Mapping the infinite region into a finite region is frequently not useful because the waves are not resolved near infinity.

In this work we develop boundary conditions at the artificial surfaces which reduce the reflections into the computational domain. In addition, the boundary conditions lead to a well posed problem. The conditions are generalizations of those previously derived for the wave equation (ref. 1).

When a subsonic outflow occurs it is necessary to specify one condition at the outflow boundary. Frequently the pressure is specified, but this leads to large reflections from the boundary. For supersonic flow the top boundary is frequently subsonic with respect to the normal velocity. This again requires the imposition of one boundary condition. Extrapolating all the variables degrades the accuracy of the solution.

These new boundary conditions involve time derivatives as well as point values of the pressure and velocity fields. Hence, these conditions are easy to implement.

Two test cases were considered. In the first case the Navier-Stokes equations were solved with initial conditions that were constant except for a localized disturbance. Analytically, the disturbance is convected out of the computational domain. This simple problem tests the ability of the boundary condition to transmit the disturbance without introducing artificial reflections. This problem was run with several different exit boundary conditions. The results of Table 1 clearly indicate the improvements obtained.

In the second case the Navier-Stokes equations with subsonic flow are solved over a flat plate. The radiation conditions were imposed at both the outflow and top boundaries. The results are presented in Table 2. It is evident that the imposition of the correct radiation conditions permits a substantial reduction in the size of the computational domain as well as in the number of iterations required to reach steady state.

Further studies involving nozzle flow also show substantial improvement with the use of the new boundary conditions.

---

*ICASE, 505-31-83-01, 804-827-2513

TABLE 1

Boundary Condition	No. Iterations
$p = p_\infty$	> 20000
Rudy-Strikwerda (ref. 2)	3383
new	1850

TABLE 2

b.c. (outflow)	b.c. (top)	Position of Top	No. Iterations
$p = p_\infty$	extrapolation	10	> 20000
new	extrapolation	10	13200
new	new	10	8750
$p = p_\infty$	extrapolation	5	inaccurate
new	new	5	8800

References:

1. Bayliss, A. and Turkel, E. "Radiation boundary conditions for wave-like equations," to appear in Comm. Pure Appl. Math.

2. Rudy, D. and Strikwerda, J. "A non-reflecting outflow boundary condition for subsonic Navier-Stokes calculations," to appear J. Computational Phys.

COMPUTATION OF THE FLOW AT A SHUTTLE-TYPE WING ELEVON JUNCTURE

Joanne L. Walsh*
and
John C. Strikwerda**

Some type of sealing arrangement is required to prevent ingress of hot boundary layer gas into the unprotected internal wing structure at the juncture of the shuttle orbiter wing and elevon. (See left portion of figure 1). The cove environment is unknown but potentially hostile, and experiments and analyses are underway at Langley to define it.

In the numerical study a 2D mathematical model using the full two-dimensional compressible Navier-Stokes equations for laminar supersonic flow at the wing-elevon cove juncture has been developed. An existing computer program (Ref. 1) which runs on the Cyber 203 computer was modified to do the study. An explicit finite difference technique is used to solve the governing equations. The computational domain is discretized into over 6500 grid points and at each point two velocity vectors, density, and temperature are determined. Numerical size of the problem and a small computational time step restricted the model to the area shown in the lower right portion of figure 1. To accelerate the convergence to steady state, the stability condition at each point is used to determine a local time step which is used to advance the solution.

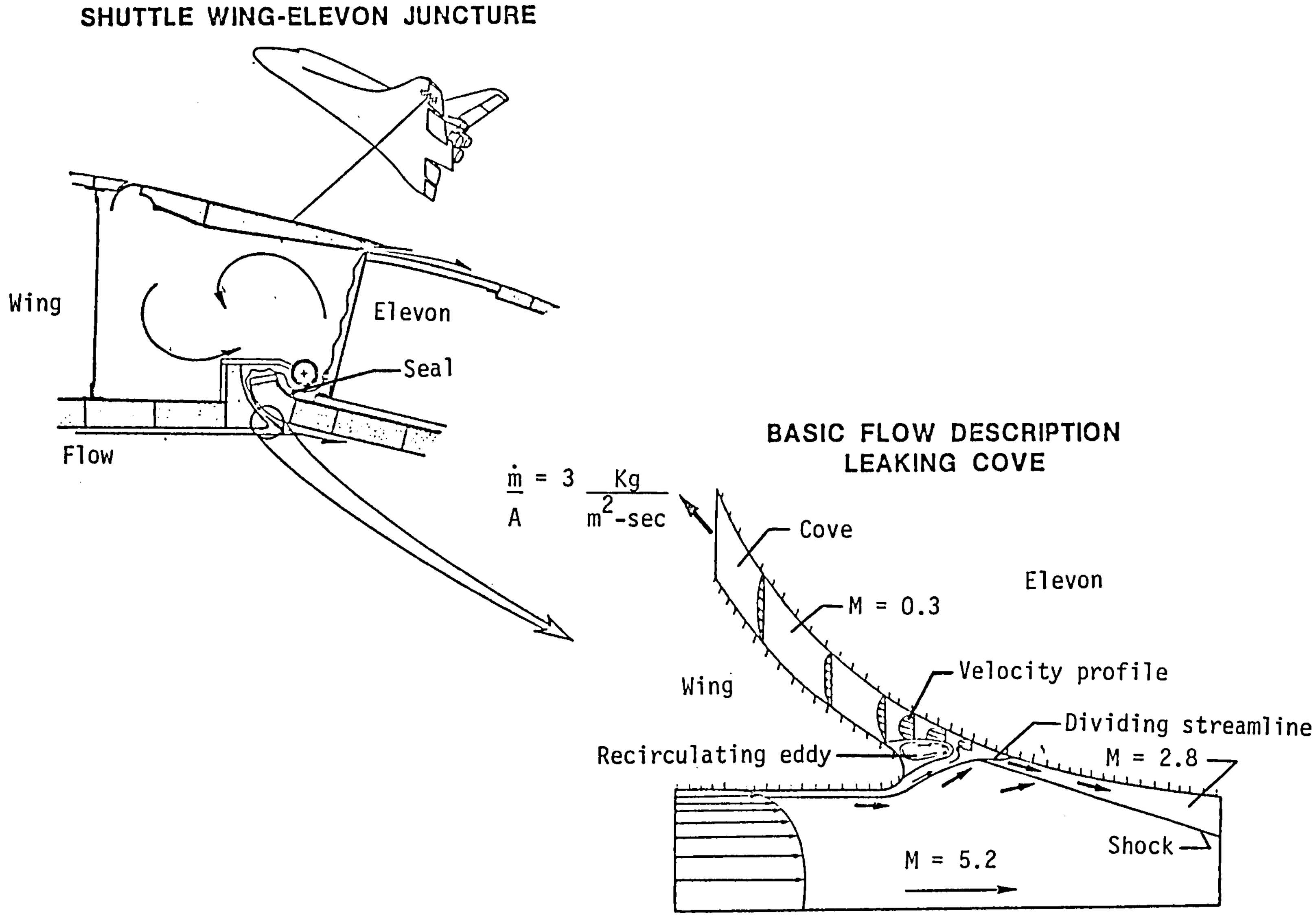
The analysis has defined the complex interaction between the external boundary layer flow and the flow in the cove. The basic flow phenomena for a nominal leak rate of 3 kg/m^2-sec (0.6 lb/ft^2-sec) is illustrated in the lower right of the figure. As the flow expands off the wing surface, a recirculating eddy develops at the entrance to the cove. A small portion of the external boundary layer flow between the body and the dividing streamline is ingested through the cove. The remainder of the boundary layer is compressed by the deflected elevon surface and results in a shock. The leak in the cove is simulated by specifying the pressure at the cove exit to be less than the free-stream pressure. The flow description obtained from the detailed analysis provides the required insight into the overall phenomena to permit simplifying assumptions that make an aerothermostructural analysis of the wing-elevon-cove structure possible. The flow analysis is being continued to study the effects of different wing surface temperatures and the effects of a range of leak sizes, including the no leak condition (i.e. sealed cove), on the cove environment.

References:

1. Strikwerda, J. C.: A time-split difference scheme for the compressible Navier-Stokes equations applied to flows in slotted nozzles. ICASE report to appear.

---

* SDD, 506-53-73, 804-827-3731
** ICASE, 804-827-2513



SHUTTLE WING-ELEVON JUNCTURE

BASIC FLOW DESCRIPTION LEAKING COVE

Wing
Elevon
Seal
Flow
$\frac{\dot{m}}{A} = 3 \frac{Kg}{m^2\text{-sec}}$
Cove
M = 0.3
Elevon
Wing
Velocity profile
Dividing streamline
M = 2.8
Recirculating eddy
Shock
M = 5.2

Figure 1.- Basic flow description at wing elevon juncture with cove leakage.

COMPUTATION OF THE FLOW IN SLOTTED NOZZLES

John C. Strikwerda*

The steady-state laminar flow in nozzles with suction slots has been computed using the compressible Navier-Stokes equations. Both two-dimensional and axisymmetric flows have been computed. The purpose of these computations was to aid in the design of the quiet wind tunnels being developed at NASA Langley Research Center (Ref. 1 and Fig. 1).

The finite difference scheme is an explicit time-split scheme with three splittings. The equations are split into two one-dimensional hyperbolic schemes for the inviscid terms and a parabolic scheme for the viscous terms. Each of these schemes is second-order accurate in space and time and highly vectorizable.

The computational grid was generated numerically using a variant of the Thompson method. The computational domain was the union of two rectangles joined along a portion of one side.

Flows were computed for Reynolds numbers as high as 20 million with Mach numbers of up to 2 at the exit boundary. The initial flow field was based on the one-dimensional inviscid theory of Laval nozzles. The steady-state solution was computed by solving the time dependent Navier-Stokes equations until a steady state was attained.

The calculations were performed on the CDC STAR-100 vector processor. The computer code is written in SL/1 and utilizes 32-bit arithmetic.

Reference:
1. Beckwith, I. E. "Development of a High Reynolds Number Quiet Tunnel for Transition Research" AIAA Journal, Vol. 13, No. 3, March 1975, pp. 300-306.

---

*ICASE, 505-31-83-01, 804-827-2513

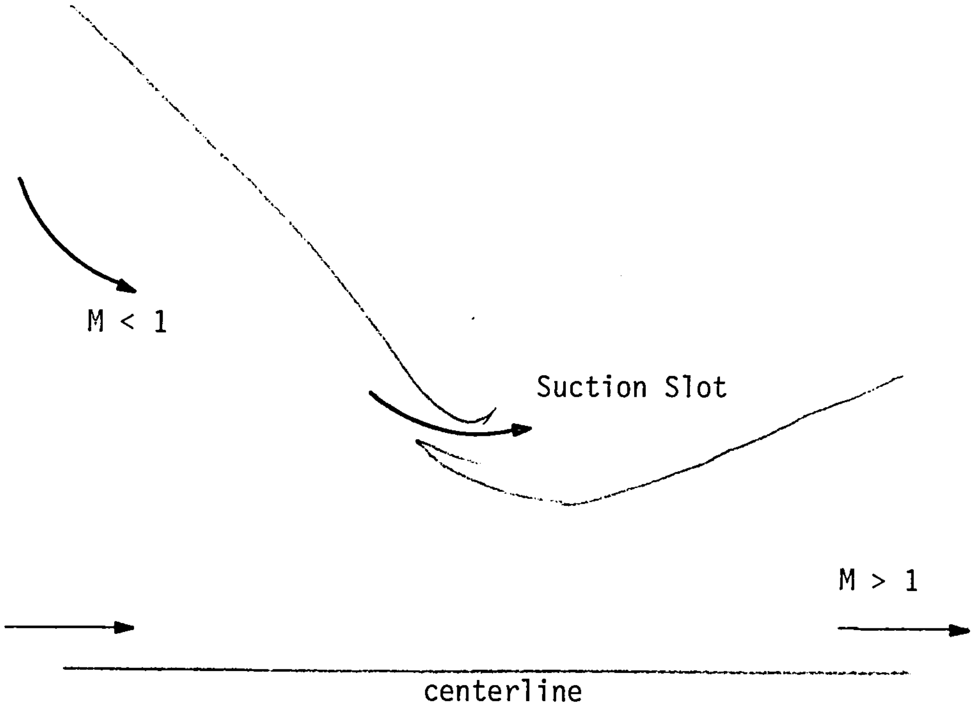

M < 1
Suction Slot
M > 1
centerline

Figure 1. Illustration of a Half-section of a Nozzle with a Suction Slot.

A CLASS OF IMPLICIT DIFFERENCE SCHEMES FOR COMPRESSIBLE NAVIER-STOKES EQUATIONS

Milton E. Rose*

This research investigates weak-element approximations (ref. 1) to mixed initial-boundary value problems for systems in conservation form. Employing linear approximations to the solution of the equation $u_t + f_x(u) = 0$ in local computational subdomains, an implicit system of equations for the flux f results which is solvable by two-point boundary-value methods; the solution u is then explicitly updated by a leap-frog scheme. One class of solution techniques is furnished by the Keller Box-scheme. For a scalar equation in two-dimensions the method has been verified to be second-order accurate using an ADI-type solution method for the flux-equations.

Two methods for extending this approach to the Navier-Stokes equations are being considered. One employs quadratic approximation elements; the other adapts the preceding discussion to a certain canonical form of the Navier-Stokes equations.

References:

1. Rose, Milton E. "Weak-Element Approximations to Elliptic Differential Equations" Numer. Math. 24, 185-204, 1975.

---

*ICASE, 505-31-83-01, 804-827-2513

COMPLETE VISCOUS FLOWFIELD SOLUTIONS ABOUT A BLUNT PARABOLIC BODY IN A SUPERSONIC STREAM

K. J. Weilmuenster and R. A. Graves, Jr.*

The calculation of complete viscous compressible flowfields about blunt bodies in a supersonic stream has been a desirable, yet unreachable, goal of computational fluid dynamics until recent advances in computer hardware provided the necessary tools to attack the problem. There have been many partial and "patchwork" solutions to the problem which tend to apply to specific conditions and which break down when subsonic flow extends from the stagnation point to the aft body corner (sonic-corner flow). Also, these specialized techniques cannot handle low Reynolds number flows where the viscous region is extensive, and where viscous forces predominate. The purpose of the work described here is to demonstrate a general technique for computing flowfields about axisymmetric blunt bodies in a supersonic stream and to use the technique to investigate blunt-body flows over a realistic Mach number/Reynolds number range.

Solutions are obtained using the physical coordinate system shown in figure 1. A complete account of the solution technique can be found in reference 1. Briefly, the flow is divided into two parts, an outer inviscid region over which the Euler equations are integrated, and a viscous region over which the full Navier-Stokes equations are integrated. The two solutions are coupled along the line $\eta = \eta_c$ in figure 1, and are marched together in time to obtain a solution. Solutions are carried out on the CDC-CYBER-203 vector processing computer.

Solutions for the present sonic-corner body have been obtained over the range $2 < M_\infty < 5$ and $500 < R_{e_N} < 2000$ where R_{e_N} is the free stream Reynolds number based on nose radius. The effects of base flow on total drag coefficient was found to be significant at low Reynolds numbers. Also, the growth rate of the base recirculation region has been investigated along with wall pressure and shear stress distributions. In figure 2, is shown a Mach number contour plot which is representative of the flowfields computed in this study.

Reference:

1. Weilmuenster, K. J. and Hamilton, H. H., "A Hybridized Method for Computing High Reynolds Number Hypersonic Flow About Blunt Bodies," NASA TP 1497, October 1979.

---

*SSD, 506-51-13, 804-827-3271

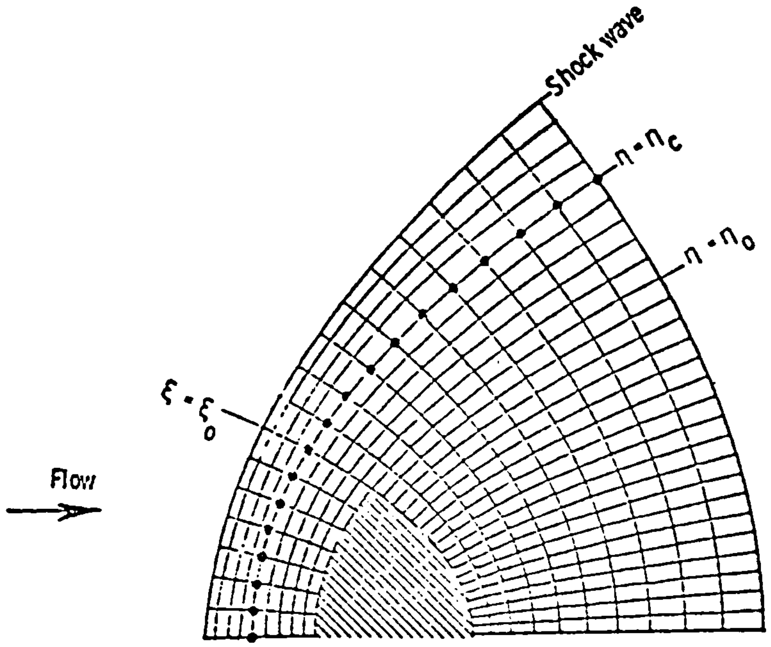

Shock wave
η = η_c
η = η_0
ξ = ξ_0
Flow

Fig. 1 Physical coordinates.

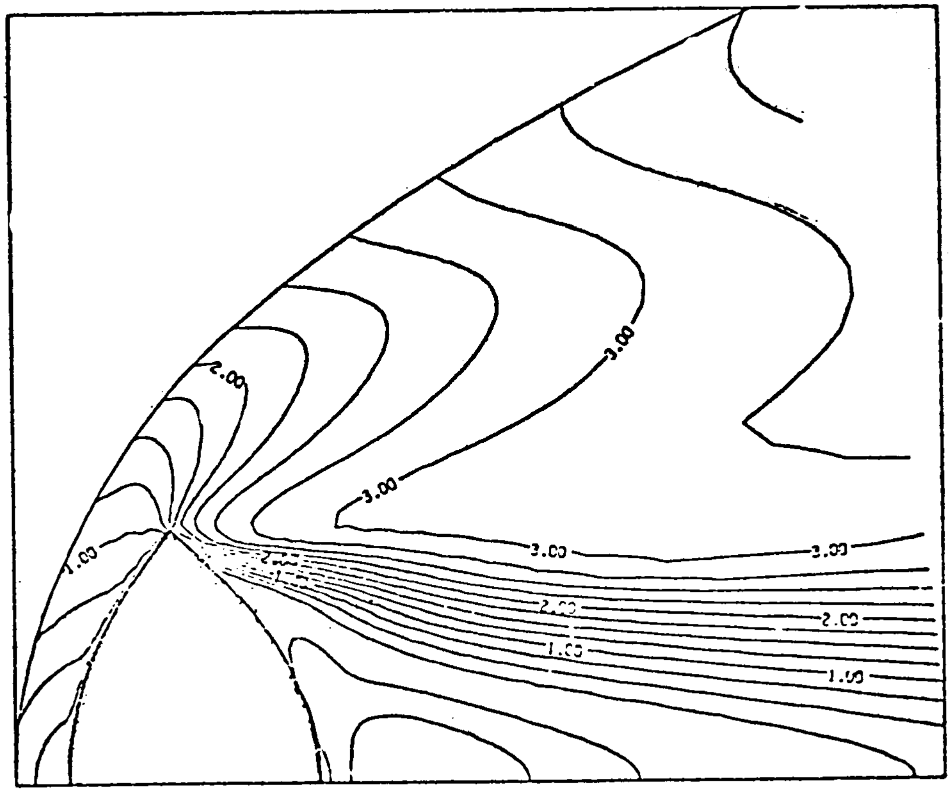


Fig. 2 Mach Number contour plot, $M_\infty = 5$, $Re_N = 2000$.

VISCOUS COMPRESSIBLE FLOW ABOUT BLUNT BODIES USING A NUMERICALLY GENERATED ORTHOGONAL COORDINATE SYSTEM

Randolph A. Graves, Jr.*
H. Harris Hamilton, II

Numerical solutions to the Navier-Stokes equations are obtained for blunt axisymmetric entry bodies of arbitrary shape in supersonic flow. These equations are solved on a finite-difference mesh obtained from a simple numerical technique[1] which generates orthogonal coordinates between arbitrary boundaries. The governing equations are solved in time-dependent form using Stetter's improved stability three-step predictor-corrector method. Since the overall solution is time-dependent, the coordinate system is regenerated at each step as dictated by the shock wave movement. This concept was successfully demonstrated[2] for a large number of inviscid flow calculations. For the present application, the metric coefficients were obtained numerically using fourth order accurate finite-difference relations and proved to be totally reliable for the highly stretched mesh used to resolve the thin viscous boundary layer.

Solutions have been obtained for a range of blunt-body nose shapes including concavities. A number of solutions have been compared with both experimental and computational results with very good to excellent agreement obtained. Figure 1 shows a series of nose shapes seen using the present technique and Figure 2 gives the computed heat transfer for these configurations. Overall results indicate that the numerically generated coordinate system performed exceptionally well and no problems were encountered in the coupling of the numerical coordinate generator and the fluid dynamic equations.

References:

1. Graves, Jr., R. A.: Application of a Numerical Orthogonal Coordinate Generator to Axisymmetric Blunt Bodies, NASA TM 80131, Oct. 1979.
2. Hamilton, H. H. and Graves, Jr., R. A.: Application of a Numerically Generated Orthogonal Coordinate System to the Solution of Inviscid Axisymmetric Supersonic Flow Over Blunt Bodies, NASA TP 1619, Jan. 1980.

---

*SSD, 506-51-13, 202-755-3277

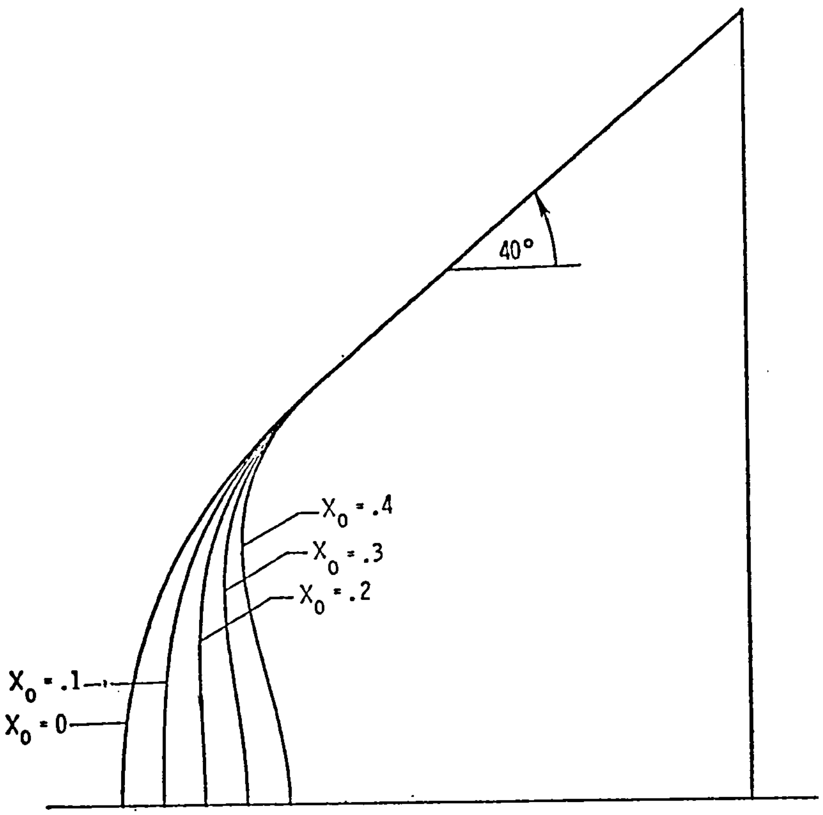

40°
$X_0 = .4$
$X_0 = .3$
$X_0 = .2$
$X_0 = .1$
$X_0 = 0$

Figure 1.- Nose bluntness variations.

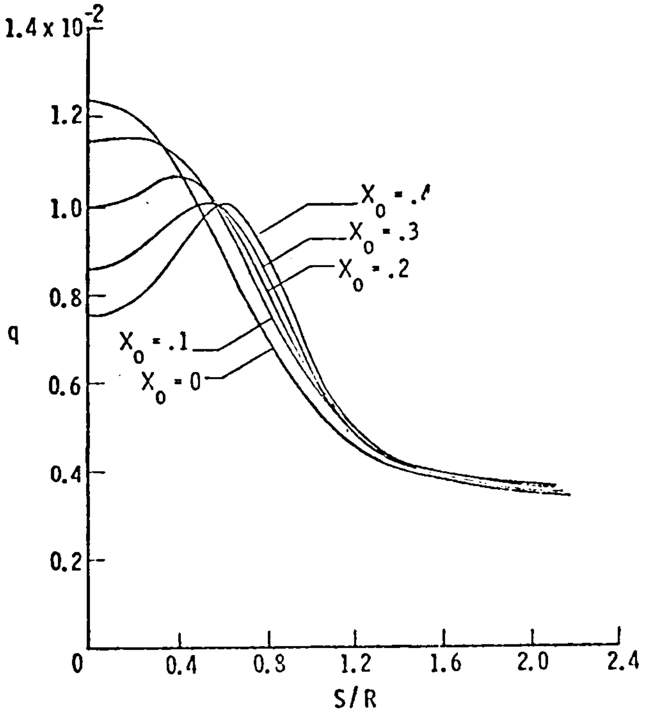


Figure 2.- Heat transfer distributions.

PREDICTION OF 3-D VISCOUS FLOWS IN THE VICINITY OF TRANSONIC WIND-TUNNEL WALL SLOTS

Douglas L. Dwoyer*

In recent years, a study has been underway to understand the detailed flow in the vicinity of transonic wind-tunnel wall slots. The original study was conducted by Dr. Charlie Cooke of ODU and consisted of solving the 2-D Navier-Stokes equations in a plane normal to the wall slot. Qualitative comparisons between the computed results and experimental measurement were good enough to encourage development of a full three-dimensional model.

As a result of the two-dimensional study, a joint effort between the author and Dr. Cooke was begun to computationally model a slotted transonic wind-tunnel test section. The model is based on the compressible split-velocity Navier-Stokes technique in which the total velocity is split into its rotational and irrotational parts. This method has the advantage of allowing the viscous effects to be included in local regions of the flow and having the remainder of the flow governed by the compressible potential equation. A schematic of the computational model is shown in the figure. It is proposed to include the viscous effects only in the immediate vicinity of the slots and treat the flow in the test section and plenum as inviscid.

The computations will be carried out on the CYBER-203 computer and, hence, vector processing compatible numerical algorithms are being used. The compressible potential equation will be solved using the ZEBRA II algorithm (ref. 1) while the viscous equations will be solved by the hopscotch algorithm.

References:

1. South, J. C., Jr.; Keller, J. D.; and Hafez, M. M. "Vector Processor Algorithms for Transonic Flow Calculations" AIAA Paper No. 79-1457, July 1979.

---

*STAD,505-31-13, 804-827-2627

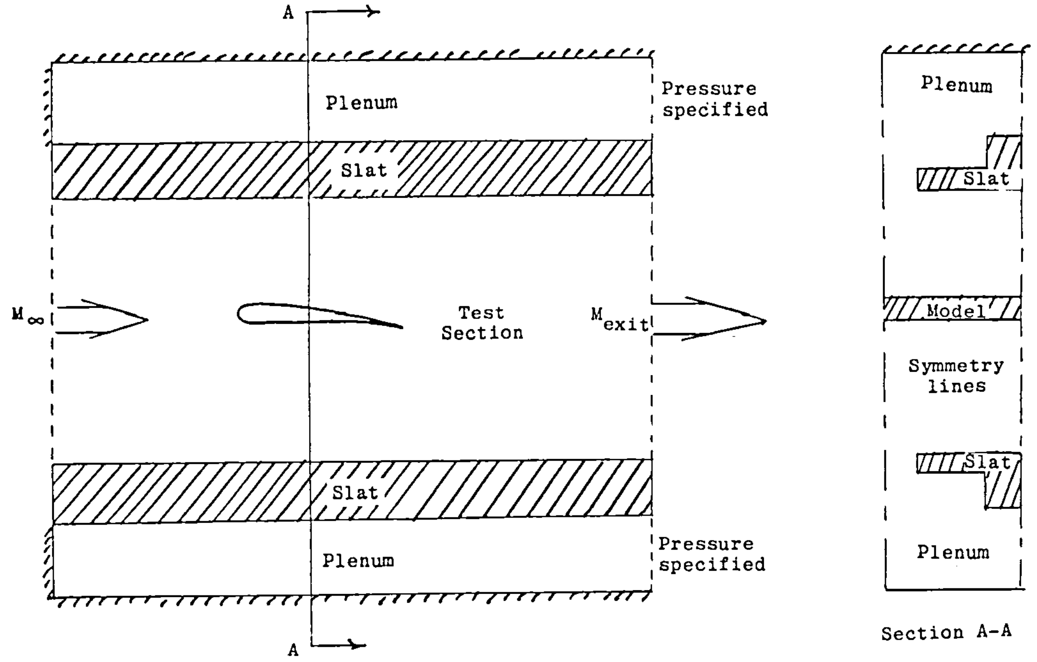

A
Plenum
Pressure specified
Slat
M_∞
Test Section
M_{exit}
Slat
Plenum
Pressure specified
A
Plenum
Slat
Model
Symmetry lines
Slat
Plenum
Section A-A

Figure 1.- Schematic of 3-D slotted wind-tunnel computational model.

PREDICTION OF VISCOUS FLOW IN AERODYNAMIC JUNCTURE REGIONS WITH THE PARABOLIZED NAVIER-STOKES EQUATIONS

Douglas L. Dwoyer*

Understanding of the details of turbulent flows in aerodynamic juncture regions such as wing/fuselage junctions is important in the design of proper fillets. Due to the complexity of such flows, however, conventional boundary-layer methods cannot be used. For this reason, an investigation was begun by Dr. A. J. Baker of CMC, Inc., to use the steady, time-averaged, parabolized Navier-Stokes (PNS) equations to predict such flows. In this work the PNS equations are used to predict the flow in the viscous layer in the corner and are interacted with a three-dimensional inviscid calculation in order to account for the global effect of the corner flow.

In this research, Dr. Baker has formulated the appropriate governing equations, developed a suitable turbulence model, and devised an interaction procedure. The governing equations are the steady time-averaged compressible Navier-Stokes equations with streamwise diffusion terms neglected and with the pressure gradient vector split into a streamwise component which is imposed from the inviscid solution and a component which corrects the normal pressure gradient which is calculated in the PNS solution. The turbulence model is based on a parabolized form of the turbulence kinetic energy equation.

Shown in the figure is a preliminary calculation showing the cross-flow velocity vector at the 12-percent chord station in the junction of two parabolic arc airfoils. These data were plotted from the first viscous/inviscid interaction. The beginnings of the rollup of the corner vortices can be seen.

---

*STAD, 534-02-13, 827-2627

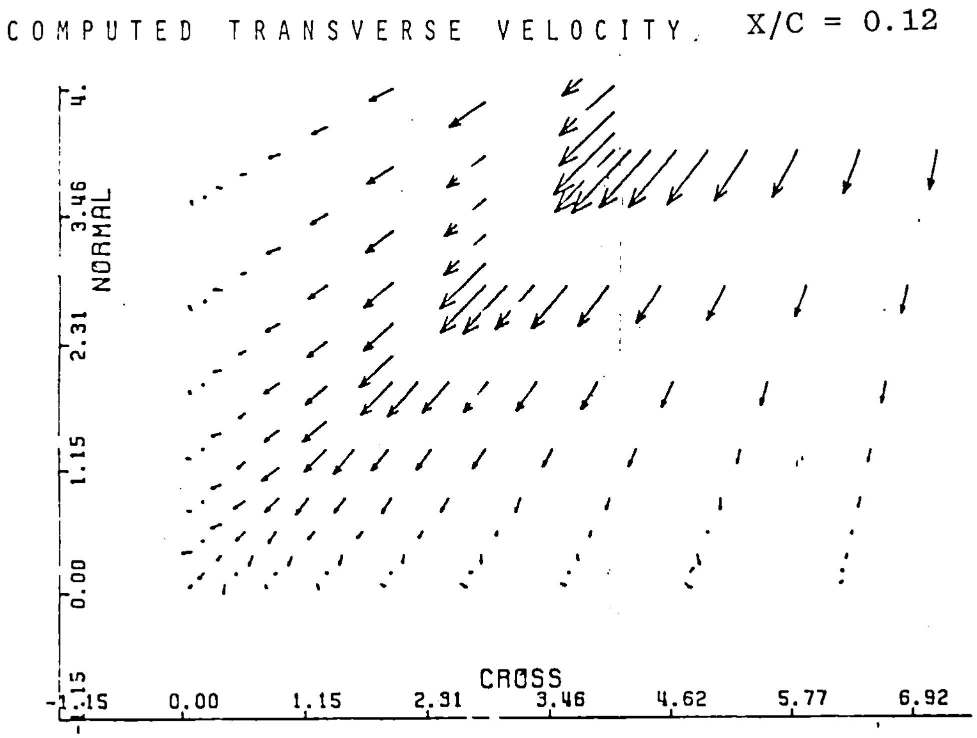


Figure 1.- Parabolic arc juncture region flow

IMPLICIT METHODS FOR THE NAVIER-STOKES EQUATIONS

Stephen F. Wornom*

Implicit factored methods, when applied to the Navier-Stokes equations, converge very rapidly as long as the flow is predominantly viscous. The case of a shock-wave interacting with a boundary layer on a plate is a good example. Results obtained via the method reported in reference 1 show acceptable convergence in 100 time steps with $u_\infty \Delta t/\Delta x = 100$ during which the residual error is reduced 4 orders of magnitude. On the other hand, for flows that are not viscous dominated, the convergence rate is much slower. Consider flow over a 2-D airfoil using the Euler equations. Results obtained using the method of reference 2 are shown in figure 1 where it can be seen that acceptable convergence has not been reached after 300 time steps with $u_\infty \Delta t/\Delta x = 5$ in which the residual error was reduced one order of magnitude.

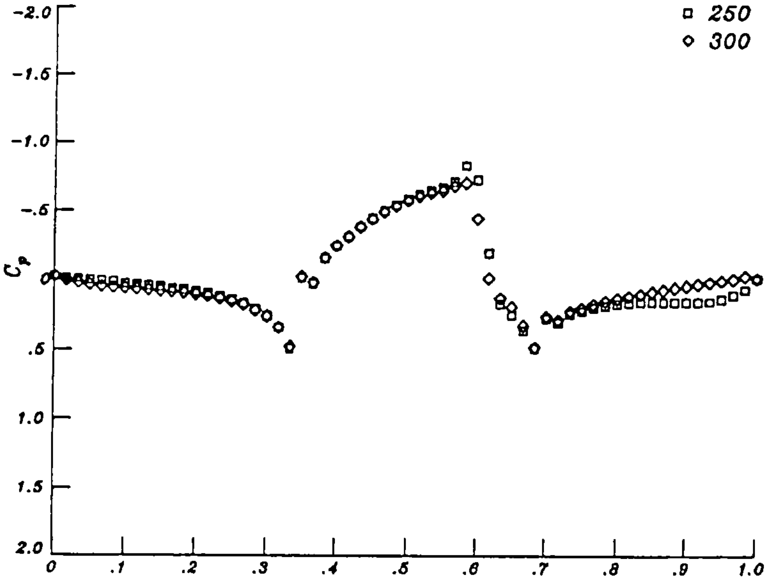
The purpose of this research is to develop an implicit two-point method for the Euler equations which avoids two weak points of the three-point method used in reference 2. The weak points of reference 2 are: (1) The difference scheme, when applied to the Euler equations, weakly couples the odd and even points and as such tends to produce oscillatory results and (2) the resulting difference equations require more boundary conditions than the P.D.E.'s being solved. These improvements are part of the overall research goal to develop an efficient method for solving the Euler equations and, likewise, the Navier-Stokes equations. The related research areas are the multi-grid and flux-splitting methods.

References:

1. Beam, Richard M.; and Warming, R. F. "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations" AIAA J., vol. 16, no. 4, April 1978, pp. 393-402.
2. Beam, Richard M.; and Warming, R. F. "An Implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation-Law Form" J. Comp. Physics, vol. 22, 2976, pp. 87-110.

---

*STAD, 505-31-13, 804-827-2627



COMPUTATIONAL COORDINATE

Figure.- Chordwise pressure coefficient on a parabolic airfoil, M = 0.83.

VECTOR COMPUTER APPLICATIONS SOFTWARE FOR THE SOLUTION OF THE COMPRESSIBLE NAVIER-STOKES EQUATIONS

Robert E. Smith and Joan I. Pitts

A three-dimensional Navier-Stokes solver for compressible laminar flow based on a proven numerical technique (MacCormack time-split algorithm) has been developed for the CYBER 203 computer. This explicit technique is used because of its robust characteristics and because it conforms to the CYBER 203 architecture. There is a large data base consisting of five state variables and nine variables for geometric description at each grid point. The virtual memory of the CYBER 203 requires that considerable attention be given to program organization. The data base has been interleaved using a four-dimensional array and intermediate storage is used to apply the finite difference algorithm (fig. 1).

In developing the Navier-Stokes solver, primary consideration has been given to the capability to solve a wide class of problems with a minimum of programing changes. This has been accomplished by programing the complete transformed equations of motion including all nine elements of the Jacobian matrix (metric data). Supplying the metric data from a grid generation technique and programing the boundary conditions for a given problem (separate subroutine, fig. 2) allows virtually any laminar fluid flow problem to be attacked. Experience, however, indicates that supersonic flow is most feasibly simulated with the MacCormack technique.

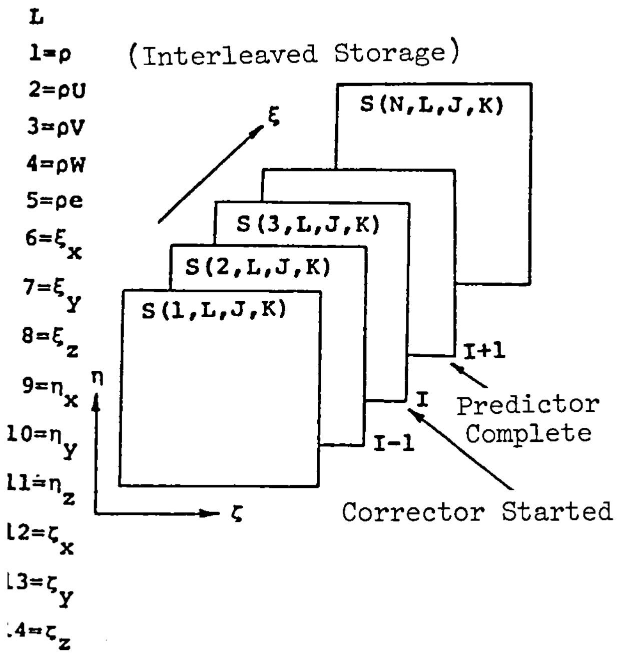
The Navier-Stokes solver is written in the SL/1 programing language and uses 32 bit arithmetic. There are approximately two million words of incore memory and a computational rate of 5.5×10^{-5} seconds per grid point per time step is observed. This is 26 times faster than the same technique written for the CDC 7600. References 2 and 3 present solutions obtained with the Navier-Stokes solver. No significant accuracy loss has been observed using the 32 bit word size. The program is independent of the grid generation technique and the metric data from any technique can be used by the program.

REFERENCES

1. Smith, R. and Pitts, J.: "The Solution of the Three-Dimensional Viscous Compressible Navier-Stokes Equations on a Vector Computer," Third IMAC International Symposium on Computer Methods for Partial Differential Equations, Lehigh University, PA, June 1979.

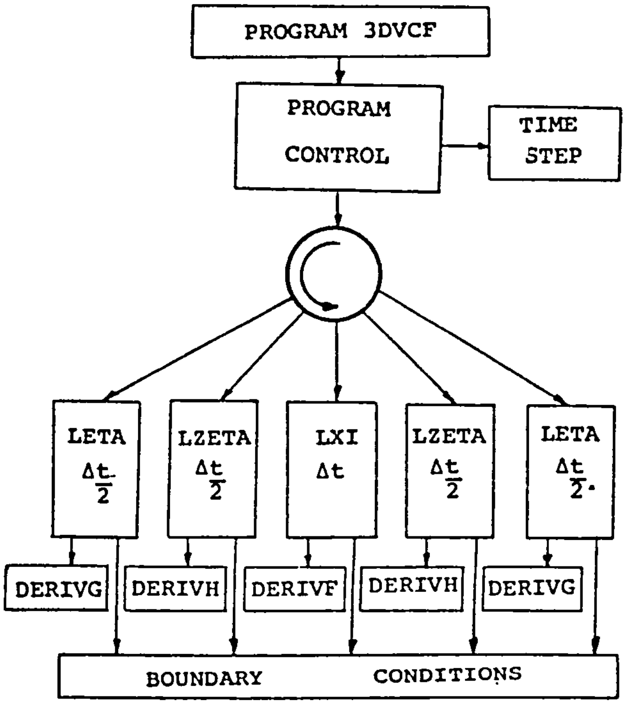
2. Shang, J. S.; Hankey, W. L. and Smith, R. E.: "Flow Oscillations of Spike-Tipped Bodies," AIAA paper 80-0062, January 1980.

3. Smith, R. E.: "Numerical Solutions of the Navier-Stokes Equations for a Family of Three-Dimensional Corner Geometries," AIAA paper 80-1349, July 1980.



L
1=ρ
2=ρU
3=ρV
4=ρW
5=ρe
6=ξx
7=ξy
8=ξz
9=ηx
10=ηy
11=ηz
12=ζx
13=ζy
4=ζz
(Interleaved Storage)
ξ
S(N,L,J,K)
S(3,L,J,K)
S(2,L,J,K)
S(1,L,J,K)
η
ζ
I+1
I
I-1
Predictor Complete
Corrector Started

Figure 1 - Data Organization



PROGRAM 3DVCF
PROGRAM CONTROL
TIME STEP
LETA Δt/2
LZETA Δt/2
LXI Δt
LZETA Δt/2
LETA Δt/2
DERIVG
DERIVH
DERIVF
DERIVH
DERIVG
BOUNDARY CONDITIONS

Figure 2 - Navier-Stokes Solver Flow Chart

FINITE ELEMENT TECHNIQUES FOR HIGH REYNOLDS NUMBER FLOW PREDICTION

A. J. Baker*

A particularly difficult problem class in computational fluid dynamics is flowfield prediction at large Reynolds number. The governing Navier-Stokes equations are strongly non-linear elliptic boundary value description, but the importance of the viscosity term can be completely dwarfed by the non-linear convection terms everywhere away from a wall. Furthermore, the continuity equation possesses no viscosity-like term, hence uniformly exhibits the hyperbolic description pervading the entire flowfield region.

The technical goal of this project is to extend the developed finite element tensor product numerical algorithm to solution of systems of equations, in particular the multi-dimensional Navier-Stokes equations for compressible flow at large Reynolds number. The primary requirement is to assess formation and solution of the tensor matrix product non-linear algebraic equation systems produced by this algorithm, with regards to solution stability, accuracy and speed. A second requirement regards evaluation of a non-iterative algorithm, with its additional core requirements but reduced number of algorithm sweeps. Thirdly, is it plausible to utilize multi-dimensional finite element matrices for formation of the non-linear algebraic equation system, while employing tensor products only for the Jacobian of the Newton iteration algorithm.

The basic algorithm statement employs interpolation and a weighted residuals formulation to approximate the Navier-Stokes system. Letting $L(q)$ and $\ell(q)$ denote respectively the governing equation and the corresponding boundary conditions, where $q = (\rho, \rho u_i)$ is a generalized dependent variable, the approximate solution $q^* \equiv \sum_{e=1}^{M} \{N\}^T\{Q\}_e$ is formulated by requiring the error to be orthogonal to the space of functions $\{N\}$ used for interpolation on the finite element domain R_e^n, i.e.

$$\int_{R^n} \{N\} L(q^*) + \vec{\beta} \cdot \int_{R^n} \{N\} \nabla L(q^*) + \lambda \int_{\partial R} \{N\} \ell(q^*) \equiv \{0\} \qquad (1)$$

Equation (1) is a system of non-linear ordinary differential equations, and the term modified by $\vec{\beta}$ introduces a dissipation mechanism for error control. Employing the trapezoidal integration rule produces a non-linear algebraic equation system $\{F\}$, the solution of which using Newton iteration is of the form

$$[J]\{\delta Q\} = -\{F\} \qquad (2)$$

where $\{\delta Q\}$ is the iteration vector and $[J]$ the Jacobian of $\{F\}$.

The emphasis in the first phase of this project, initiated 1 March 1980, is to construct regular and tensor product forms for $\{F\}$, using both the conservative and non-conservative equation forms, and to compare algorithm performance. The Jacobian is always formed using matrix tensor products for solution economy, and its formulation is invariant for the non-iterative algorithm as well. The various matrix forms have been derived and the corresponding computer code modifications are underway. Preliminary results for the rotating cone test case, corresponding to solution of the continuity equation alone, indicates the iterative full matrix $\{F\}$ solution form, and the approximate tensor product $\{F\}$ form, in essential agreement. The former required four iterations/step for convergence to 10^{-4} using the tensor product Jacobian. The complete approximate tensor product form is about a factor of four faster on the grid sweeps. The next step is to assess formulations of the exact $\{F\}$, to improve the solution economy for application to the complete Navier-Stokes system. This will primarily involve the approximation to the initial-value term, which appears the key to solution accuracy.

General Aerodynamic Simulation
Three-Dimensional, Compressible, Navier-Stokes Equations

Julius E. Harris*

Substantial progress has been made over the past decade in numerically solving the Navier-Stokes equations. Recent developments in coordinate generation procedures, for example, boundary-fitted approaches have allowed solutions for systems that could not be previously obtained with conventional Cartesian grids. The boundary-fitted approach has also been extended to three-dimensional flows for cases where two-dimensional boundary-fitted regions are joined by piecewise analytic functions to bound the three-dimensional computational domain. Most existing programs treat only perfect-gas, laminar flows; however, some progress has been made in implementing turbulence closure for pressure driven (boundary-layer like) flows. With the exception of a few specialized programs (NASA ARC/ILLIAC IV), almost all software to date has been developed for scalar-mode processing on systems such as the CDC-7600.

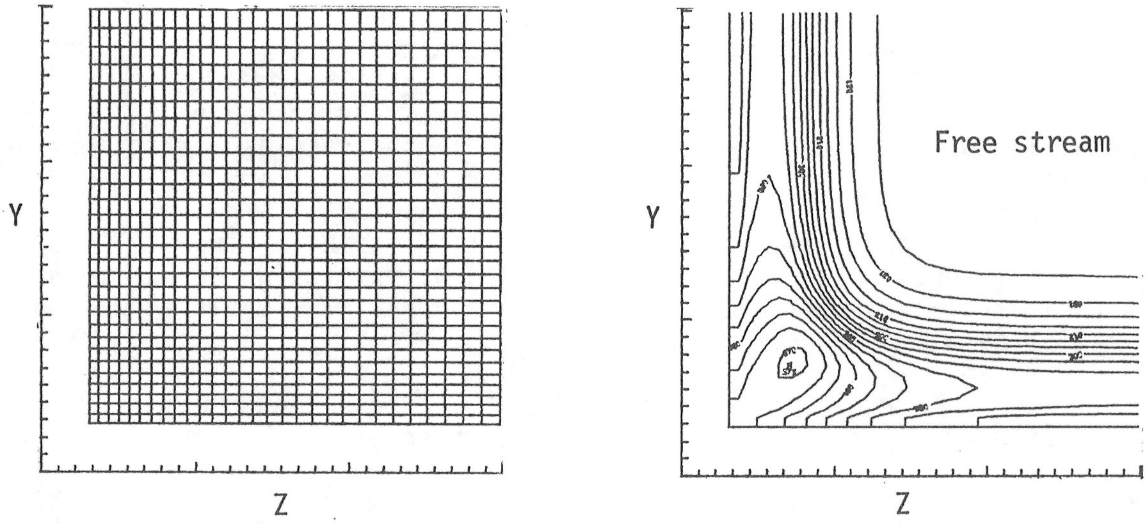
The introduction of advanced computer architecture (for example; CYBER-203 at LaRC) presents the challenging opportunity to develop software that could take full advantage of the increased processing speed associated with vector-pipeling processing as well as virtual memory storage. Accordingly, an in-house program was initiated in the following areas: (1) development of an algorithm for processing on the CYBER-203; (2) algorithm structure to minimize page-fault penalties associated with virtual memory to maximize processing speed; (3) development of a data base management procedure to allow maximum use of all computer software/hardware procedures at LaRC; (4) development and/or utilization of a general mesh generation program external to the algorithm module; (5) implementation of turbulence closure (algebraic to multi-equation systems). To date, goals (1) to (3) have been achieved, progress has been made on goal (4) and equation systems have been developed for goal (5).

An explicit, time-split algorithm was developed and programmed as a stand-alone modular package. Modifications occur only in the modules that treat boundary and initial conditions and grid generation. Variable dimensions are used to minimize computer resource requirements. Page-fault penalties are minimized through data base interleaving and vector turn-over as the operators sweep through the data base. Flow visualization includes line, contour, surface, and color plot capabilities.

As a first series of test cases the program was applied to supersonic flow past sharp and filleted corners. The corner fillets were chosen to be circular arcs. Numerical results are presented in Figures 1 and 2. Calculations are now being made to determine the effect of discontinuities in the second derivative of wall shape on the vorticity field. Current emphasis is focused on developing/incorporating a more general procedure for coordinate generation.

---

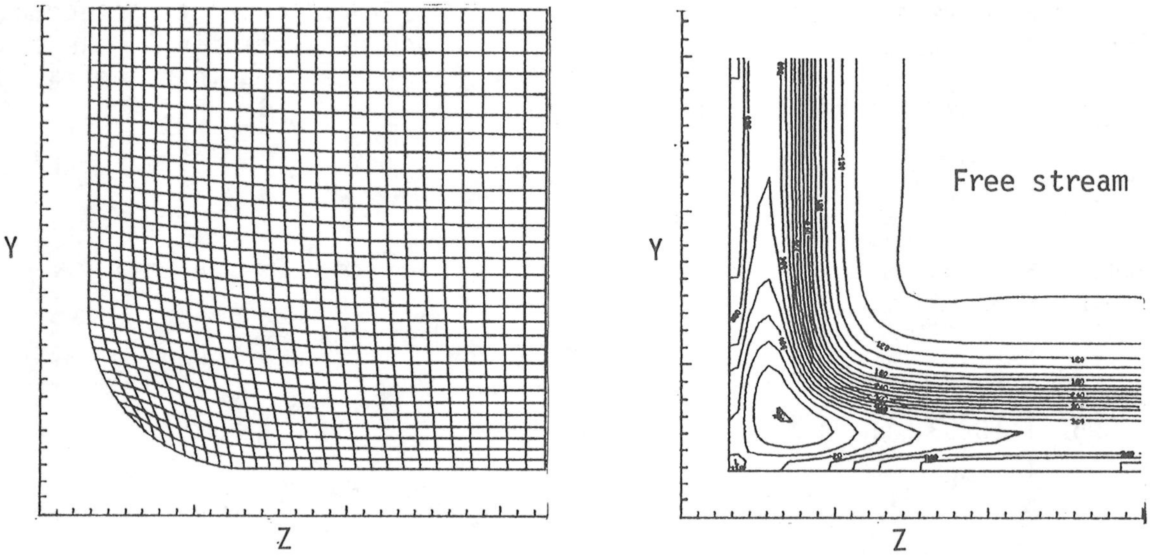
*HSAD, 505-31-13-03, (804) 827-4546



(a) Grid

(b) Static pressure

Figure 1 - Sharp corner flow $M_\infty = 3$; $N_{R_\infty} = 10^3$ ft. $^{-1}$; $T_w = T_{aw}$



(a) Grid

(b) Static pressure

Figure 2 - Circular fillet corner flow $M_\infty = 3$, $N_{R_\infty} = 10^3$ ft. $^{-1}$;$T_w = T_{aw}$

UNSTEADY OPEN CAVITY FLOW/TWO-DIMENSIONAL INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Joshua C. Anyiwo*

A computer code is being developed to numerically study unsteady, incompressible shear flow over a series of open cavities of arbitrary aspect ratios and spacing (Fig. 1).

The computer code processes the vorticity equation (using a simple ADI method), and the stream function and pressure Poisson equations (using an SOR method), Reference 1. For generality all velocities are non-dimensionalized with u_τ and all lengths with Δ where:

u_τ = (wall shear stress/fluid density)$^{1/2}$, at some convenient reference position; here, the inflow plane;

$\Delta = k \{\Sigma A_i\} \{\Sigma (\ell_i + w_i)\}; \; i = 1, 2, 3 \ldots$

A_i = aspect ratio of i-th cavity

ℓ_i = width of i-th cavity

w_i = spacing between i-th and (i + 1)-th cavities

k = arbitrary numerical constant

Particular attention will be directed towards the effects of vortex shedding and inflow velocity profile on the general evolution of vorticity in the flow system. The possibility of applying these results to studies in potential drag reduction experiments will be explored.

Reference:

1. Numerical Studies of Incompressible Viscous Flow in a Driven Cavity. NASA SP-378, 1975.

---

*HSAD 505-31-13-03, (804) 827-2806

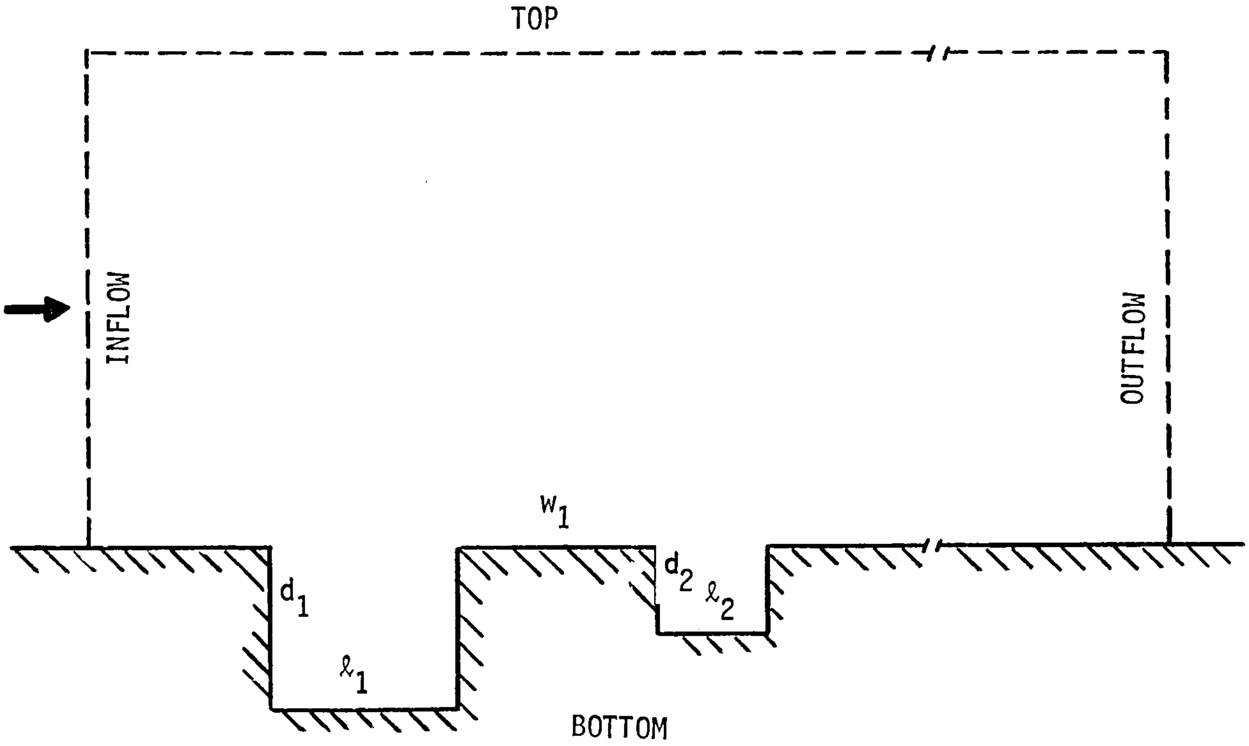

TOP
INFLOW
OUTFLOW
w_1
d_1
ℓ_1
d_2
ℓ_2
BOTTOM

Figure 1. Schematic of geometry

A TWO-DIMENSIONAL NAVIER-STOKES SOLVER USING SPECTRAL METHODS FOR FLOWS OVER MOVING WALL GEOMETRIES

R. Balasubramanian and Steven A. Orszag*

The study of mechanisms of possible drag reduction effects in flows over moving wavy walls as well as the study of mechanisms of generation of surface waves by wind requires understanding of the detailed flows over wavy surfaces. We have developed a computer code based on spectral methods for the study of two-dimensional incompressible flows in wavy geometries. The full time-dependent two-dimensional Navier-Stokes equations are solved using spectral methods to achieve high spatial accuracy and high-order time-splitting methods together with conformal mapping methods to allow simulation of flow over steep waves.

Applications have been made to laminar flows over wavy surfaces. Some experimental results have been given by Kachanov Kozlov, Kotjolkin, Levchenko and Rudnitsky at a Reynolds number of 1.1×10^5. The results obtained using our code are in excellent agreement with experiment.

In addition comparison of the results of the computer codes with the theoretical predictions of Benjamin show that the theoretical analysis is valid for small values of ka and δ/a, where a is the amplitude of the wall displacement, k is its wavenumber, and δ is the boundary layer thickness.

Our main motivation for the present calculations is to study the possibility for drag reduction in flow over non-sinusoidal periodic surfaces. The total drag for the surfaces are compared to the total drag over a sinusoidal surface of the same wave length and amplitude.

We have also studied the laminar flow over a localized bump. The results are compared with the results obtained by Taylor and Gent. This study should be useful in understanding the effect of local roughness on flows.

In the area of turbulent flows, the computer codes have been used to study (1) channel flows with one wall of the channel replaced by a wavy surface and (2) boundary layer flows over a wavy wall. One of the key problems addressed in this study is the kind of turbulence modeling necessary for proper simulation of turbulent flow over wavy walls.

For the boundary layer flow problem, both curvature and pressure gradient effects are of importance. Results are obtained using the Beckwith-Bushnell model. Current work is being done to improve the turbulence modeling in the boundary layer case.

---

*Cambridge Hydrodynamics, Inc.

TIME-DEPENDENT COORDINATE SYSTEMS FOR THE NUMERICAL SOLUTION OF FLUID FLOW

Liviu Lustman*

Powerful algorithms have been used recently to map complicated geometrical regions over standard squares or cubes. In the corresponding new coordinates, uniform grids may be used efficiently, as well as techniques necessitating simple geometry, such as spectral methods. However, the grid generation is usually effected only once, before starting the solution of the differential system. The coordinates thus obtained are appropriate for boundary layer resolution or sharp corners, but are essentially static and probably not applicable to time-dependent problems. It is a serious challenge to design a time-dependent grid, which will map a uniform computational mesh, at various times, onto surfaces in the physical domain fitted to wave phenomena - sharp gradients or shock discontinuities propagating in space. Proper bunching of these curvilinear coordinates may solve the shock resolution problem, since - as numerical experience shows - Gibbs phenomena and shock smearing usually occurs not on a fixed length scale, but on a fixed number of mesh points. Clearly, such an ambitious project must overcome several difficulties:

(1) Grid generation is time consuming, if done by the elliptic system method of Thompson et al. Simpler geometrical methods are hard to match to flow properties in the physical domain.

(2) After being expressed in the new coordinates, the transformed equations will contain new terms due to coordinate evolution, posing stability and accuracy problems.

However, recent results of K. Miller and R. N. Miller show that a very simple node algorithm - which is moreover independent of the physical system to be solved - is quite efficient in one-dimensional problems.

I propose to study similarly conceived algorithms for sample two-dimensional computations, in order to derive a gradient sensitive, time-dependent coordinate generator, and to test its usefulness for various numerical procedures.

---

[1] Miller, Keith, Miller, Robert N., "Moving Finite Elements," to appear.

[2] Eisemam, Peter,; J.C.P. 26(1978)307, J.C.P. 33(1979)118

[3] Thompson, J., Thames, F., Mastin, C.; J.C.P.15(1974)299, J.C.P.24 (1977) 274, NASA-CR 2729.

*Old Dominion University

PREDICTION OF THE TRANSONIC FLOW OVER AXISYMMETRIC BOATTAIL NOZZLES USING A NAVIER-STOKES CODE DEVELOPED FOR THE STAR COMPUTER

R. C. Swanson*

The time-dependent Navier-Stokes equations in mass-averaged variables are solved for transonic flow over axisymmetric boattail-plume simulator configurations. Numerical solution of these equations is accomplished with the explicit finite difference algorithm of MacCormack. A grid subcycling procedure and computer code vectorization are used to improve computational efficiency. The two-layer algebraic turbulence models of Cebeci-Smith (C-S) and Baldwin-Lomax (B-L) are employed for investigating turbulence closure. Two relaxation models based on these baseline models are also considered.

In this study solutions for the Reubush (Ref. 1) configuration 1 afterbody, which has a boattail terminal angle of 34 degrees, have been computed. For all cases the flow on the boattail is highly separated. In figure (1) the variation of the surface pressure coefficient C_p with the nondimensionalized axial distance z/D_E at the Mach numbers of 1.3 and 0.8 is shown. Results obtained with each of the four eddy viscosity models are given. The predictions with the base-line turbulence models show poor agreement with experiment. This is not surprising since these models do not account for upstream history effects. That is, if the flow is disturbed in some way (i.e., severe adverse pressure gradient) the turbulence field does not respond immediately. However, it does retain memory of the event. The relaxation models attempt to account for this delayed response through a relaxation length scale (λ). The Relax (C-S) model solutions compare well with the data except for the separated flow region. In both the supersonic and subsonic cases the results determined with the Relax (B-L) model capture the pressure plateau region. These pressure distributions suggests that the relaxation turbulence models have potential to accuratley predict such boattail flows.

Reference:
Reubush, D. E., "Experimental Study of the Effectiveness of Cylindrical Plume Simulators for Predicting Jet-On Boattail Drag at Mach Numbers Up to 1.30," NASA TN D-7795, November 1974.

---

*HSAD, 505-32-13, 804-827-2673

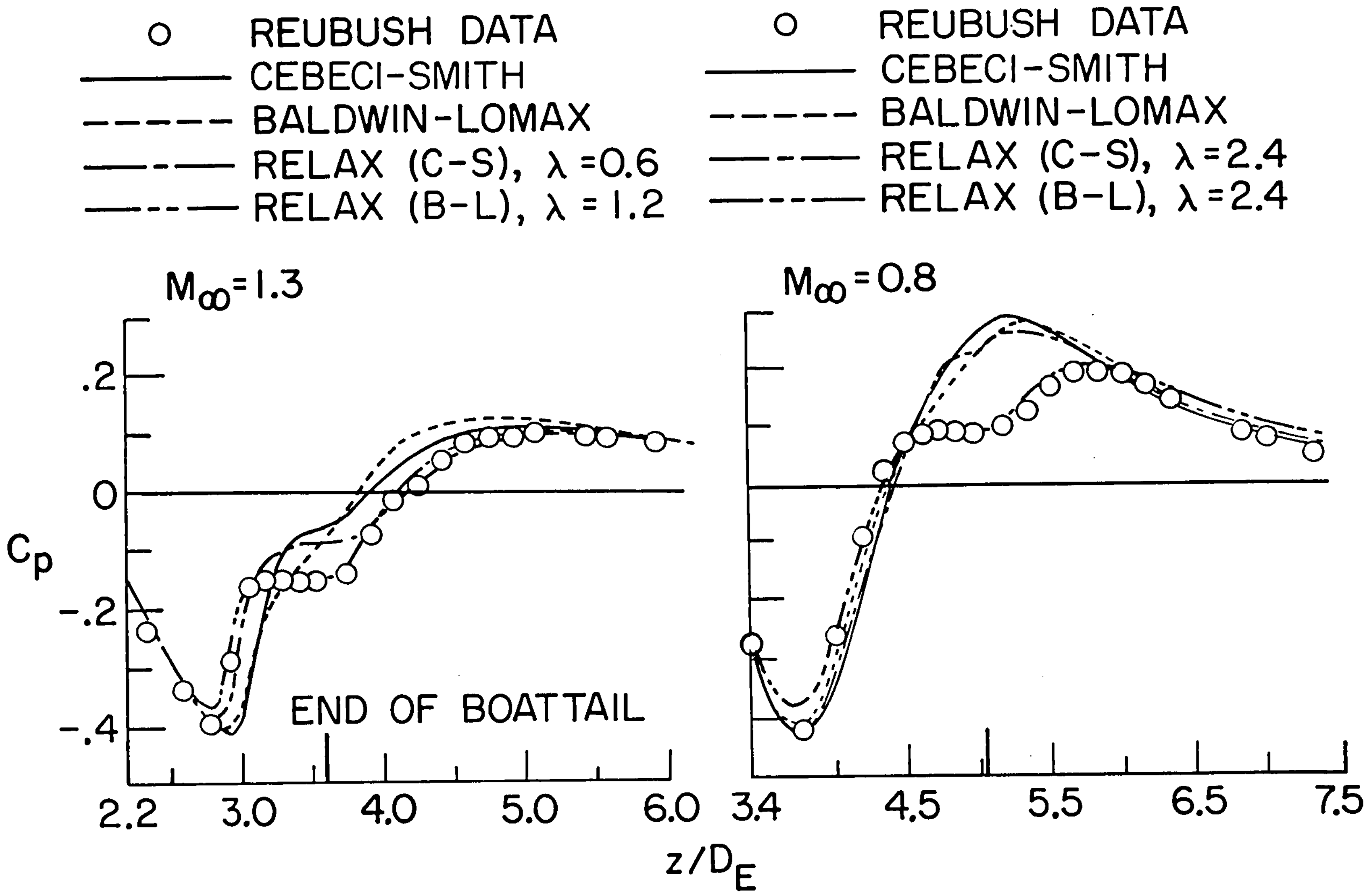


Figure 1.- A comparison of pressure distributions using algebraic Eddy viscosity models, configuration 1.

SPLINE COLLOCATION FOR VISCOUS FLOW OVER AFTERBODY CONFIGURATIONS

S. J. Rubin* and R. C. Swanson**

For a number of years solutions for inviscid transonic flow have been obtained efficiently by solving a potential flow equation with some type of relaxation procedure. Also, the viscous regions of such flows have been determined with efficient solvers of the boundary-layer equations. In this work some of the principles of these techniques are combined to solve the Navier-Stokes equations.

The subsonic compressible Navier-Stokes equations are solved using a velocity field decomposition procedure. That is, the streamline velocity component is represented by a sum of potential and viscous flow contributions. In this way the boundary-layer and potential flow behaviors are recovered in their respective regions of validity. The fully coupled flow equations, which are written in a body-fitted coordinate system, are solved with Stone's strongly implicit method. The time derivative merely provides an artiface for obtaining the steady state solution. In incompressible computations, both unseparated and separated, with this method (Ref. 1) sufficiently large time steps have been achieved to collapse this procedure to a relaxation process. This characteristic is an objective in the compressible calculations of this effort. To further enhance computational efficiency or to increase numerical accuracy a higher-order spline interpolation deferred corrector is incorporated.

At this point compressible attached flow solutions have been determined for a boattail like configuration. Work is in progress to compute separated flows over such geometries.

References:

Rubin, S. G. and Khosla, P. K., "Navier-Stokes Calculations With a Coupled Strongly Implicit Method - Part 1: Finite-Difference Solutions," AIAA Paper 79-0011, January 1979.

---

*University of Cincinnati
**HSAD, 505-32-13, 804-827-2673

A TWO-STREAM SOLUTION OF THE NAVIER-STOKES EQUATIONS FOR UNINSTALLED NOZZLES

Michael C. Cline* and Richard G. Wilmoth**

The need for a fully viscous solution technique for the prediction of nozzle afterbody-exhaust jet interactions at subsonic and transonic speeds has been amply demonstrated in recent years. The purpose of this work is to develop a two-stream solution technique for the two-dimensional, compressible Navier-Stokes equations which can be applied to a wide variety of uninstalled nozzle flow problems. This technique will provide not only a basic predictive tool but will also be used to guide the development of flow-field models for the more approximate patched predictions.

A two-stream, Navier-Stokes computer code (VNAP2) has been developed as an extension of a code developed mainly for treating nozzle internal flows (ref. 1). The time-dependent flow equations are marched to steady state in the computational domain shown in figure 1 using the explicit, two-step MacCormack algorithm. Variable grid spacing in both the physical x- and y-directions is used to resolve boundary layers and shear layers. The principal features of the computer program are listed in figure 2. A special quick-solver technique is also provided to reduce the computational time needed in regions of small y-grid spacing (such as near walls and shear layers).

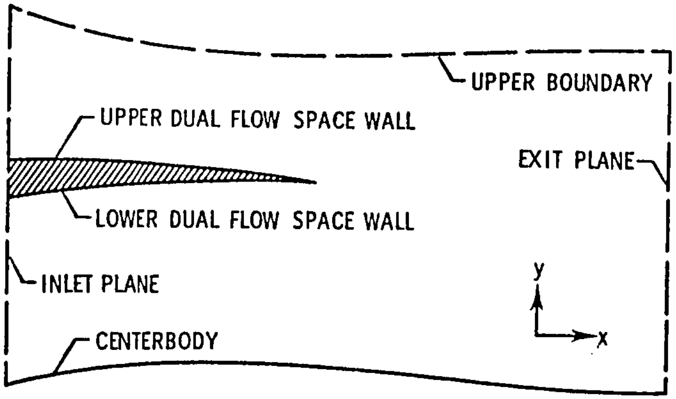
The program is currently being applied to the calculation of subsonic external flow-supersonic exhaust jet problems for which an extensive experimental data base exists.

References:

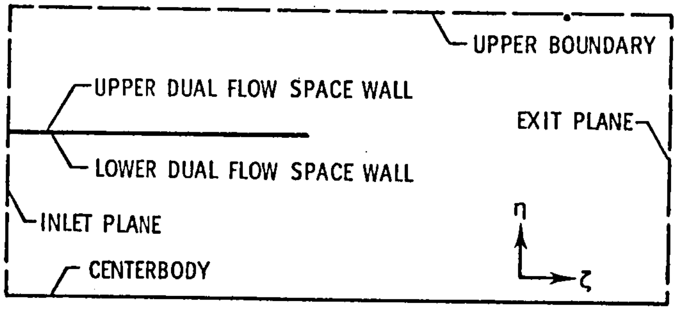
1. Cline, Michael C.: VNAP: A Computer Program for Computation of Two-Dimensional, Time-Dependent Compressible, Viscous, Internal Flow. Los Alamos Scientific Laboratory Rept. No. LA-7326, 1978.

---

* Los Alamos Scientific Laboratory
** HSAD, 505-32-13, 804-827-2675


UPPER BOUNDARY
UPPER DUAL FLOW SPACE WALL
EXIT PLANE
LOWER DUAL FLOW SPACE WALL
INLET PLANE
y
x
CENTERBODY

PHYSICAL SPACE


UPPER BOUNDARY
UPPER DUAL FLOW SPACE WALL
EXIT PLANE
LOWER DUAL FLOW SPACE WALL
η
INLET PLANE
CENTERBODY
ζ

COMPUTATIONAL SPACE

Figure 1.- Physical and computational flow spaces.

EQUATIONS OF MOTION:

2-D (PLANAR OR AXISYMMETRIC), COMPRESSIBLE, TIME-DEPENDENT NAVIER-STOKES FOR TURBULENT FLOW
SINGLE-COMPONENT IDEAL GAS

TURBULENCE MODEL OPTIONS:

MIXING LENGTH
1-EQUATION TURBULENT KINETIC ENERGY (TKE)
2-EQUATION (TKE AND DISSIPATION RATE)

SHOCK TREATMENT:

SHOCK SMEARING WITH ARTIFICIAL VISCOSITY ADDED

NUMERICAL TECHNIQUE:

INTERIOR - TWO-STEP MacCORMACK FINITE DIFFERENCE SCHEME
BOUNDARY - SECOND-ORDER, REFERENCE-PLANE CHARACTERISTIC SCHEME
VARIABLE GRID - USED TO RESOLVE BOUNDARY LAYER AND SHEAR FLOW REGIONS

INITIAL CONDITIONS:

SPECIFY u, v, ρ, p AT ALL MESH POINTS
OR COMPUTE INITIAL CONDITIONS BASED ON 1-D FLOW THEORY

BOUNDARY CONDITIONS:

INLET - SPECIFY p_t, T_t, θ, OR u, v, ρ
EXIT - SPECIFY p
UPPER BOUNDARY (WALL) - CAN BE SOLID WALL OR CONSTANT PRESSURE INFLOW-OUTFLOW
ALL SOLID WALLS - CAN BE FREESLIP OR NOSLIP

Figure 2.- Principal features of viscous flow computer program.

A 3-D SOLUTION OF THE NAVIER-STOKES EQUATIONS FOR THE FLOW CHARACTERISTICS AND PERFORMANCE OF NONAXISYMMETRIC NOZZLES

P. D. Thomas* and Lawrence E. Putnam**

Recent experimental studies have shown that the nonaxisymmetric nozzle properly integrated into twin-engine fighter-aircraft configurations has the potential for reducing drag, increasing instantaneous maneuverability and subsonic agility, providing STOL capability, and reducing fabrication and maintenance costs (ref. 1). To date, the design and integration of nonaxisymmetric nozzles have been based, primarily, on semi-empirical techniques and experience. A solution technique for the 3-D Navier-Stokes equations has, therefore, been developed (ref. 2) to predict the performance and characteristics of the viscous flow field for uninstalled 3-D nozzle configurations at subsonic, transonic, and supersonic speeds.

The 3-D time-dependent Navier-Stokes equations are solved using the Beam and Warming alternating-direction implicit numerical technique. Boundary conditions are computed with an implicit technique compatible with the ADI technique employed at interior points of the flow region. The equations are formulated and solved in a boundary-conforming curvilinear coordinate system. The curvilinear coordinate system and computational grid is generated numerically as the solution to an elliptic boundary valve problem. A new method has been developed that automatically adjusts the elliptic system so that the interior grid spacing is controlled directly by the user defined grid spacing on the boundary of the flow region. The viscous terms are modeled using a modification of the Baldwin and Lomax method.

A comparison of predicted and experimental surface pressures on the center line of a flap of a convergent-divergent rectangular nozzle operating at a nozzle pressure ratio of about 3 is shown in figure 1. For these calculations, the flow was assumed to be two-dimensional. The computational grid used is shown in figure 2 and a density contour plot is presented as figure 3. The predicted surface pressure are in very good agreement with the experimental data. The theory smears the shock located near the throat of the nozzle with a resultant decrease in prediction accuracy. Further studies are underway to determine the capabilities of the solution technique for predicting the flow characteristics of 3-D nozzle configurations.

References:
1. Capone, Francis J.: The Nonaxisymmetric Nozzle-It is For Real. AIAA Paper No. 79-1810, Aug. 1979.
2. Thomas, P. D.: Numerical Method For Predicting Flow Characteristics and Performance of Nonaxisymmetric Nozzles-Theory. NASA CR-3147, 1979.

---

*Lockheed Palo Alto Research Laboratory
**HSAD, 505-32-13, 804-827-2673

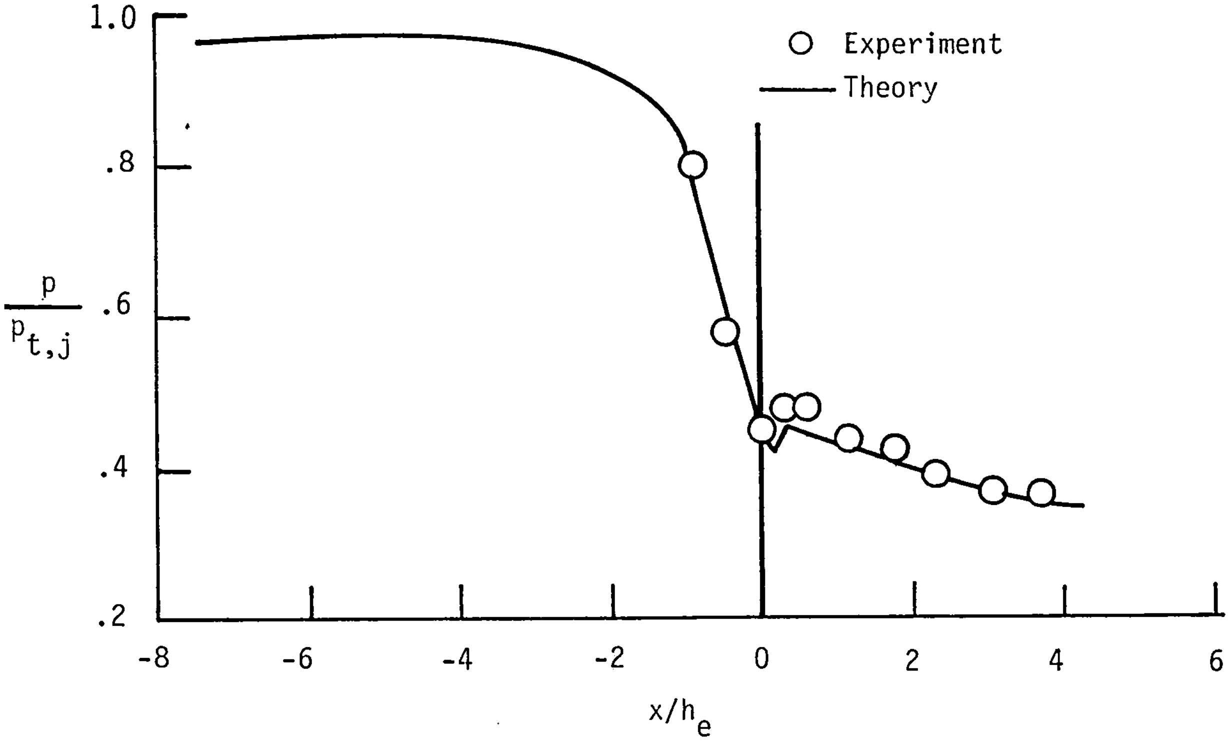


Figure 1.- Comparison of predicted and experimental surface pressures inside nozzle at $p_{t,j}/p_\infty \approx 3$.

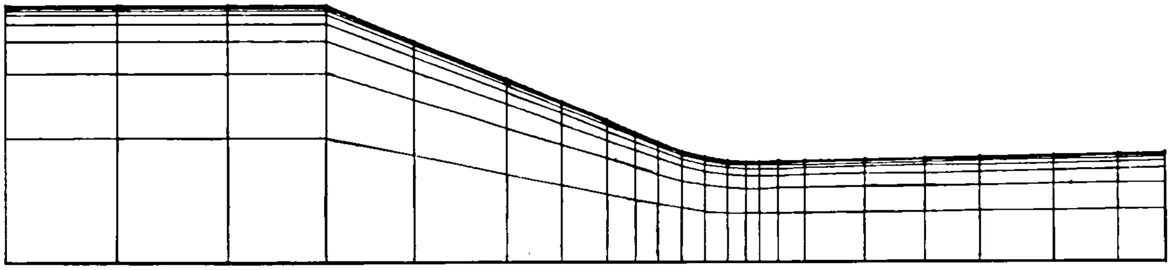


Figure 2.- Computational grid.

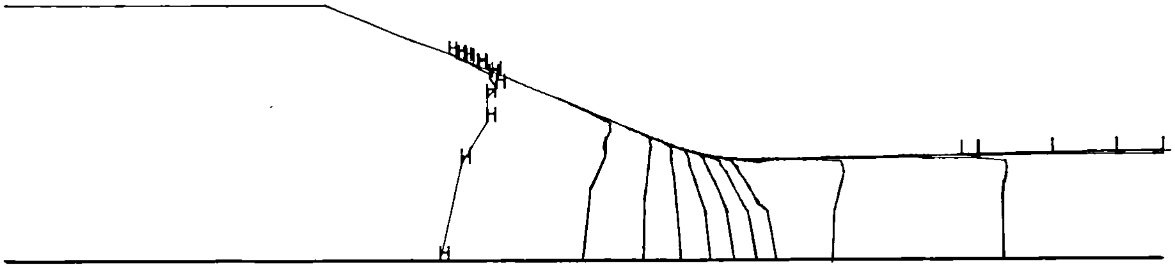


Figure 3.- Density contours.

BOUNDARY CONDITIONS FOR SUBSONIC COMPRESSIBLE NAVIER-STOKES CALCULATIONS

David H. Rudy* and John C. Strikwerda**

A systematic study has been undertaken of inflow and outflow boundary conditions for the numerical solution of the compressible Navier-Stokes equations. Combinations of representative inflow and outflow boundary conditions were applied in the solution of the subsonic flow over a flat plate in a finite computational domain. These boundary conditions were evaluated in terms of their effect on the accuracy of the solution and the rate of convergence to a steady state.

An outgrowth of this study was the development of a non-reflecting outflow boundary condition which has been shown to be effective in reducing reflections from subsonic outflow boundaries. These reflections can have a large effect on the rate of convergence of the solution to a steady state, and substantial improvements in the rate of convergence to steady state were obtained using this non-reflecting boundary condition. In computations of transient solutions this new boundary condition is useful to reduce the magnitude of the non-physical reflections from the outflow boundary.

The non-reflecting boundary condition involves a parameter whose optimal value is estimated using an analysis of a simplified set of equations.

References:

1. Rudy, D. and Strikwerda, J. "A Non-Reflecting Outflow Boundary Condition for Subsonic Navier-Stokes Calculations" J. Comp. Phys., to appear.

2. Rudy, D. and Strikwerda, J. "A Non-Reflecting Outflow Boundary Condition for Subsonic Navier-Stokes Calculations" ICASE Report 79-2, January, 1979.

3. Rudy, D. and Strikwerda, J. "Boundary Conditions for Subsonic Compressible Navier-Stokes Calculations" ICASE Report 79-18, August, 1979.

---

*HSAD, 505-31-31-03, 804-827-2806
**ICASE, 505-31-83-01, 804-827-2513

NUMERICAL SIMULATION OF HIGH REYNOLDS NUMBER, THREE DIMENSIONAL, COMPRESSIBLE FLOW

M. Y. Hussaini* and Steven A. Orszag

We have developed a high resolution mixed finite-difference-spectral code for the direct numerical simulation of high Reynolds number three-dimensional compressible flow. The code runs with up to 64 × 32 × 32 degrees of freedom on the STAR-100 computer in about 5 sec. per time step. Semi-implicit time splitting methods are used to achieve second-order accuracy in time while maintaining stability on convective time scales for low Mach number runs.

The code is being applied to the study of the inviscid stability of boundary and free-shear layers. The code is validated by comparison with known linear results of Mack and others. Finite amplitude nonlinear effects in plane-parallel compressible shear flows are also studied. The role of three-dimensionality is examined as a function of Mach number in order to compare mechanisms of transition to turbulence in compressible and incompressible flows. Detailed flow properties are being studied. Finally, the decay of three-dimensional homogeneous, compressible turbulence is being studied. Effects of compressibility on turbulence spectra are being investigated.

---

*ICASE, 505-31-83-01, 804-827-2513

Parabolic Approximation

Methods in which the steady N-S equations are approximated in such a manner that a predominant spatial "marching" direction is used. Not including Boundary-Layer methods.

APPLICATION OF PARABOLIC ANALYSIS TO PRACTICAL SCRAMJET FLOW FIELDS

John S. Evans*

A vital part of the effort to develop a supersonic combustion ramjet engine (scramjet) is the capability for theoretical calculations of flow properties in the combustor, where hydrogen is injected into and burned in a supersonic airstream. In various parts of the engine the flow includes shock waves, expansions, turbulent mixing of gas streams, complex boundaries, recirculating flows, and chemical reactions. No computer code exists which can handle such an array of difficult problems simultaneously, and, even if there were such a program available, it would be too expensive and cumbersome for general use.

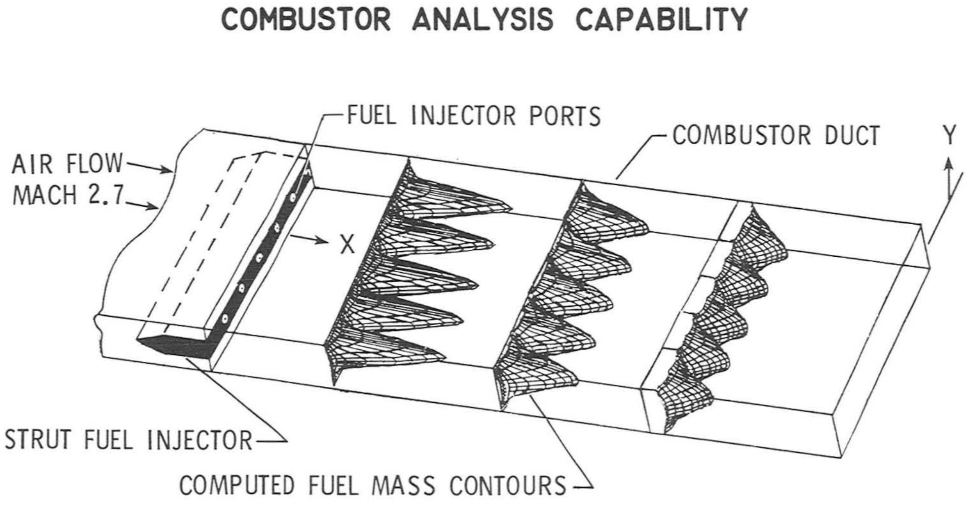
There are many situations in supersonic combustion ramjets in which simple analyses will suffice. If the flow has a predominant flow direction, and if it is supersonic, the parabolized Navier-Stokes equations adequately characterize the flow field. These conditions are often satisfied in the downstream portion of a scramjet combustor, where the near-field effects of fuel injection have disappeared.

Calculated profiles (ref. 1) typical of such a downstream combustor flow field are compared in figure 1 with experimental results. Helium was injected from a fuel injector strut parallel to the supersonic airstream, and profiles were taken at the center exit plane of helium mass fraction, temperature, and flow velocity. The agreement between the data and the calculated profiles is fairly good and is representative of the kind of predictive capability needed for scramjet combustor development.

References:
1. Drummond, J. Philip, Rogers, R. Clayton, and Evans, John S.: "Combustor Modeling for Scramjet Engines." Paper presented at the AGARD 54th(B) Specialists' Meeting on Combustor Modeling, Propulsion and Energetics Panel, Cologne, Germany, Oct. 3-5, 1979.

---

*HSAD, 505-32-93, 804-827-2803


COMBUSTOR ANALYSIS CAPABILITY
FUEL INJECTOR PORTS
COMBUSTOR DUCT
Y
AIR FLOW
MACH 2.7
X
STRUT FUEL INJECTOR
COMPUTED FUEL MASS CONTOURS

- 3 - D TECHNIQUE FOR DOWNSTREAM COMBUSTOR FLOW HAS BEEN DEVELOPED
  - MIXING
  - REACTION
  - TURBULENCE

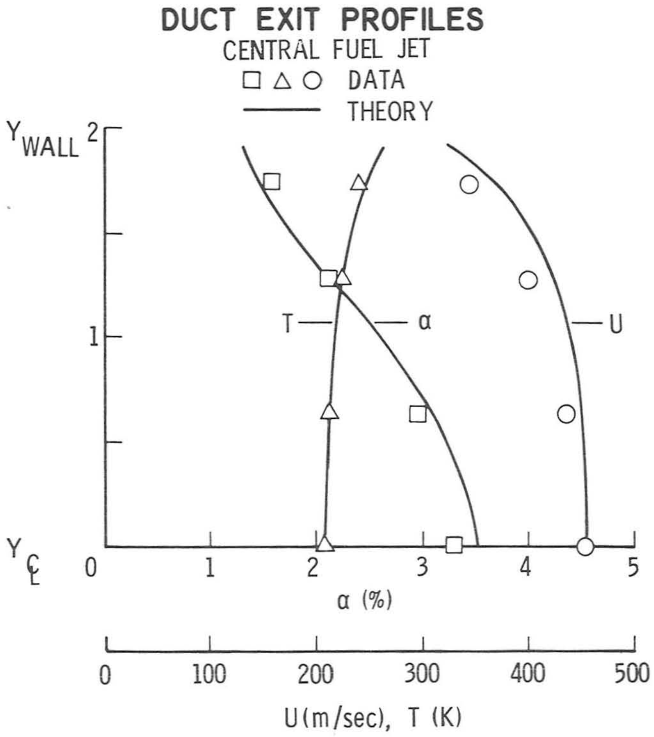


Figure 1. Simulated fuel injection profiles (helium); theoretical and experimental

TURBULENT MIXING AND REACTION IN THREE-DIMENSIONAL PARTIALLY ELLIPTIC FLOWS OF SUPERSONIC COMBUSTORS

R. Clayton Rogers*

The ability to computationally predict details of the turbulent, chemically reactive flows in supersonic combustor configurations has been steadily improving over the past ten years. Until recently, these flows have been modeled using parabolic assumptions so that the computer code could employ a spatially marching solution algorithm. As long as the combustor flow is supersonic with a predominant flow direction, at conditions such that chemical reaction rates are fast, and the turbulence is properly initialized, these codes can be used successfully. However, in certain regions of the combustor, such as near fuel injectors where flow separation and recirculation may occur and in zones where chemical reactions may produce local regions of embedded subsonic flow, the flow may be elliptic. Even with the more powerful computer hardware now available, and improved numerical algorithms, the solution of the fully elliptic form of the governing equations is intractable at the present time. Therefore, a partially elliptic scheme has been developed to handle flows which are elliptic due to local subsonic flow but do not have separation and recirculation.

The present computer code was adapted from the SHIP computer program (ref. 1). SHIP is capable of computing the three-dimensional parabolic flow of the turbulent mixing and combustion of hydrogen in a supersonic airstream. In the partially elliptic version, all variables except static-pressure are computed as if the flow was purely parabolic. The static pressure, however, is stored in a three-dimensional array and is used in subsequent iterations to determine the approximate pressure gradient source term in the momentum equations. The numerical procedure combines the marching integration scheme of SHIP with an iterative scheme to solve the elliptic pressure field. Thus, the procedure is termed "partially" elliptic. Additional details of this code may be found in reference 2.

Current plans are to evaluate this partially elliptic code with supersonic combustion data in which local subsonic flow regions are expected. Such evaluations will lead to further development of the iterative procedure and a possible criterion for convergence of the iterative solution.

References:

1. Markatos, N. C., Spalding, D. B., and Tachell, D. G.: "Combustion of Hydrogen Injected into a Supersonic Airstream", (The SHIP Computer Program), NASA CR-2802, April 1977.

2. Pan, Y. S.: "The Development of a Three-Dimensional Partially Elliptic Flow Computer Program for Combustor Research", NASA CR-3057, November 1977.

---

*HSAD, 505-32-93, 804-827-2803

BLUNT-BODY VISCOUS-SHOCK-LAYER ANALYSIS THAT INCLUDES TURBULENCE, MASS INJECTION, AND RADIATION

James N. Moss*

A viscous-shock-layer analysis has been developed in-house at the Langley Research Center to specifically study the flow field and heating environment experienced by planetary probes. The computer program for this analysis is called HYVIS. The motivation for developing the program was the fact that the aerothermal environment encountered by entry probes cannot be duplicated in existing ground experimental facilities. Consequently, probe heat shield designs must rely extensively on numerical predictions. Because of the complexity of the flow about entry probes, the numerical solutions must account for physical phenomena such as ablation injection, radiation transport, reacting chemistry, and turbulence.

The HYVIS program has been written in CDC FTN Fortran Extended and has been designed for optimum execution in the CDC CYBER environment. The flow equations are obtained from the steady-state Navier-Stokes equations by retaining terms up to second-order in the inverse square root of a Reynolds number (Ref. 1). Consequently, one set of equations are solved that are uniformly valid throughout the shock layer. Real-gas effects are accounted for by treating the gas as an equilibrium gas mixture where the equilibrium composition is determined by a free energy minimization procedure. Radiation transport is calculated with a nongray radiation model that accounts for molecular band, atomic line, and continuum transitions. Also, the flow may be either laminar or turbulent. Turbulence is calculated by using a two-layer eddy-viscosity model. For solutions with mass injection, the injection rate may be either specified or calculated for quasi-steady ablation.

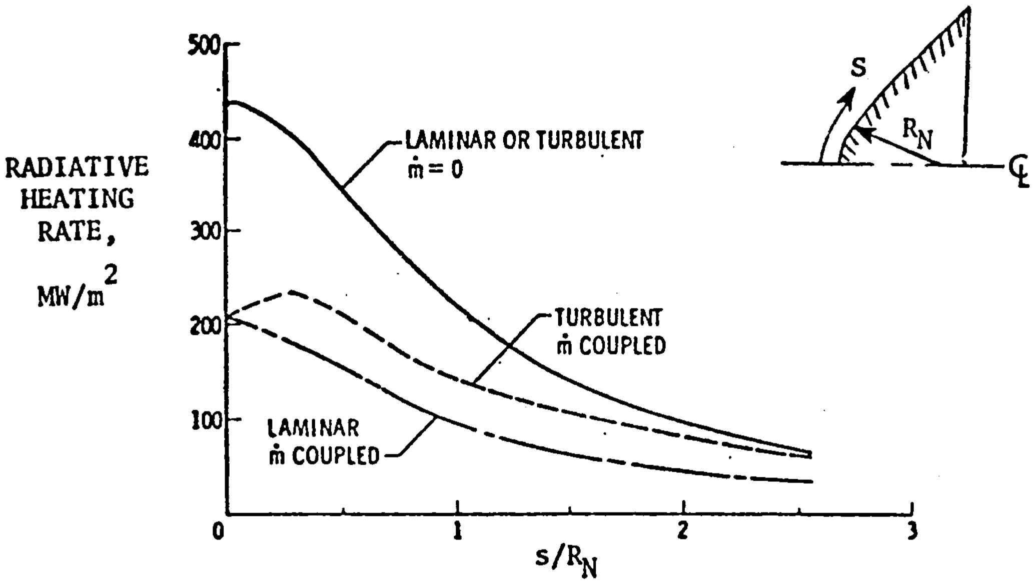
Figure 1 presents results of calculations for a probe (35° hyperboloid) entering the atmosphere of Jupiter. The effects of both turbulence and coupled (the mass injection rate $\dot{m}$ is governed by the net surface heating) ablation injection on radiative [Fig. 1(a)] and convective [Fig. 2(b)] heating are demonstrated. Results such as these are used to establish correlations that can be used in approximate engineering calculations.

Reference:

1. Moss, J. J., "A Study of the Aerothermal Entry Environment for the Galileo Probe," AIAA Paper 79-1081, June 1979.

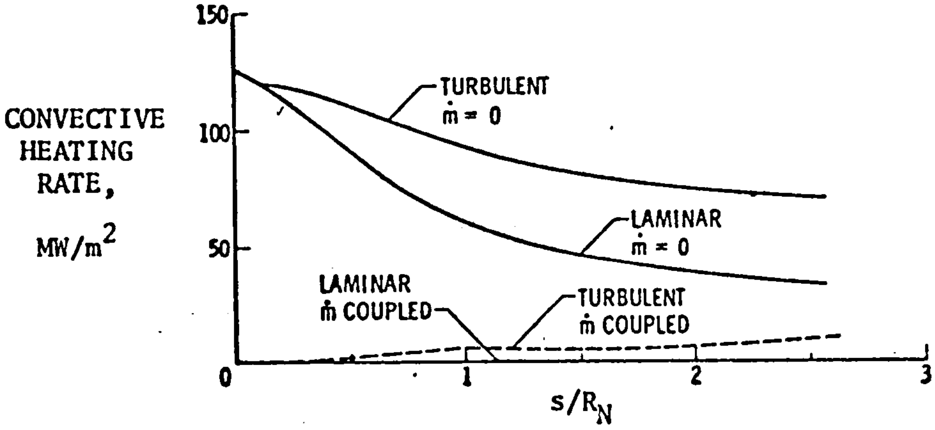
---

*SSD, Langley, 804-827-3770



(a) Radiative heating.

Fig. 1 Effect of coupled carbon-phenolic injection and turbulence on surface heating for Jupiter entry (freestream conditions: velocity = 39.29 km/s, density = 4.36×10^{-4} kg/m^3, and Mach number = 47.6. The nose radius R_N = 0.3112 m).



(b) Convective heating.

Fig. 1 Concluded.

PRELIMINARY THERMAL ANALYSIS FOR SATURN ENTRY

E. V. Zoby and J. N. Moss*

The planetary exploration program of NASA includes possible missions to the planet Saturn. Since experimental facilities cannot adequately duplicate the aerodynamic and thermal conditions expected during Saturn entry, much of the required information must be obtained from computational analyses. The purposes of this investigation are to scope the thermal problem for Saturn entry as well as assess the adequacy of current numerical techniques. For a realistic evaluation, the analysis is based on a recent mission study referred to as the Saturn Orbiter Dual Probe (SO2P) mission.

The preliminary thermal study,[1] using viscous-shock-layer (VSL) and engineering codes developed for Project Galileo, investigated nonequilibrium chemistry effects on the Saturn thermal environment, defined the primary heat-transfer mode for heat-shield design, delineated some problem areas for future thermal studies, and validated an engineering code[2] for parametric or design studies.

The effect of nonequilibrium chemistry appears to significantly influence only the radiative fluxes with effects localized to the stagnation region. This latter result is demonstrated in Figure 1. The freestream conditions are based on the nominal Saturn entry (SO2P). Note the radiative heating rates based on an equilibrium VSL code[3] are approximately 2 MW/m^2 where the corresponding value for Project Galileo is approximately 500 MW/m^2. The heat-transfer mode pertinent to the overall heat-shield design is convection as demonstrated by the results presented in Table 1. These results are given in terms of the integral of the heating rate over the body surface area.

References:

1. Zoby, E. V. and Moss, J. N., "Preliminary Thermal Analysis for Saturn Entry," AIAA Paper 80-0359, January 1980.

2. Zoby, E. V., Moss, J. N., and Sutton, K., "Approximate Convective Heating Equations for Hypersonic Flows," AIAA Paper 79-1078, June 1978.

3. Moss, J. N., "A Study of the Aerothermal Entry Environment for the Galileo Probe," AIAA Paper 79-1081, June 1979.

4. Tiwari, S. N. and Szema, K. Y., "Effects of Precursor Heating on Radiating and Chemically Reacting Viscous Flow Around a Jovian Entry Body," NASA CR-3186, September 1979.

---

*SSD, Langley, 804-827-2707

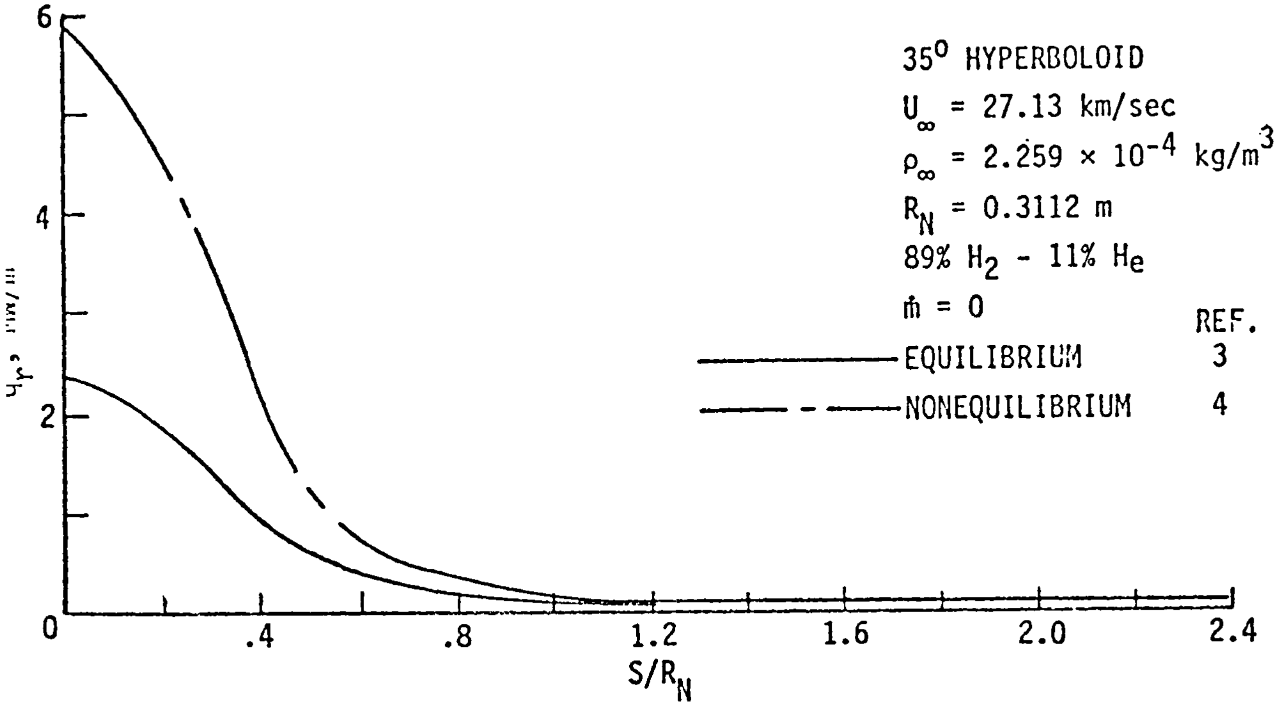


FIGURE 1. NONEQUILIBRIUM AND EQUILIBRIUM RADIATIVE HEAT-TRANSFER DISTRIBUTIONS.

TABLE 1. EQUILIBRIUM RADIATIVE AND CONVECTIVE POWER[a]

TIME, sec	$\int \dot{q}_r \, dA$	$\int \dot{q}_c \, dA$[b]
13.00	0.04	6.4
15.00	0.10	10.3
16.75	0.16	15.1
17.75	0.16	17.2
19.00	0.12	20.9

[a] 35^o HYPERBOLOID; POWER = MW, $\dot{m}$ = 0

[b] LAMINAR FLOW, WALL TEMPERATURE = 4000 K

7. Turbulence Effects and Turbulence Modeling

Studies of the effects of using various turbulence models in essentially existing Navier-Stokes or Boundary-Layer codes, or methods for modeling turbulence, and turbulence simulation.

TURBULENCE MODELS FOR SUPERSONIC COMBUSTORS

P. T. Harsha* and R. B. Edelman*

Severe and conflicting demands are made upon a computer code for engineering calculations of fluid-flow properties in supersonic combustors. In each step solutions must be found for several partial differential equations, grid-spacing must be fine enough for the resolution of gradients, and the step size must be small enough to avoid inaccurate or diverging answers. There is a motivation to make all parts of the code as simple and efficient as possible so as to minimize computer time and cost. Typical of the kinds of compromises which must be made is the selection of a turbulence model.

The transport of energy, momentum, and mass in the turbulent flow is far too complex to calculate in detail and must be modeled. Mixing length models (ref. 1) of varying degrees of complexity serve well for boundary layers near walls. For free shear layers with a dominant flow direction, the 2-equation k-e model (ref. 2) is popular because the dissipation length scale is automatically calculated and because it models turbulence properties fairly accurately for a wide range of flows without changing the empirical constants. Beyond these cases, there is no consensus as to the best model, but there is a need to determine suitable models to use in complex geometry, in recirculating flow, and in reacting flow.

An investigation is being carried out under Contract NAS1-15988 to determine the best turbulence model (or models) for use in scramjet combustor calculations. Present plans call for tests of multiple dissipation length scale k-e models (ref. 3), use of an algebraic Reynolds stress model (ref. 4), combinations of both of these, and, finally, use of a full Reynolds stress model. The models will be tested with 2-D and 3-D parabolic and elliptic programs. To the extent that data are available, the models will be adjusted so as to maximize agreement with data. To be useful for engineering calculations the models chosen cannot be so complex as to increase computation costs by an unreasonable amount.

References:

1. Cebeci, T.; and Smith, A. M. O.: "Analysis of Turbulent Boundary Layers," Applied Math. and Mech., Vol. 15, Academic Press, N.Y., 1974.

2. Harsha, P. T.: "Kinetic Energy Methods," Ch. 8 of Handbook of Turbulence, Vol. 1, W. Frost and T. Moulden, eds., Plenum Publishing Corp., N.Y., 1977.

3. Launder, B. E.; and Schiestel, R.: "On the Utilization of Multiple Time Scales in the Modeling of Turbulent Flow," C. R. Acad. Sci., Paris, Vol. 286, Series A, pp. 709-712, Apr. 29, 1978. (In French)

4. Rodi, W.: "The Prediction of Free Turbulent Boundary Layers by Use of a Two-Equation Model of Turbulence," Ph.D. Thesis, Univ. of London, 1972.

---

*Science Applications, Inc., Dept. of Combustion Science and Advanced Technology, 2113 Victory Blvd., Canoga Park, CA 91364, NAS1-15988, 213-348-8520

MERGING OF TOROIDAL VORTICES

C. H. Liu*

Vortex rings have been of interest for many years due to their widespread appearance in axisymmetric flows. Many fluid dynamic phenomena can be explained using models consisting of vortex rings moving in an inviscid stream. Asymptotic solutions for the interaction of free vortices with finite viscous vortical cores were based on the assumption that the ratio of the effective viscous core size over a certain length scale is small. It is well known that the asymptotic solutions may remain as good approximations even when this ratio is of the order of unity. In order to establish the accuracy of the asymptotic solutions, finite difference approximations to the axisymetric, unsteady incompressible Navier-Stokes equations are constructed to study the merging of vortex rings.

An initial distribution of vorticity, representing one or more vortex rings, is specified and then allowed to evolve by numerical solution of the governing equations. Since the vorticity decays exponentially as the square of the distance from a vortex, the vorticity can be assigned zero on the boundary which is at sufficient distance from the vortex. The coordinates are moving with instantaneous mean velocity of the vortex rings relative to the inertial coordinates so that the free vortices will remain inside an assigned boundary.

Numerical results are obtained for the self-merging of a single vortex ring which happens as the inner radius of the ring shrinks to zero. Figure 1 compares the asymptotic solutions for the decay of the maximum vorticity with the numerical results for Re = 12.56. It is clear that the asymptotic solution remains quite accurate even at $t = 16$. The error is caused by the nonlinear terms, therefore, the disagreement with the numerical results will increase as Re increases.

---

*ANRD, 505-32-03, 804-827-2617

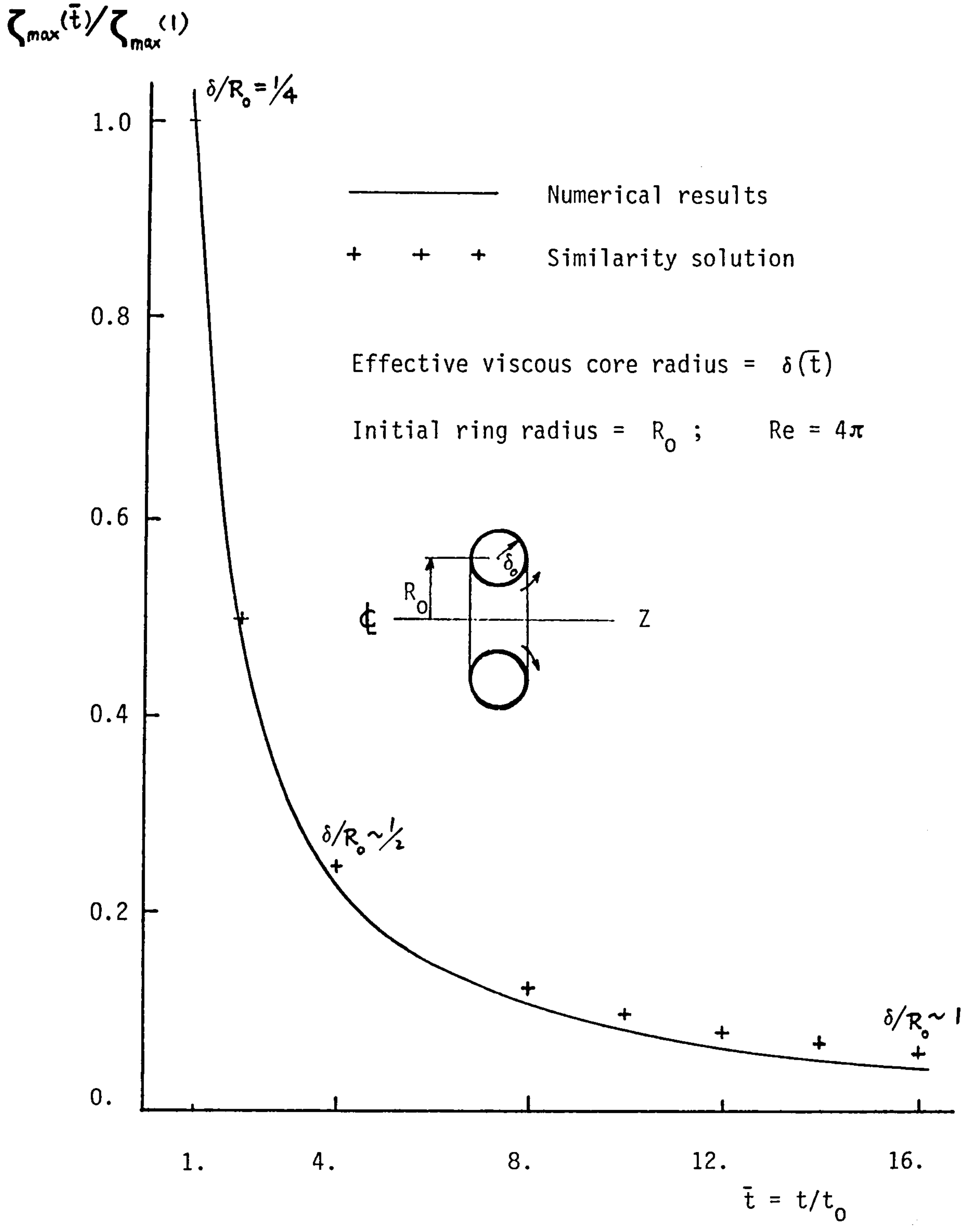


Figure 1. The decay of the maximum vorticity for the self-merging of a vortex ring

FINITE-ELEMENT SOLUTION ALGORITHM FOR AEROACOUSTIC TRAILING EDGE FLOW PREDICTION

A. J. Baker

Turbulent boundary layer flows departing the trailing edge of an aerodynamic surface are experimentally verified to be strong sources of noise. Alteration of the aerodynamic surface can alter far-field intensity by local absorption as well as modification to the turbulent flow structure prior to departing the trailing edge. A considerable volume of experimental data has been obtained for the case of a jet flow directed over a surface. Attention is now turning to aeroacoustic analysis of the near-field wake associated with airfoil flowfields induced solely by forward flight.

A theoretical analysis is presented yielding sets of partial differential equations for determination of turbulent aerodynamic flowfields in the vicinity of an airfoil trailing edge. A four-phase interaction algorithm is derived to facilitate the analysis. Following input, the first computational phase is an elementary viscous-corrected two-dimensional potential flow solution yielding an estimate of the inviscid-flow induced pressure distribution. The next phase involves solution of the turbulent two-dimensional boundary layer equations up to the trailing edge, with transition to a two-dimensional parabolic Navier-Stokes equation system describing the near-wake merging of the upper and lower surface boundary layers. An iteration provides refinement of the potential flow induced pressure coupling to the viscous flow solutions. The final phase, if desired, is a complete two-dimensional Navier-Stokes analysis of the wake flow in the immediate vicinity of the trailing edge.

A comparison of computed mean velocity, U_1, with experimental data from reference 1 downstream of the trailing edge for the airfoil NACA 63-012 at zero angle of attack is shown in figure 1. The computed results were obtained using the two-dimensional parabolic Navier-Stokes formulation and compare quite favorably with data except in the region where the upper and lower streams merge.

This work has been conducted by Computational Mechanics Consultants under NASA contract NAS1-14855/MOD. 1.

Reference:

1. Yu, J. C. "Mean Flow and Reynolds Stress Measurements in the Vicinity of the Trailing Edge of a NACA 63-012 Airfoil," NASA TM-80224.

ANRD, 505-10-43-01, 804-827-2617

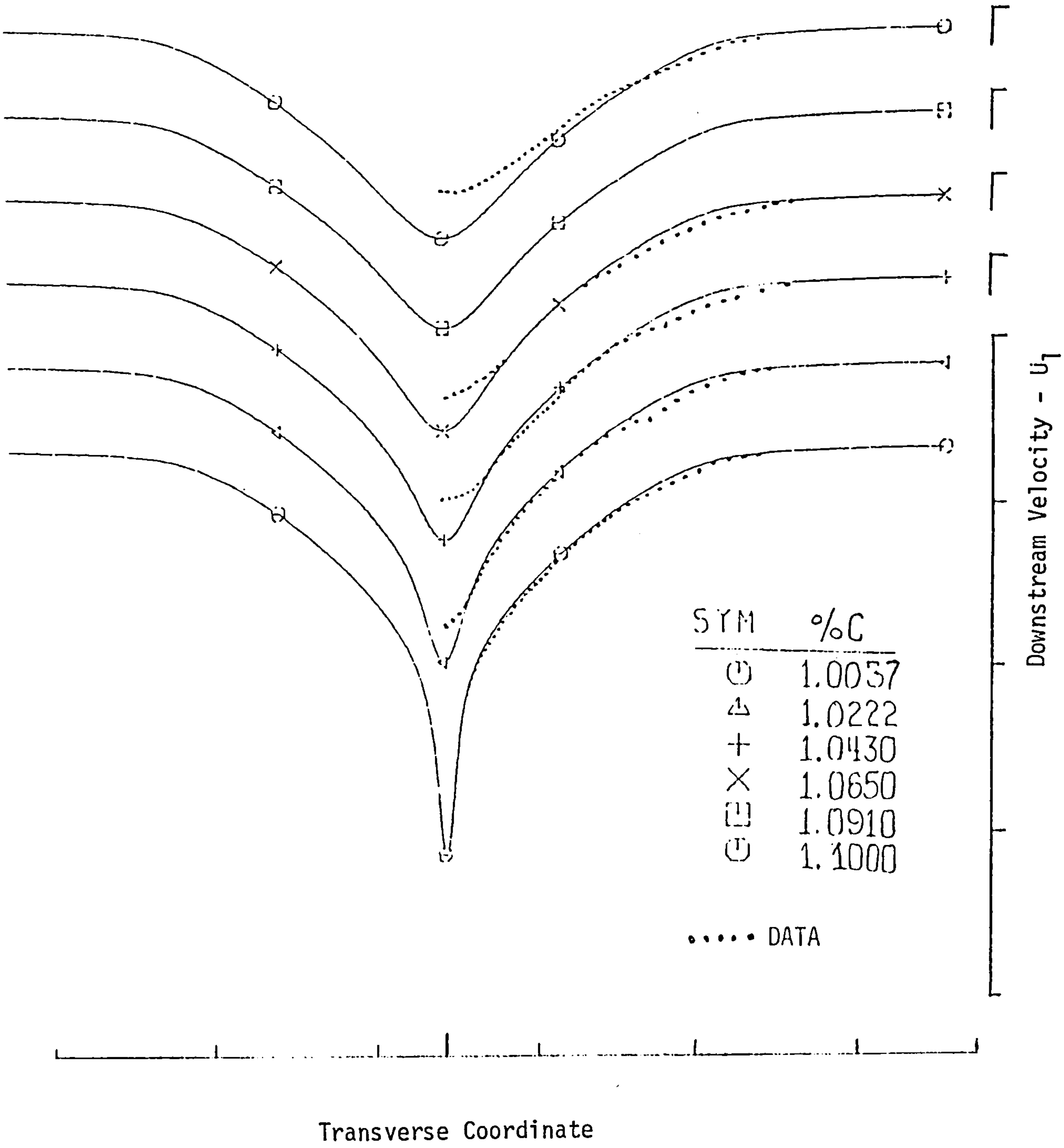


Fig. 1 U_1 Mean Velocity Profile

TIME DEPENDENT NUMERICAL SOLUTION OF TURBULENT JET FLOWS USING REYNOLDS STRESS CLOSURE MODELS

T. B. Gatski

In the study of turbulent flows, it has become common practice to numerically solve a set of conservation equations for stationary flow variables to determine the mean characteristics of each flow. The level of sophistication utilized in the closure of the governing differential equations is determined by which flow variables needed to be known and the flow geometry. One can classify these closure approximations according to the number of differential equations used to describe the required Reynolds stresses.

In the study of sound generated by free turbulent shear flows, it is becoming apparent that the large coherent scales of motion in the flow play a significant role in the sound generation process, either directly or through an interaction with the smaller component scales of motion. This fact has led to experimental investigations which have attempted to enhance these coherent scales by acoustic excitation in order to better understand their specific role. If the coherent motion mentioned earlier can be modeled in a deterministic manner then a time history of certain scales of motion can be computed.

In this vein, consider a triple decomposition of the flow field into a time mean component, a phase-averaged component, and a small scale random component. Phase-averaged equations for the vorticity, stream function and (phase-averaged) turbulent stresses are formulated and the entire system solved numerically in a radially stretched cylindrical coordinate system from the jet exit to eight diameters downstream. The transport equations for the phase-averaged stresses are those obtained by Launder, et al. (ref. 1) for the time-averaged Reynolds stresses with the constants adjusted for the forced flow under consideration.

The numerical experiment under consideration is the modeling of a forced axisymmetric turbulent jet. The initial solution stage consists of the solution of the unforced jet problem to provide a consistent set of initial conditions for the forced case. Since the governing differential equations are time dependent, the initial solution phase requires that the equations be computed through to a steady-state solution. This allows for validation of the differential model and numerical algorithm as well as insight into the transient characteristics of the flow.

Reference:

1. Launder, B. E., Reece, G. J., Rodi, W. "Progress in the Development of a Reynolds-Stress Turbulence Closure," J. Fluid Mech., 1975, vol. 68, pt. 3.

ANRD, 505-32-03-05, 804-827-2617

COMPUTED EVOLUTION OF LARGE SCALE WAVE-LIKE STRUCTURES IN TURBULENT JET FLOWS AND THEIR SOUND PRODUCTION

P. J. Morris

It is well known in the study of free turbulent jet flows that excitation of the jet by a pure tone causes broadband jet noise amplification. In order to explain this phenomenon, it is necessary to understand the effects of the pure tone excitation on the structure of the turbulent flow. Fortunately, recent studies of turbulent flows indicate that there is a high degree of organization at the larger scales, and, in the case of a free turbulent jet, that the organized structure may be adaptable to a wave representation.

One method of isolating the large scales is to decompose the flow field into a time average component, a phase averaged component, and a small scale random component. From this triple decomposition, energy equations for the respective component scales can be obtained. The closure problem present in such a formulation is handled by way of gradient diffusion hypotheses and dimensional analysis considerations in the turbulent energy equation and by way of shape assumptions, invariant with respect to downstream distance, in the large scale energy equation.

The time mean continuity and momentum equations along with the mean turbulent kinetic energy equation are rewritten in terms of a modified stream function coordinate and the system solved using the method of cubic splines. The large scale energy equation is solved by taking the large scales as instability waves of fixed real frequency whose wavelength and growth rate are the eigenvalues of a local viscous stability calculation and integrating the energy equation in the radial direction. This reduces the partial differential equation for large scale kinetic energy to an ordinary differential equation, in the axial direction, for the magnitude (squared) of the wave amplitude. A calculation of the local growth rate, $-\alpha_i$, as a function of jet half width, b, is shown in figure 1 for the potential core region of the jet. As is seen, the agreement between the viscous and inviscid solution is very good, even in the damped region, $\alpha_i > 0$. The far-field pressure distribution produced by the large scale structure can be readily calculated using a matched asymptotic expansion method developed by Morris and Tam (ref. 1).

This work is being conducted at the Pennsylvania State University under NASA grant NSG 1580.

Reference:

1. Morris, P. J. and Tam, C. K. W. "Near and Far Field Noise from Large-Scale Instabilities of Axisymmetric Jets," AIAA 4th Aeroacoustics Conference, Atlanta, Georgia, 1977, Paper No. 77-1351.

ANRD, 505-32-03-05, 804-827-2617

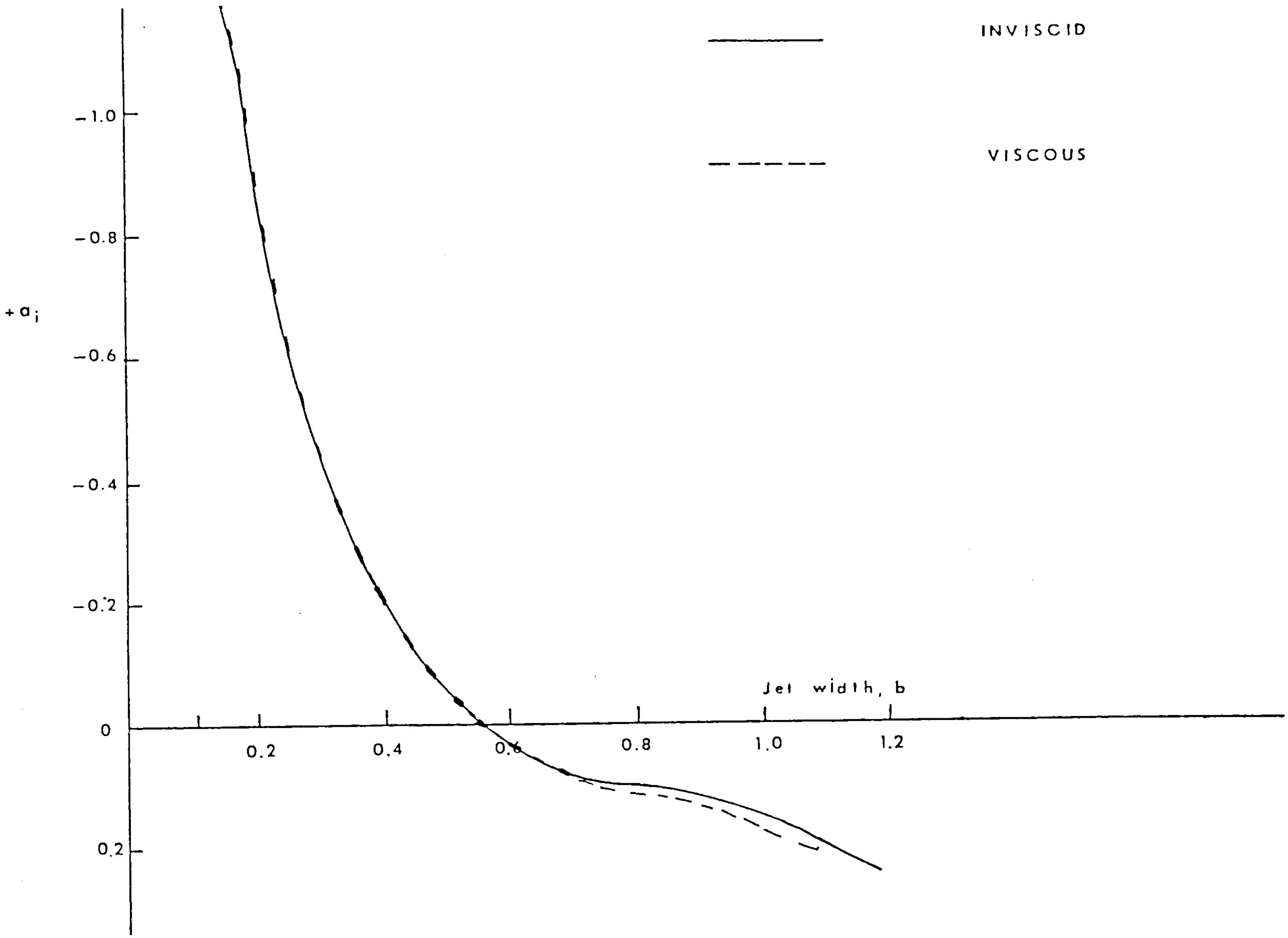


Figure 1. Comparison of viscous and inviscid solutions

8. Viscous Flow Stability and Transition

Methods whose primary focus is the stability of viscous flows.

CALCULATION OF 3-D LAMINAR BOUNDARY LAYER TRANSITION IN THE PRESENCE OF FREE-STREAM TURBULENCE

E. Clay Anderson*

The design suction rate distributions required to maintain laminar flow on the supercritical airfoil to be used in the Langley LFC swept-wing experiment have been determined using a linearized stability analysis, which does not include provisions to account directly for the turbulence level in the approaching freestream. Recent developments in "higher-order" or two-equation turbulence models have led to predictive methods that directly account for the intensity of turbulence in the freestream. Some of the two-equation turbulence models have been applied successfully to predict the influence of freestream turbulence levels upon laminar-turbulent transition in two-dimensional flow fields. More recently, Dr. Wilcox of DCW Industries has developed a three-dimensional computer code incorporating an advanced two-equation turbulence model which appears ideally suited for the analysis of the boundary layers on the LFC airfoil. Dr. Wilcox has recently demonstrated an ability to perform calculations which showed the importance of including the influence of free-stream turbulence in laminar flow applications.

Because of the extreme sensitivity of laminar boundary layers to the free-stream turbulence intensity, and the high degree of imprecision involved in all current prediction methods, an independent analysis of the LFC-airfoil boundary layers in the presence of freestream turbulence levels that are representative of the known freestream turbulence levels in the Langley 8'-TPT will be done by DCW Industries for NASA Langley.

---

*STAD, 534-01-13, 804-827-2627

NONLINEAR STABILITY THEORY AND OPTIMUM NUMERICS FOR LAMINAR FLOW CONTROL ANALYSIS

Steven A. Orszag*

A number of extensions and generalizations of the SALLY computer code for stability analysis are being studied. First, the extension of SALLY to study compressible flows is underway. Second, the generalization of SALLY to include nonparallel and nonlinear effects is underway. Finally, some new efficient techniques for the resolution of critical layers are being investigated to further improve the efficiency of SALLY.

SALLY is based on the solution of the Orr-Sommerfeld equation using expansions in Chebyshev polynomials. For incompressible flow, the critical layer lies close to the wing so these expansions provide good accuracy with modest resolution. On the other hand, in compressible flows, the critical layer moves away from the wing so additional resolution is required to achieve acceptable accuracy. However, we have developed a new method to resolve the critical layer which promises significant efficiency improvements in both compressible and incompressible environments.

Nonlinear and nonparallel effects are included using a spectral transform method. This technique permits efficient implementation of these terms with minimal technical complications.

---

*Cambridge Hydrodynamics, Inc.

THREE-DIMENSIONAL, COMPRESSIBLE, NONPARALLEL, LINEAR STABILITY THEORY FOR LAMINAR FLOW CONTROL ANALYSIS

Ali H. Nayfeh*

A theory is developed for the linear stability of three-dimensional growing incompressible or compressible boundary layers. The method of multiple scales is used to derive partial-differential equations describing the temporal and spatial evolucion of the complex amplitudes and wavenumbers of the disturbances. In general, these equations are elliptic unless certain conditions are satisfied. For a monochromatic disturbance, these conditions demand that the ratio of the components of the complex group velocity be real, thereby relating the direction of growth of the disturbance to the disturbance wave angle. For a non-growing boundary layer, this condition reduces to $d\alpha/d\beta$ being real, where α and β are the complex wavenumbers in the streamwise and crosswise directions, in agreement with the result obtained by using the saddle-point method. For a wavepacket, these conditions demand that the components of the complex group velocity be real. In all cases, the evolution equations are reduced to inhomogeneous ordinary-differential equations along real group velocity directions.

The theory is being used in conjunction with the e^n method to develop a prediction method for transition. The method will be used to evaluate the effects of the proposed suction distributions on laminar flow control over swept back supercritical wings.

---

*Engineering Science and Mechanics Department
Virginia Polytechnic Institute and State University

TWO - AND THREE - DIMENSIONAL BOUNDARY-LAYER STABILITY ANALYSIS

Mujeeb R. Malik and Steven A. Orszag*

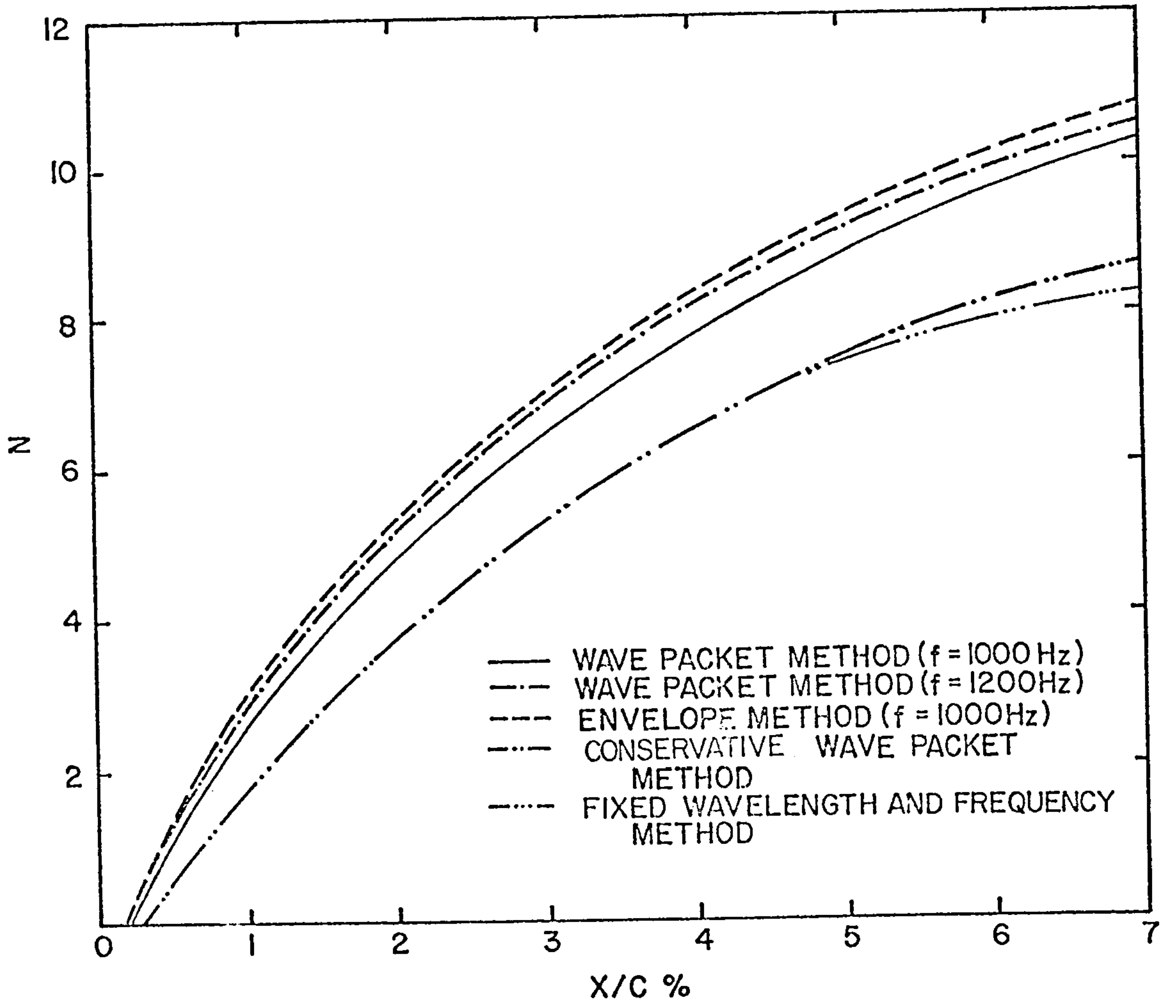
We compare several methods of transition prediction by linear stability analysis. We use the incompressible linear stability code SALLY in various ways to study flows over laminar flow control wings. In particular, we compare the so-called envelope method with wave-packet methods to predict transition. We conclude that the envelope method is at least as reliable as the more complicated wave-packet method.

The code SALLY is based on numerical solution of the Orr-Sommerfeld equation for three-dimensional disturbances in boundary layer flows using expansions in series of Chebyshev polynomials. In its original form, SALLY allowed computation of (i) maximum amplification among all wave lengths and propagation angles; (ii) amplification at fixed wave-length and propagation angles; (iii) amplification at fixed frequency and fixed wave length; (iv) amplification at fixed wave frequency and fixed propagation angle. We have extended the SALLY code to calculate the propagation of wave packets.

In the Figure, we compare various transition prediction methods applied to data taken at Cranfield for a large, untapered, 45° swept half wing mounted as a dorsal fin upon the mid-upper fuselage of an Avro Lancaster airplane. In this case, the angle of attack is 0° and experimental transition occurred at $x/c = 7\%$. The envelope method gives an N factor of 10.8 at a frequency of 1000 Hz. The wave packet method gives a maximum N factor of 10.5 at a frequency of 1200 Hz, which is close to the prediction of the envelope method. A conservative wave packet approximation gives an N factor of 8.6 rather than 10.5.

---

*Systems and Applied Sceinces, Inc. and Cambridge Hydrodynamics, Inc.



A plot of N versus precent of chord x/c for various methods applied to a 45° swept wing mounted on an Avro Lancaster airplane at 0° angle of attack.

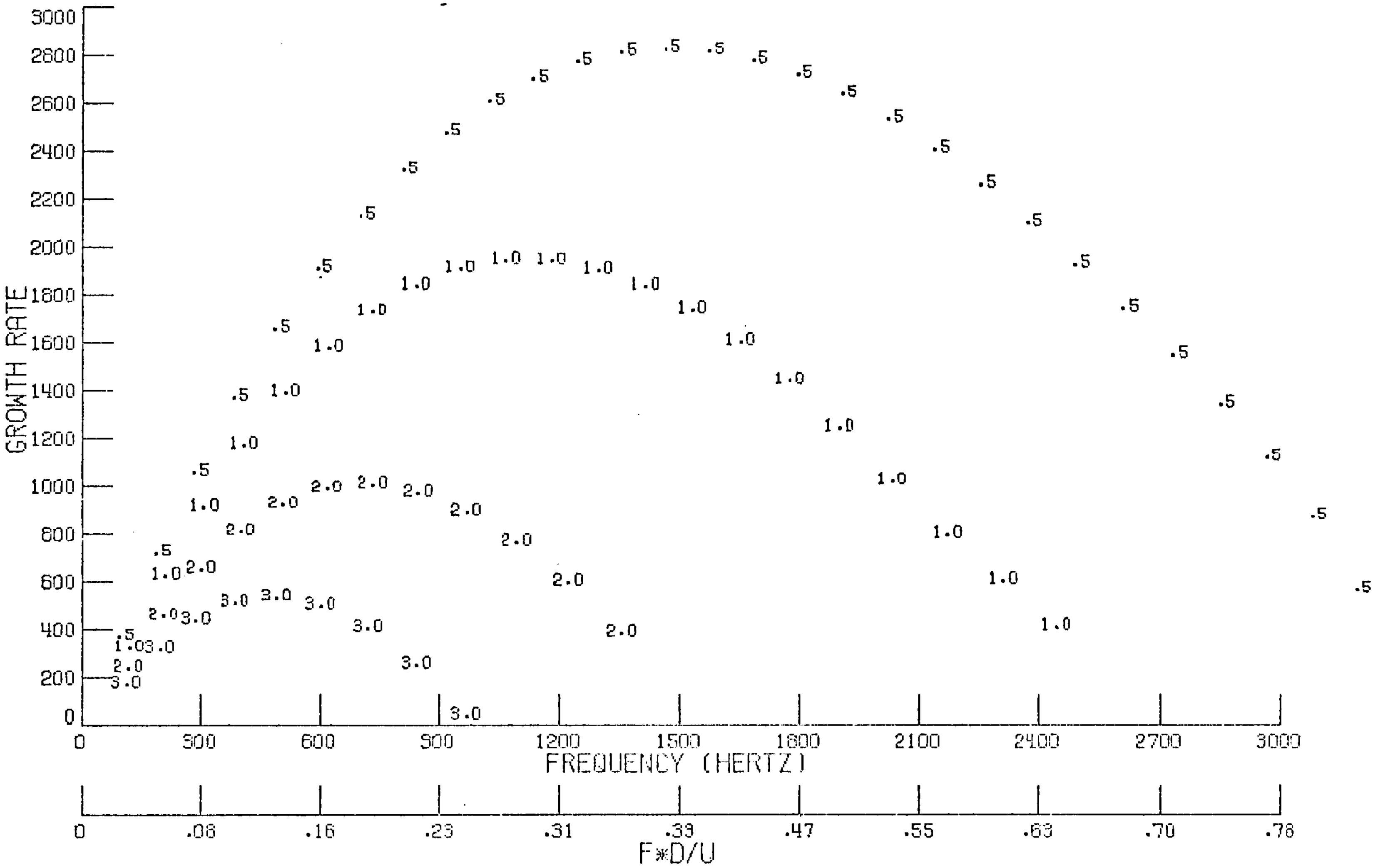
ON THE STABILITY OF THE FREE SHEAR LAYER FOR AN AXIAL SYMMETRIC JET AND PLUG FLOW JETS

Steven A. Orszag*, Lucio Maestrello**, and
Stan Lamkin***

The stability characteristics of inviscid axial symmetric jets and plug jets are being investigated to explain recent experimental observations by Maestrello and collaborators on the effects of plugs in the reduction of turbulent jet noise production. Both compressible and incompressible jets are being studied. The stability characteristics are then used to compute a typical sound radiation pattern from a form of Lighthill's integrals. Both axisymmetric and non-axisymmetric disturbances are studied. Some typical results for axisymmetric jets are given in the attached figure, which shows the growth of flow instabilities in the acoustical frequency range.

* Cambridge Hydrodynamics, Inc.
** NASA-Langley Research Center, ANRD
*** Kentron International, Inc., Hampton Technical Center

MAX. IMAG. VS FREQ.
FOR X DIAMETERS DOWNSTREAM
TWO INCH FULL JET TEMPORAL NDEX=1



ON THE STABILITY OF ROTATING DISK FLOW

Mujeeb R. Malik*

Linear stability analysis of flow due to a rotating disk results in a sixth order differential equation. However, if the Coriolis and non-parallel effects are neglected, then we obtain the usual Orr-Sommerfeld equation. Calculations were made using this relatively simple fourth order system. For the purposes of transition correlation, the envelope method was employed in which the growth rate is maximized with respect to wave number vector and the growth direction is taken to be the one defined by real group velocity vector. The results indicate that the transition N factor is ∿ 23.5 (at R = 520). The predicted number of stationary vortices at this Reynolds number is 33 which is in close agreement with the experimental evidence.[1]

The critical Reynolds number obtained by using Orr-Sommerfeld equation is ∿ 171. However, at such a small Reynolds number it cannot be justified to neglect the Coriolis force and non-parallel terms. So the sixth order system is solved by use of Chebyshev spectral technique employing complex QR-algorithm. The preliminary results indicate that the critical Reynolds number is much less than 171 and that the growth rates are larger than those given by Orr-Sommerfeld equation.

---

[1] Gregory, N., Stuart, J. T., and Walker, W. S., "On the Stability of Three-Dimensional Boundary Layers with Application to the Flow Due to a Rotating Disk," Phil. Tran. Roy. Soc., A-248, 1955, pp. 155-199.

*Systems and Applied Sciences, Corp.

STABILITY OF FLOWS OVER BODIES WITH SUCTION THROUGH POROUS STRIPS

Ali H. Nayfeh*

One of the approaches proposed for laminar flow control is suction through porous strips. To determine the effectiveness of such an approach and to optimize the number, spacing, and flow rate through such strips, one needs to determine the stability of the flow over a body with suction strips. The first step in such an approach is the calculation of the mean flow. Nayfeh and El-Hady[1] used a nonsimilar boundary-layer code. They found a strip-suction distribution that is as effective as uniform suction for one flow condition with an increase of less than 7 percent in the overall suction. These results could be optimized even further. However, the computations are very expensive because nonsimilar boundary-layer calculations require a large computation time. Moreover, they fail to account for the upstream influence (deviation in pressure gradient) and they are limited by the magnitude of suction through a strip. To overcome these difficulties, a linearized triple-deck formulation has been developed, which is more efficient and requires much less computer time. Consequently, it is ideal for optimization of the suction distribution for laminar flow control. Since the linearized triple-deck equations are not expected to be valid for large suction levels and very wide strips, their validity limits are being determined by comparison of their results with those of the interacting boundary-layer equations. The resulting mean-flow subroutine is being coupled with the nonparallel stability codes to produce a design tool that can optimize the strip-suction distribution for laminar flow control.

---

[1] Nayfeh, A. H., and El-Hady, N. M.: "An Evaluation of Suction Through Porous Strips for Laminar Flow Control." AIAA Paper 79-1494, 1979.

*Engineering Science and Mechanics Department
Virginia Polytechnic Institute and State University

9. Atmospheric Modeling

Methods for simulating atmospheric circulation, weather, storms.

PRIMITIVE EQUATION SPECTRAL MODEL OF THE ATMOSPHERIC GENERAL CIRCULATION

Dr. William L. Grose,* W. Thomas Blackshear,* and Richard E. Turner*

A three-dimensional model of the atmospheric general circulation has been adapted to the STAR computer system. The model is based upon the primitive equations with dependent variables represented as truncated expansions of surface spherical harmonics. Vertical derivatives are expressed as finite differences.

In its current form, the model utilizes truncation with maximum zonal wave-number of 21. A realistic representation of the Earth orography has been implemented in the model. Thermal forcing effects due to differential land-sea heating have been included by an appropriate parameterization. A parameterization for surface drag has been adopted to prevent unrealistic acceleration of the surface winds. Incorporation of a scale-dependent dissipation was included to prevent non-linear cascade of energy to the small-scale waves. A time-filter was introduced to guarantee stability during long-term (~ 1 year) integrations.

Current modifications are directed to incorporating diabatic heating in the dynamics model. Long-wave radiative processes will be treated by a Newtonian formulation. Ozone heating in the ultraviolet and visible spectrum will be included in the model stratosphere.

Several extended runs (~ 60 days) have been made with the model to study the characteristics of the model and to examine the effects of the various parameterizations included for physical processes. Initial studies will be conducted with the model to study the formation of large-scale blocking anticyclones.

---

*AESD, 304-91-00-03, 804-827-2537

SUMMARY OF A ZONALLY-AVERAGED CIRCULATION MODEL OF THE ATMOSPHERE

Richard E. Turner*

The zonally-averaged primitive equations of atmospheric circulation are integrated numerically to forecast the seasonal, zonally-averaged values of temperature, zonal wind, and meridional wind. The purpose of these computations is to provide transport simulation for stratospheric trace constituents related to environmental pollution problems.

The tendency equations for temperature, zonal wind, and meridional wind are formulated in the flux conserving form. Spatial derivatives are approximated by second-order accurate finite differences. Time integration is performed with the second-order accurate "leap frog" time differencing scheme.

The turbulent transports of synoptic scale cyclones are provided by two techniques. The turbulent transport of zonal wind is specified from atmospheric data and with seasonal variation. The turbulent transport of quasi-conserved quantities, such as potential temperature, are computed from eddy diffusivity coefficients.

Diabatic heating rates are computed from a detailed heating routine that accounts for atmospheric ozone, carbon dioxide, and water vapor as functions of season.

In a typical experiment, the model atmosphere is started from a given set of initial conditions for temperature, zonal wind, and meridional wind. The integration is performed over several yearly heating cycles until subsequent years repeat. The resulting flow fields are used to simulate transport for chemically reacting atmospheric constituents.

---

*AESD, 147-30-01, 804-827-2537

10. Aeroacoustic Methods

Methods designed primarily for the study of aeroacoustics phenomena.

EXTENSION OF FARASSAT THEORY TO INCLUDE HELICOPTER BLADE VORTEX INTERACTION NOISE

Danny R. Hoad*

Impulsive noise, when present, is typically rated as the most objectionable form of helicopter noise. Two forms of impulsive noise are known to be generated by most helicopters: high-speed thickness noise at the advancing blad tip and low-speed partial-power, descending-flight blade-vortex interaction (BVI) noise. The latter form of impulsive noise is the result of the interaction of a rotor blade with the shed tip vortex from a previous blade passage.

A theory for the determination of the acoustic pressure-time history for the helicopter has been developed by Farassat (ref. 1). It is based on an equation derived by Ffowcs Williams and Hawkings (ref. 2). The formulation includes a so-called thickness noise term associated with the local normal velocity of the moving blade and a so-called loading noise term associated with the local stress acting on the medium at the blade surface. The second term requires detailed input aerodynamic pressures on the blade. In fact, the sensitivity and accuracy of the formulation due to pressure variations are as yet not well defined.

Figures 1 and 2 are typical pressure-time histories of helicopter noise without and with BVI, impulsive noise, respectively. Since a BVI primarily results in a local flow angle (loading) change over a short period of time, it can be modeled theoretically by local pressure impulses on the blade. Using a simple momentum theory strip analysis for blade-span azimuth pressure calculations and coupling the rotor geometry analysis developed by Landgrebe (ref. 3), the present effort is designed to provide a program to predict noise generated by helicopter BVI.

One uncertainty in the calculations is the prediction of blade-vortex encounter. Figure 3 indicates the complexity of the rotor wake in the presence of a four-bladed rotor system. This figure is that predicted for a rotor condition at which wind-tunnel noise data are available. Using the noise data, it was possible to determine the precise location of the BVI, thus the sensitive parameters in the wake geometry program can be evaluated to determine their effect on the exact prediction of occurrence. With this in hand, the wake geometry program can be coupled with the noise prediction program to estimate the BVI noise characteristics as seen in figure 2.

References:
1. Farassat, F.: Theory of Noise Generation from Moving Bodies with an Application to Helicopter Rotors. NASA TR R-451, 1975.
2. Ffowcs Williams, J. E.; and Hawkings, D. L.: Sound Generation by Turbulence and Surfaces in Arbitrary Motion. Phil. Trans. Roy. Soc. London Ser. A, vol. 264, May 8, 1969, pp. 321-342
3. Landgrebe, A. J.; and Egolf, T. A.: Prediction of Helicopter Induced Flow Velocities Using the Rotorcraft Wake Analysis. Proceedings of 32nd Annual Am. Hel. Soc. Forum, A.H.S., May 1976

---

*STAD, 505-42-13, 804-827-3611

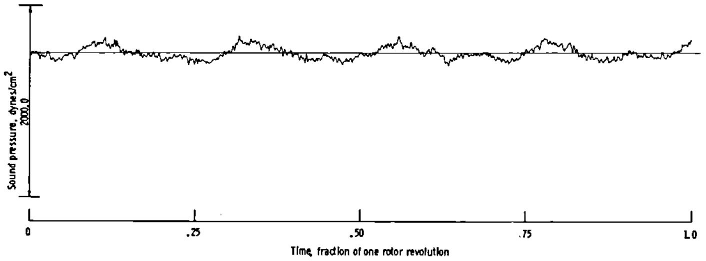


Figure 1.- Sample acoustic pressure-time history without blade-vortex interaction impulsive noise.

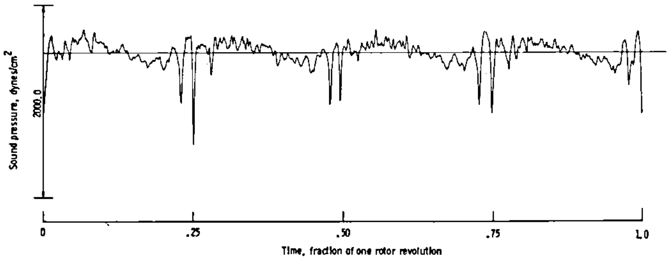
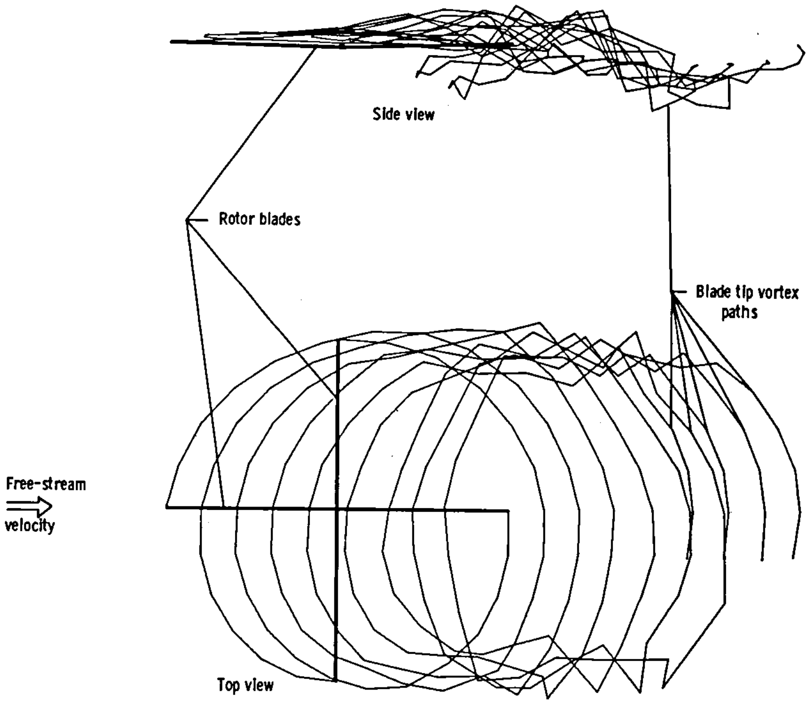


Figure 2.- Sample acoustic pressure-time history with blade-vortex interaction impulsive noise.



Side view
Rotor blades
Blade tip vortex paths
Free-stream velocity
Top view

Figure 3.- Sample rotor-wake geometry calculations.

NONLINEAR SONIC-BOOM PROPAGATION INCLUDING ASYMMETRIC EFFECTS AND SHOCK COALESCENCE

Christine M. Darden*

Experimental sonic-boom studies in recent years have been plagued with the necessity of keeping model size to a maximum of five or six inches. This constraint on model size is necessary because all available propagation methods assume the input to be axisymmetric, and at distances of three body lengths, all disturbances are assumed to be approaching their mid-field, axisymmetric shape. In order to acquire data at three body lengths in currently used supersonic wind tunnels, the size of the model would have to be small.

Recent results in low-boom design studies indicate great promise if design techniques such as wing camber, twist, etc., can be used to meet both low-boom area constraints and aerodynamic performance criteria. Larger models are needed to efficiently incorporate all of these design techniques, resulting in one model which can be tested for both sonic boom and aerodynamic performance. Presently two models are required.

In addition to the size problem caused by available propagation methods, there are also problems associated with high altitudes and high Mach numbers (fig. 1). These methods are all based on modified linear theory and at high Mach numbers and high altitudes the omitted terms in the governing equations become significant, thus, introducing inaccuracies in the results.

A sonic-boom propagation computer program developed under NASA Grant NGL-33-016-119 promises help in several of these weak areas. It will relax the size constraint on the models by allowing three-dimensional effects to be included in the input as illustrated in figure 2. A detailed description of the flow field in the near-field is required. Such a description is provided by wind-tunnel tests or by computer programs which analyze the complete flow field about a body. Shock and flow patterns are then extrapolated to the ground by means of a Modified Method of Characteristics (MMOC). This modification changes outgoing characteristics to a curved shape consistent with Whitham Theory and thus allows step sizes to increase to the size of a body length in the far field. Calculations are restricted to the plane of symmetry beneath the body and all asymmetric effects are felt through second derivatives in "ψ," such as $V_{\psi\psi}$.

Additional benefits of the MMOC program are listed in figure 3. Because of the higher order and entropy terms included in the governing equations, this program also provides more accurate sonic-boom predictions from high altitudes and high Mach numbers. One weakness of the program is its inability to calcuate the coalescence of embedded shocks. This prevents solutions from being obtained when such calculations are necessary.

An in-house effort is now in progress to modify the program, enabling it to calculate shock coalescence for axisymmetric bodies and then for nonaxisymmetric bodies (fig. 4). This feature is needed in the program so that results from realistic low-boom transport designs can be propagated to the ground. Work on the axisymmetric coalescence is complete.

---

*HSAD, 533-01-43-04, 804-827-3181

- STRENGTHS
  - ★ MODIFIED MOC (MMOC)
  - ★ REAL ATMOSPHERE
  - ★ HIGHER-ORDER TERMS
  - ★ ENTROPY VARIANCE ACROSS SHOCKS
  - ★ NON-AXISYMMETRIC TERMS
- WEAKNESS
  - ★ NO SHOCK COALESCENCE ROUTINE

Figure 1. - Features of Available Sonic-Boom Propagation Methods.

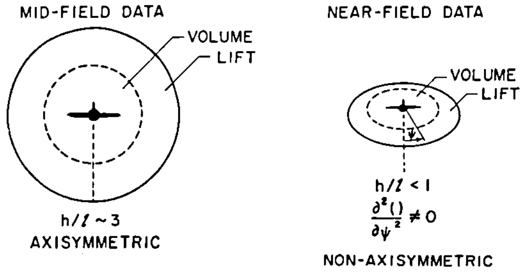

MID-FIELD DATA
VOLUME
LIFT
$h/l \sim 3$
AXISYMMETRIC
NEAR-FIELD DATA
VOLUME
LIFT
$h/l < 1$
$\frac{\partial^2()}{\partial\psi^2} \neq 0$
NON-AXISYMMETRIC

Figure 2. - Input Data Assumptions.

- AXISYMMETRIC INPUT
  - → MID-FIELD INPUT DATA
  - → SMALL WIND TUNNEL MODELS
  - → INABILITY TO MAKE PROPER USE OF CAMBER, TWIST AND OTHER DESIGN TOOLS.
- MODIFIED LINEAR THEORY
  - → INACCURACIES AT HIGH MACH NUMBERS
  - → INACCURACIES AT HIGH ATTITUDES

Figure 3. - MMOC Sonic-Boom Program.

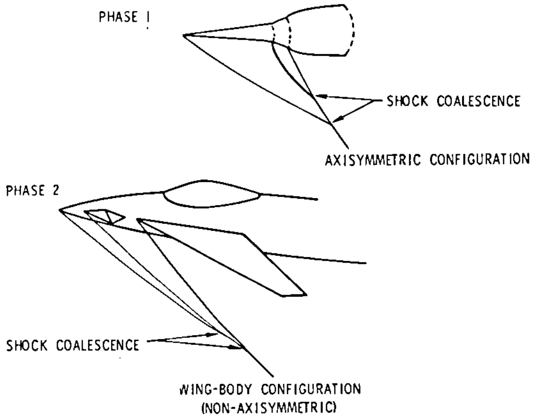

PHASE 1
SHOCK COALESCENCE
AXISYMMETRIC CONFIGURATION
PHASE 2
SHOCK COALESCENCE
WING-BODY CONFIGURATION
(NON-AXISYMMETRIC)

Figure 4. - Features Being Added In-House.

VORTEX MODELING OF TURBULENT FLOWS

Jay C. Hardin*

A recently completed in-house study compared the flow field predicted by a two-dimensional discrete vortex model with experimental data for high Reynolds number turbulent flow over a cavity, both on a time-average and a spectral basis. The data were obtained during an earlier in-house assessment of sound generation by flow over cavities in aircraft surfaces which was conducted in the Aircraft Noise Reduction Laboratory. The present comparative study was motivated by the increasing use of vortex models to investigate the large scale structure in turbulent flows as well as their utility in computation of aeroacoustic noise generation which had been shown in previous in-house and out-of-house analyses. In spite of this widespread interest, only the most elementary comparisons between flow parameters predicted by the vortex models and actual data had been attempted.

Typical results of the comparison are shown on the accompanying montage. In the upper left, a schematic of the experimental geometry is shown. Comparisons are made with flow data taken along the line marked "Measurement Location" which spans the intense shear layer over the cavity near its back wall. In the upper right, the measured mean velocity in the streamwise direction along this line is compared with that predicted by the vortex model. Note the good agreement except down in the cavity where three-dimensional effects become important. In the lower left, the streamwise turbulent intensity profile is similarly compared. Note that the levels are generally reasonable above the cavity but are too high by a factor of two below the top of the cavity. This is thought to be due to the fact that the model has no third dimension into which turbulent energy may be transferred. This would also account for the peak of the profile occurring somewhat lower in the cavity in the model prediction. Finally, in the lower right, the narrowband unnormalized streamwise velocity spectal densities are compared for the point Y/D = 1.35 where the measured and predicted intensities (integral of the spectrum) are nearly equal. Note that remarkably good agreement exists both in level and frequency range, with the model somewhat overpredicting at lower frequencies and underpredicting at high frequencies. This can be explained due to the lack of an energy cascade mechanism in the model. The narrowband spikes which appear in the spectra are the well-known self-sustained oscillation of flow over cavities. The model reproduces this phenomenon slightly shifted in frequency due to the assumption of incompressibility.

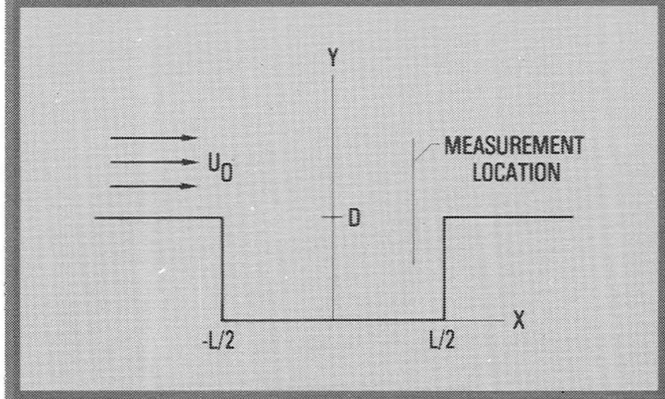
These results are most encouraging for the use of vortex models to calculate noise and to simulate turbulent flow fields. Apparently, even in turbulent flow field, part of the flow is deterministic, "determined" by the initial and boundary conditions on the flow. The most crucial areas for further research are indicated to be the inclusion of three-dimensionality and energy transfer in a computationally efficient manner.

---

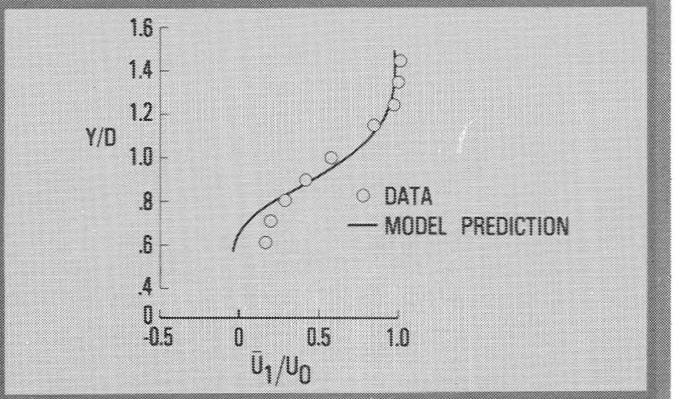
*ANRD, 505-06-23, 804-827-2617

VORTEX MODEL OF CAVITY FLOWS

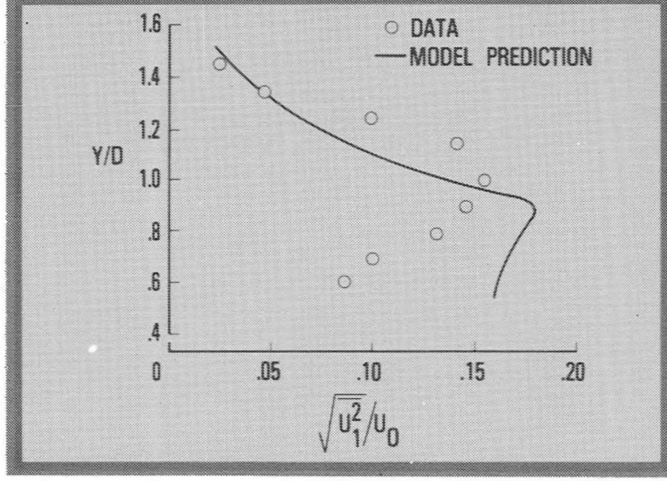
SCHEMATIC OF EXPERIMENT


Y
U_0
MEASUREMENT LOCATION
D
X
-L/2
L/2

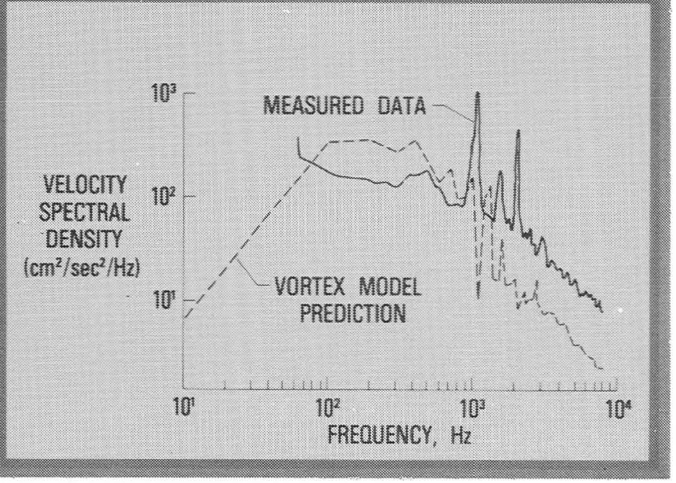
MEAN VELOCITY PROFILE



TURBULENT INTENSITY PROFILE



COMPARISON OF VELOCITY SPECTRA



THE INTERACTION BETWEEN A SOUND PULSE AND A JET SHEAR LAYER

I - FAR FIELD

Lucio Maestrello*, Alvin Bayliss**, Eli Turkel

A code has been developed to compute the fluctuating field of an acoustic source embedded in a jet by solving the full, time dependent Euler equations, linearized about a realistic spreading jet. The code is fully vectorized, and realistic simulation can be obtained efficiently with CDC Cyber 203. Parallel experimental investigations have been conducted, and these confirm the qualitative feature predicted by the code, pointing out a new feature that can be studied.

An amplification of the total acoustic power output is observed when a source is located within the potential flow core of the jet. This amplification occurs in the range of frequencies where the local instability waves have the strongest growth rate. The acoustic power amplification exhibits a peak which is similar to that which is observed both experimentally and analytically for instability waves in an unexcited jet. This is particularly true when the amplification rate is plotted as a function of Strouhal number (fz/U) based on the distance of the source from the nozzle, figure 1.

These results indicate that modification of the stability characteristic of the jet flow can cause significant attenuation of sound. In this regard, it is intended to use the code to study the effect of a plug flow nozzle and minimize the noise radiated from a given source.

References:

1. Maestrello. L.; Bayliss, A.; and Turkel, E. "Experimental and Numerical Results on a Shear Layer Excited by a Sound Pulse" NASA TM 80183, 1979.

2. Maestrello, L.; Bayliss, A.; and Turkel, E. "On the Interaction Between a Sound Pulse with the Shear Layer of an Axisymmetric Jet" Sent for publication to the Journal of Sound and Vibration.

---

*ANRD, 505-32-03, 804-827-2617

**ICASE, 505-31-83-01, 804-827-2513

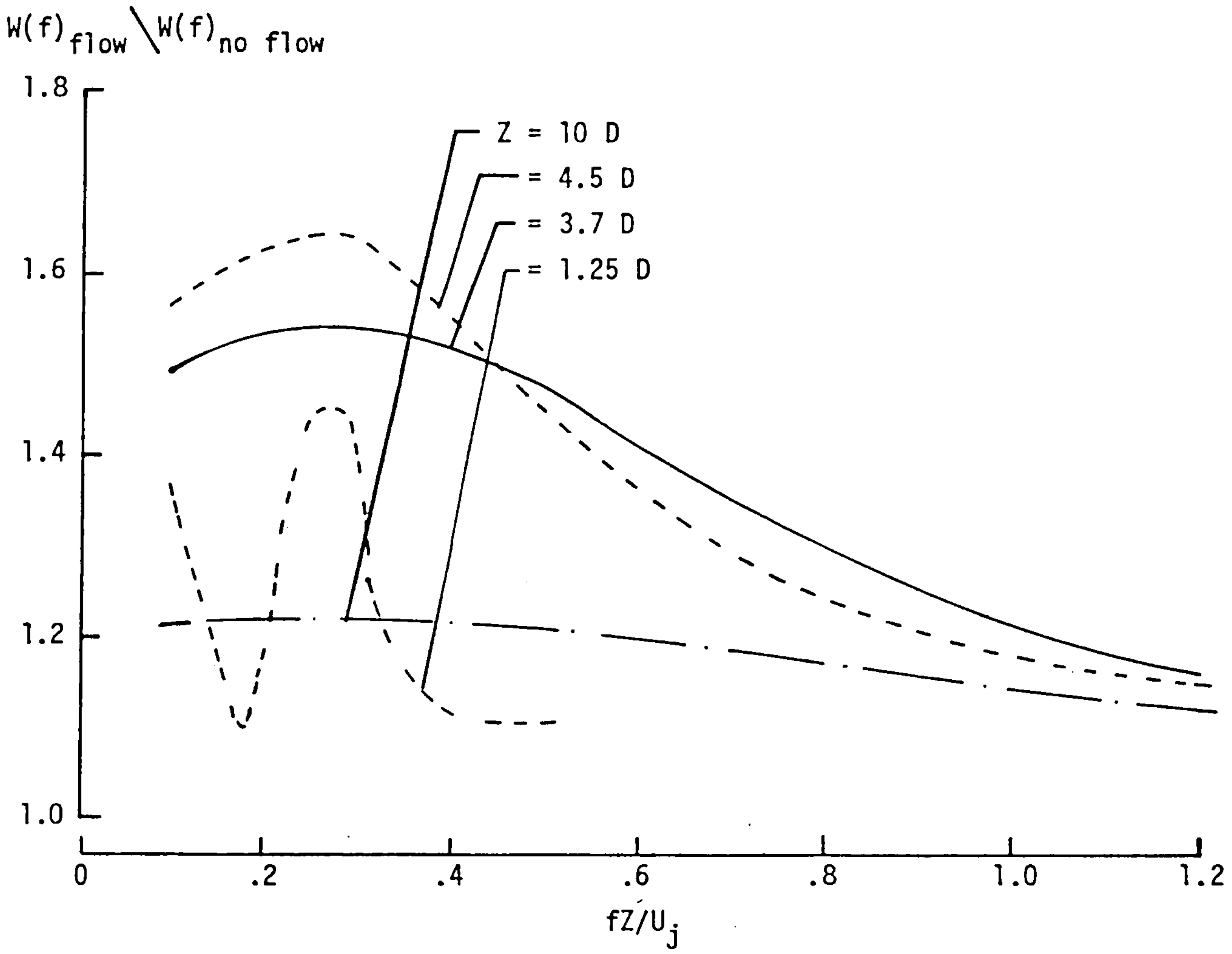


Figure 1. Amplification rate for different source position, M = 0.66 (numerical)

THE INTERACTION BETWEEN A SOUND PULSE AND A JET SHEAR LAYER

II - INFLOW FIELD

Alvin Bayliss*, Lucio Maestrello**

The code developed in Part I has been applied to study the near and inflow fields for the fluctuating perturbation. This work is also being combined with experimental investigation.

The numerical simulation predicts a large amplification of the inflow fluctuating field in a jet excited by a pulse. This is accompanied by an increase in the far-field acoustic intensity. Both phenomena occur primarily in the low to medium frequency range and have been observed experimentally. The frequency of the peak far-field power amplification is found to correspond to the frequency of the maximum growth rate of the instability wave at the position of the source.

The numerical results presented here were obtained by solving a hyperbolic initial boundary value problem. A family of boundary conditions are presented, which enable the fluctuating field to be computed in a computational domain which is localized in the vicinity of the flow. This suggests that the inflow data, computed in a relatively small region near the flow, can be input as the shear noise sources into existing theories of aerodynamic noise to compute the far-field power amplification due to the shear interaction, see for example the variation of the Reynolds stresses and vorticity in figure 1 and the evolution of the inflow pressure in figure 2.

Reference:
1. Bayliss, A.; Maestrello, L. "The Near Field Interaction Between a Sound Pulse and a Jet Shear Layer" Presented at the AIAA 6th Aeroacoustics Conference, Hartford, Connecticut, June 1980.

---

*ICASE, 505-31-83-01, 804-827-2513
**ANRD, 505-32-03, 804-827-2617

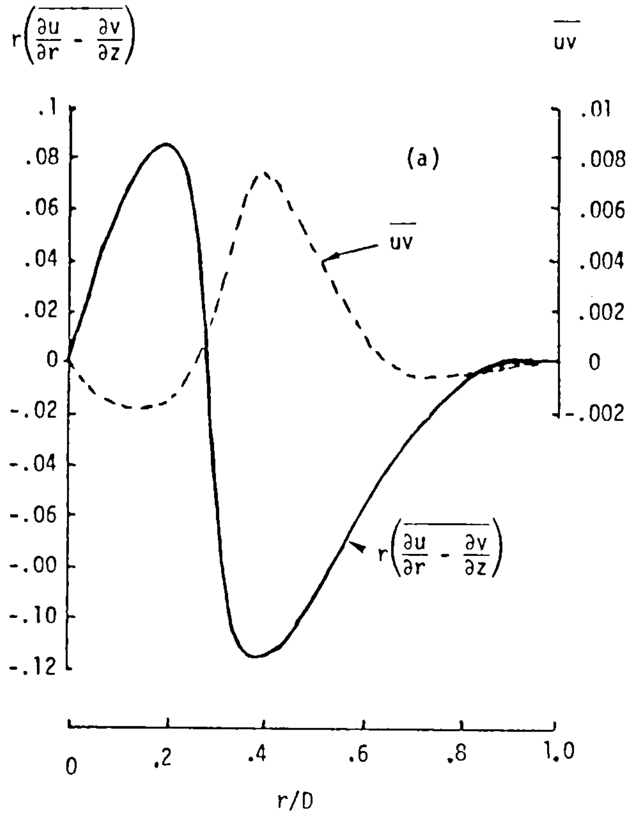


Figure 1. Normalized time average Reynolds stresses and vorticity. z/D = 2.

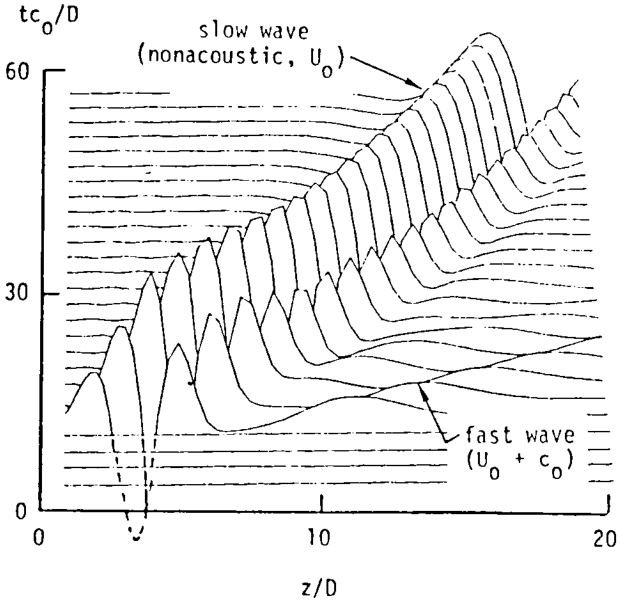


Figure 2. Three-dimensional prospective of the evolution of the pressure amplitude at r/D = 0.

11. Grid Generation

Methods for constructing computational grids for solution of flow problems, and studies of the effects of grid on the numerical solutions.

DEVELOPMENT OF MESH GENERATION TECHNIQUES IN COMPUTATIONAL FLUID DYNAMICS FOR APPLICATIONS TO FLOW FIELD ANALYSES OF AERONAUTICAL COMPONENTS AND SYSTEMS WITH COMPLEX GEOMETRIC CONFIGURATIONS

Peter R. Eiseman*

In an approach to the mesh generation problem, numerous specifications of mesh properties became possible with the development of the multi-surface method [1]. With the multi-surface method, intermediate control surfaces are introduced between bounding surfaces so that the mesh properties can be naturally prescribed by parametric alignment. In particular, the control surfaces, which should not be mistaken for coordinate surfaces, define a discrete vector field which is interpolated, integrated, and normalized to obtain the desired transformation. In the basic study, only polynomial interpolates were considered. The resulting transformations provided all of the desired boundary specifications for the form of the mesh.

In the present study, the multi-surface transformation is derived with interpolants each of which vanishes identically outside of a small region around its corresponding point of interpolation. The size of each region and the shape of each interpolation function is chosen so that certain important properties such as uniformity are admissible. These local interpolants then lead directly to precise local controls on the form of the mesh both along the boundaries and internally. Although local controls are available with methods based upon Poisson equations and with the polynomial version of the multi-surface transformation, the capability for precise internal specifications are not. As an example of the desired degree of precision, suppose that we wish to transform a region with curved boundaries in such a way that most of the interior is covered by a uniform Cartesian mesh. A schematic illustration of such a smoothly embedded Cartesian region is given in the figure below. The smoothness of the embedding means that derivative continuity can be specified up to a desired level so that, for example, finite difference approximations which extend over both Cartesian and curvilinear regions are well represented. A discrete representation of the fluid dynamic equations over the illustrated mesh would then smoothly change from a fully curvilinear form to a simpler Cartesian form over most of the mesh area. With the fully curvilinear portions constrained to regions near boundaries, the simplification in problem formulation is clear. In this example, the previous methods of coordinate generation would not be very successful. However, the multi-surface transformation with local interpolants would be both successful and efficient in the generation of such a mesh.

Reference:
1. Eiseman, Peter R. "A Multi-Surface Method of Coordinate Generation" J. Computational Physics, Vol. 33, (1979), pp. 118-150.

---

*ICASE, 505-31-83-01, 804-827-2513

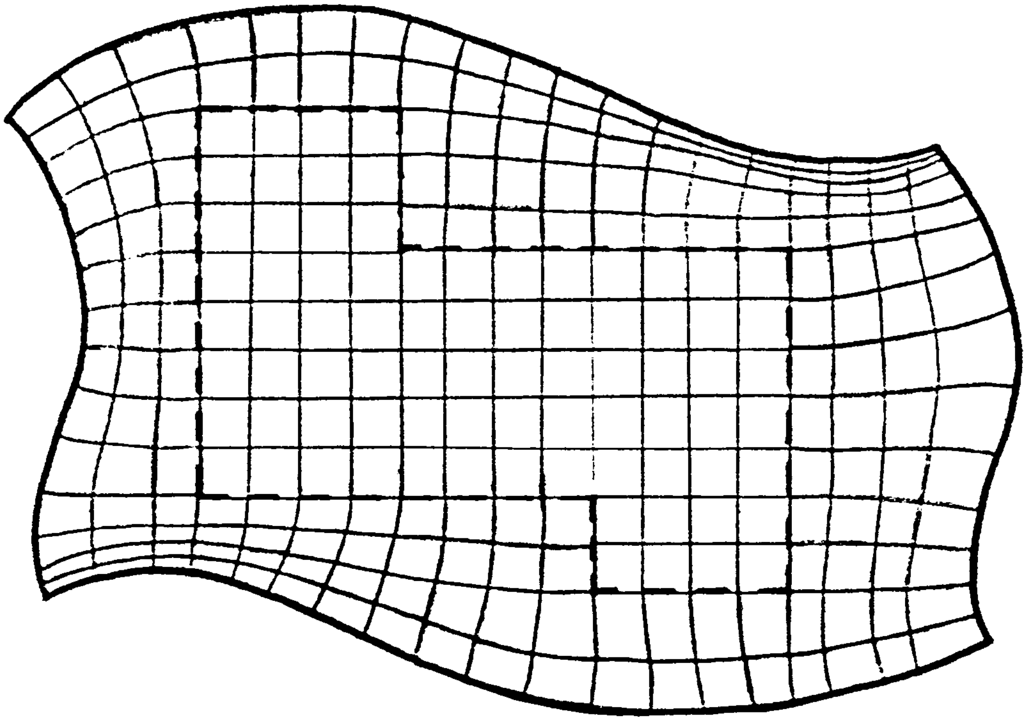


Figure 1. A Coordinate System with a Rectilinear Embedded Cartesian System.

APPLICATION OF A NUMERICAL ORTHOGONAL COORDINATE GENERATOR TO AXISYMMETRIC BLUNT BODIES

Randolph A. Graves, Jr.*

In recent years, a great deal of effort has been expended to develop coordinate transformations and/or mesh generators for varying degrees of geometric complexity. These techniques range from the very simple algebraic manipulators to the more mathematically elegant complex variable transformations. For flow over blunt axisymmetric bodies, it is imperative that the boundary conditions on the body be represented as accurately as possible and in most applications, the body surface is generally used as one of the coordinate lines. This is called the body-oriented coordinate system and has found wide application, however, a major problem with this method is that any discontinuity in slope on the body makes it impossible to describe the complete flow field. Thus, the body oriented system cannot handle a body with a concavity.

To overcome these difficulties, a simple numerical orthogonal coordinate generator is applied to blunt bodies both for total flow fields and for forebody regions only. In this technique, a significant improvement in handling boundary conditions is gained, for not only is the body a coordinate line, but the forebody shock for supersonic flow also is a coordinate line. Thus, the mesh is orthogonal on both boundaries.

The coordinate generator has been applied to a range of blunt bodies both with equal and unequal coordinate spacing. Additionally, the origin of the outer boundary can be displaced to produce a "natural" compression of mesh where needed. Figure 1 shows a typical blunt entry body with a displaced outer boundary and unequal spacing in the normal direction. This technique handles a wide range of body shapes easily and is computationally efficient.

References:
1. Graves, Jr., R. A.: Application of a Numerical Orthogonal Coordinate Generator to Axisymmetric Blunt Bodies, NASA TM 80131, Oct. 1979.

---

*SSD, 506-51-13, 202-755-3277

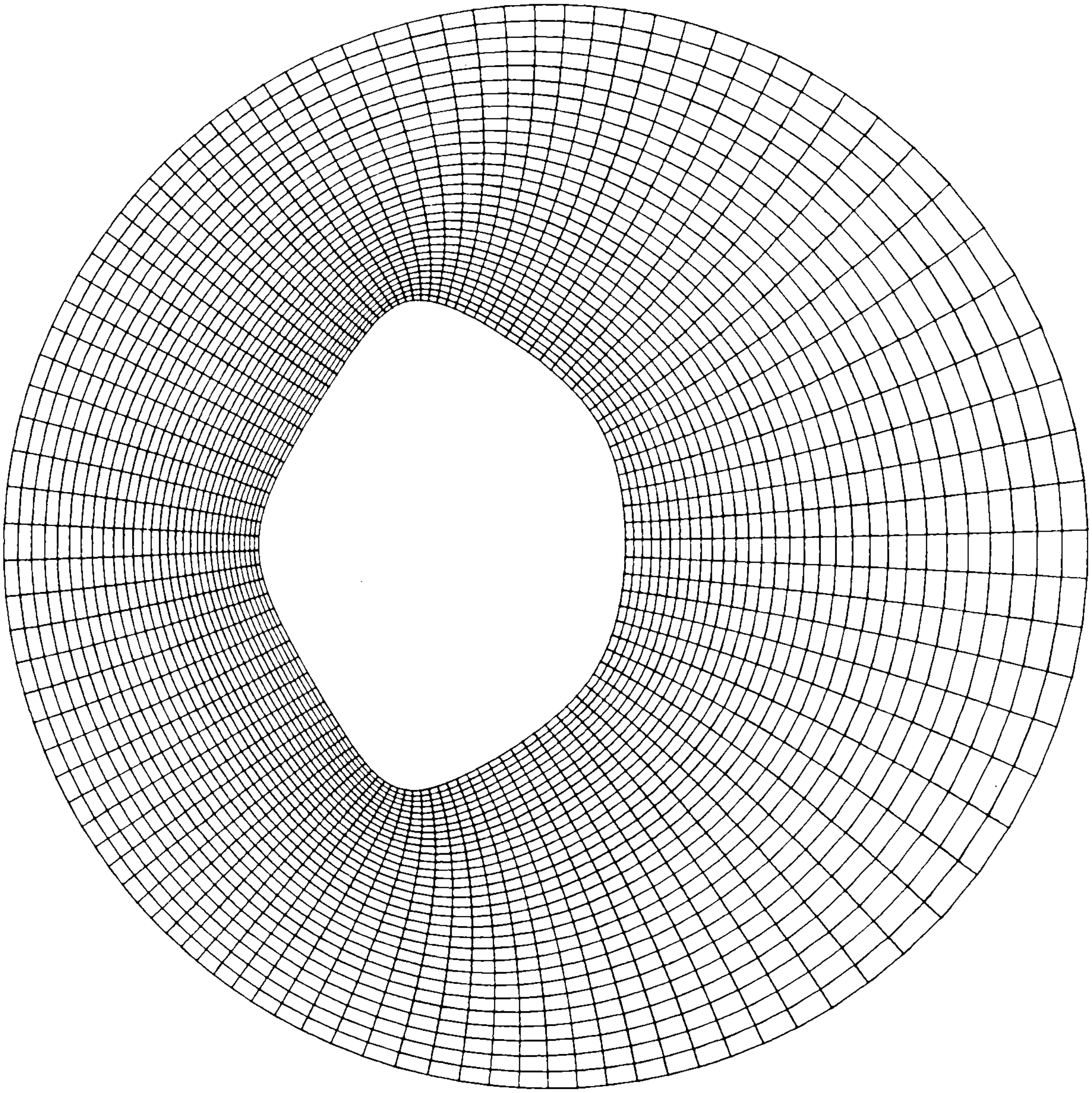


Figure 1 - Orthogonal coordinate system about planetary entry body with displaced outer boundary

Algebraic Techniques for Grid Generation

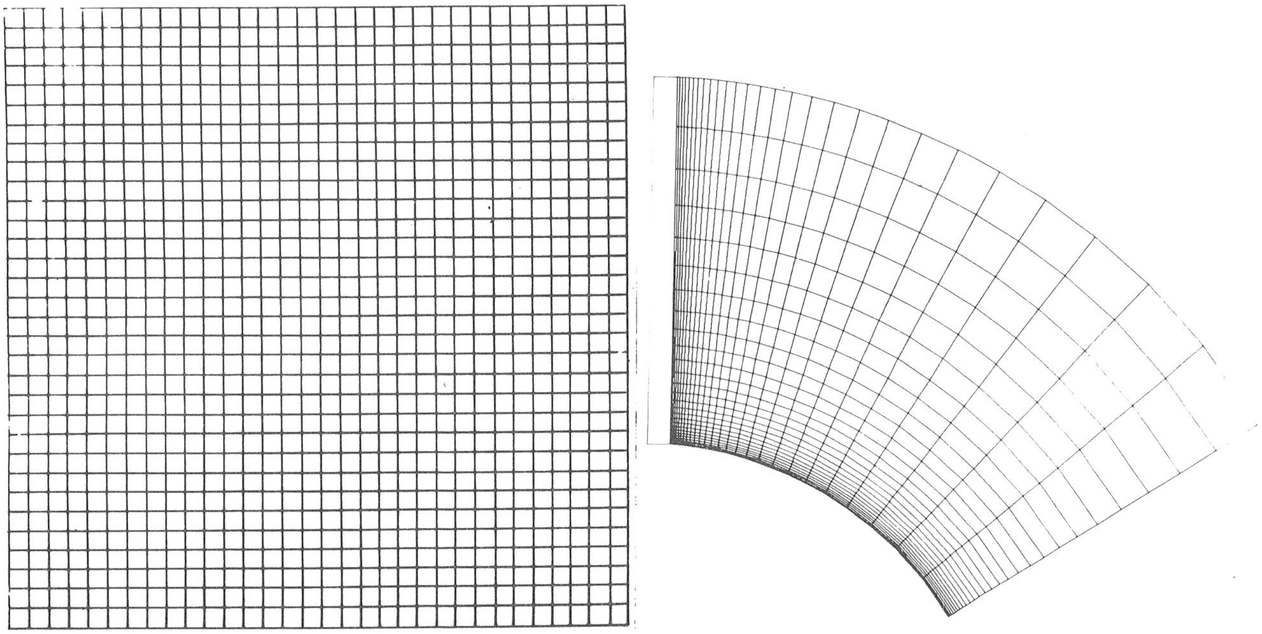
Robert E. Smith

Boundary-fitted coordinate system are well suited for the application of finite difference techniques to the governing equations of fluid flow. Since grid lines lie on boundary surfaces, boundary conditions can be accurately applied and corresponding computer program logic minimized. An algebraic technique for boundary-fitted coordinate systems is one where an explicit functional relation establishes the transformation between a uniform rectangular computational domain and the physical domain. This approach has two advantages - rapid, precise, grid control and availability of analytical derivatives.

An approach for establishing an algebraic relationship between a computational domain and a physical domain is based on defining two disconnected boundaries and a connecting function. This is call the "two-boundary technique" and is described in reference 1. The suggested connecting functions are simple linear and cubic parametric polynomials with parameters based on position and derivatives at the disconnected boundaries. The boundaries can be defined analytically or approximately. A tension spline approximation to the boundaries is one approach. The density of the grid in specified regions is controlled by embedding control functions on the independent parametric variables for the surface definition or on the independent variable for the connecting function. Grids for spike-nosed bodies and wedge-cylinder corners described in Reference 2 and 3 have been generated using this technique. Figure 1 shows the computational domain and the physical domain for one plane of a 6° wedge-cylinder corner. The density solution obtained from a Navier-Strokes solution is also shown.

REFERENCES

1. Smith, R. E. and Weigel, B. L., "Analytical and Approximate Boundary-Fitted Coordinate Systems for Fluid Flow Simulation," AIAA Paper 80-0192, Jan. 1980.
2. Shang, J., Hankey, W. L., and Smith, R. E., "Flow Oscillations of Spike-Tipped Bodies," AIAA Paper 80-0062, Jan. 1980.
3. Smith, R. E., "Numerical Solutions for the Navier-Strokes Equations for a Family of Three-Dimensional Corner Geometries," AIAA Paper 80-1349, July 1980.



Computational Domain

η vs ζ at $\xi = 1$

Physical Domain

y vs z at $x = x_L$

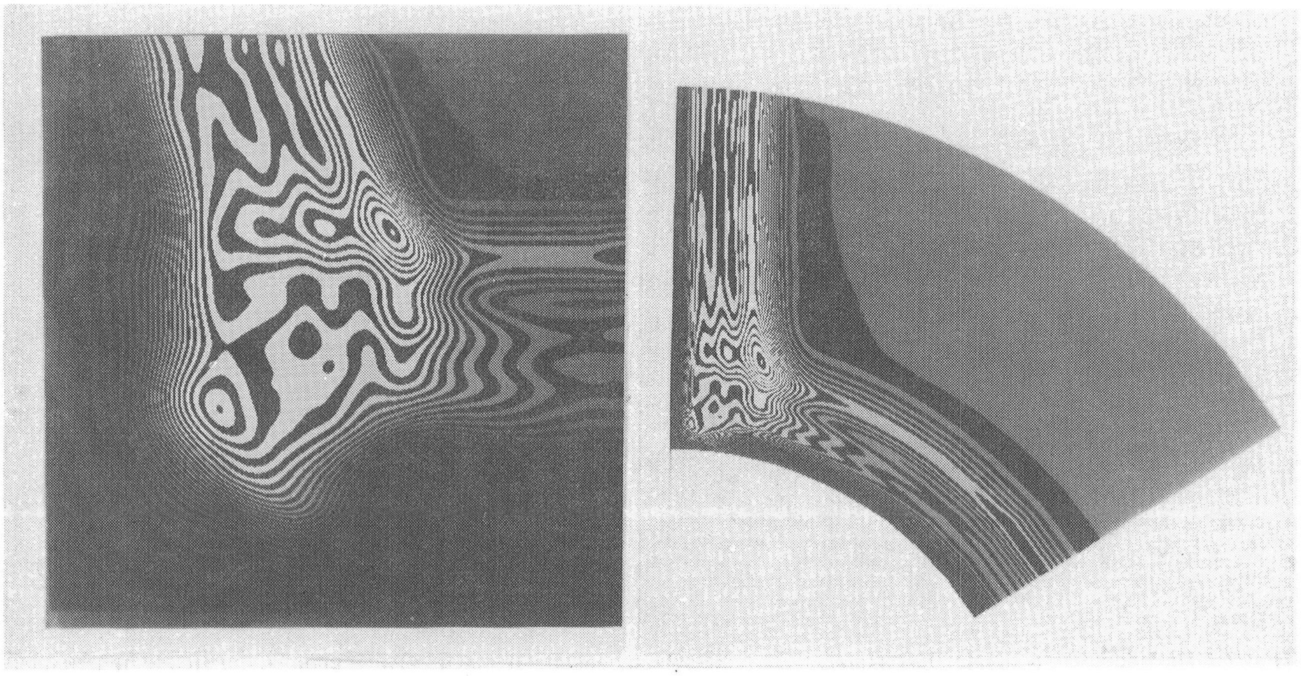


Figure 1 - Density Distribution

EFFECTS OF COORDINATE SYSTEMS ON THE NUMERICAL SOLUTION OF FLUID FLOW

Joe F. Thompson and C. Wayne Mastin*

The efficient solution of a fluid flow problem requires a variable grid spacing which depends on both accuracy and resolution requirements. The concentration of grid points may vary greatly throughout the physical region. Rapid change in coordinate line spacing increases the truncation error in difference approximations and, in the case of viscous flow problems, introduces a numerical viscosity which may swamp the physical viscosity in high Reynolds number calculations.

Taylor series expansions about grid points in the physical region can be used to analyse the truncation error resulting from the classical difference formulations as well as to develop higher order schemes on arbitrary physical regions. Further study could lead to developing optimal coordinate systems.

An example of the error caused by varying the coordinate line spacing and the improvement in using a higher order scheme can be seen in the solution of the following problem. The potential function for ideal flow about a circular cylinder was computed numerically on the coordinate system in Figure 1. In the traditional discretization of Laplace's equation, the physical region is transformed to a rectangular region and the chain rule is used to approximate the second order derivatives. This result is compared in Figure 2 with an approximation obtained by a higher order method using a Taylor series expansion at grid points in the physical region.

Other sources of error when using variable grids are under investigation. For example, there is the problem of error caused by extremely nonorthogonal coordinate lines. In the discretization of the transformed equation on a rectangular coordinate system, the coefficients may be computed numerically or analytically. Preliminary results suggest that the analytic computation is more sensitive to variations in grid spacing. When a Neumann boundary condition must be imposed, as in the above example, there are presently two available options. Either a one-sided difference approximation can be used or a false boundary can be added so that central difference approximations are feasible. Both methods have performed equally well in the above example and in the computation of ideal flow about a Joukowski airfoil.

---

*Mississippi State University, Mississippi State, MS 39762, 601-325-3623

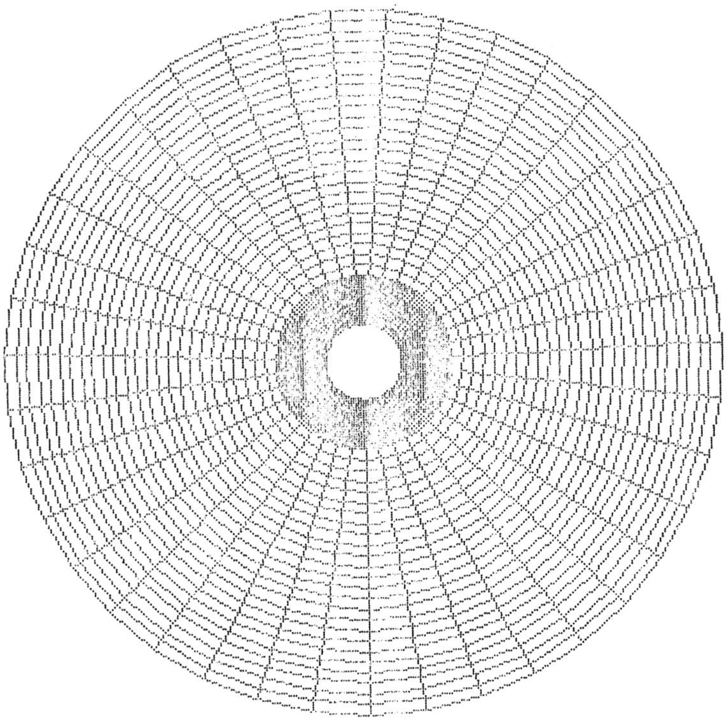


Figure 1. Coordinate system with a five to one jump in coordinate line spacing.

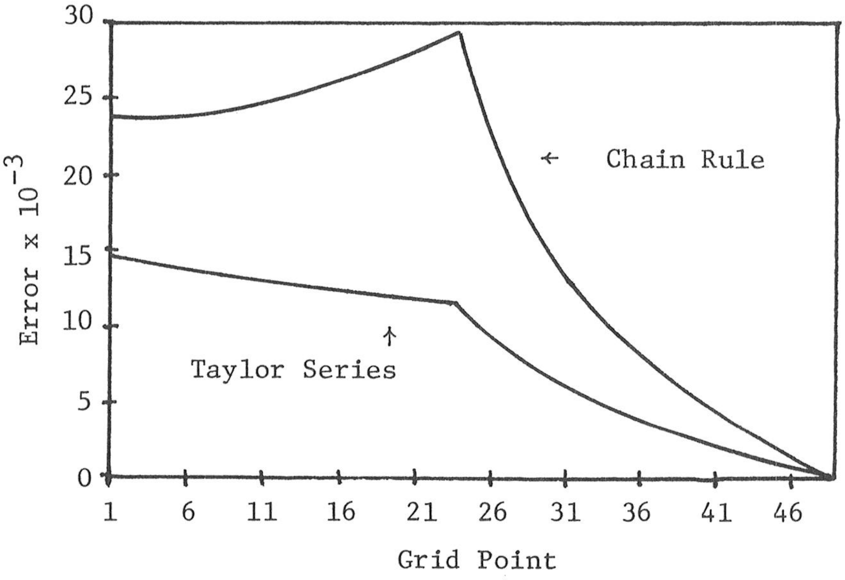


Figure 2. Error at grid points on radial line from rear stagnation point to outer boundary.

TIME DEPENDENT COORDINATE SYSTEMS FOR THE NUMERICAL SOLUTION OF FLUID FLOW

Liviu Lustman
Old Dominion University

Powerful algorithms have been used recently to map complicated geometrical regions over standard squares or cubes. In the corresponding new coordinates, uniform grids may be used efficiently, as well as techniques necessitating simple geometry, such as spectral methods. However, the grid generation is usually effected only once, before starting the solution of the differential system. The coordinates thus obtained are appropriate for boundary layer resolution or sharp corners, but are essentially static and probably not applicable to time dependent problems. It is a serious challenge to design a time dependent grid, which will map a uniform computational mesh, at various times, onto surfaces in the physical domain fitted to wave phenomena - sharp gradients or shock discontinuities propagating in space. Proper bunching of these curvilinear coordinates may solve the shock resolution problem, since - as numerical experience shows - Gibbs phenomena and shock smearing usually occurs not on a fixed length scale, but on a fixed number of mesh points. Clearly, such an ambitious project must overcome several difficulties:

1) Grid generation is time consuming, if done by the elliptic system method of Thompson et al. Simpler geometrical methods are hard to match to flow properties in the physical domain.

2) After being expressed in the new coordinates, the transformed equations will contain new terms due to coordinate evolution, posing stability and accuracy problems.

However, recent results of K. Miller and R. N. Miller show that a very simple moving node algorithm - which is moreover independent of the physical system to be solved - is quite efficient in one-dimensional problems.

I propose to study similarly conceived algorithms for sample two-dimensional computations, in order to derive a gradient sensitive, time dependent coordinate generator, and to test its usefulness for various numerical procedures.

References:

Keith Miller, Robert N. Miller, Moving Finite Elements, to appear

Peter Eisemam - J.C.P 26(1978)307, J.C.P. 33(1979)118

J. Thompson, F. Thames, C. Mastin - J.C.P. 15 (1974)299, J.C.P. 24(1977)274, NASA-CR2729

12. Additional Topics

DEVELOPMENT OF A RAREFIED FLOW-FIELD ANALYSIS USING THE DIRECT SIMULATION MONTE-CARLO METHOD

James N. Moss*

The in-house development of a numerical capability to analyze the flow about planetary and advanced space transportation vehicles for rarefied and transition flow conditions is in progress. Substantial impetus for this development comes from the recent (ref. 1) advocacy for using aerodynamic forces, often at low-density conditions, to capture a vehicle into a closed orbit and/or to modify an orbit. The two techniques being studied for future application are aerobraking and aerocapture.

On the basis of previous investigations of analytical and numerical techniques (refs. 2 and 3), the Direct Simulation Monte-Carlo Technique of Bird (ref. 3) has been selected for application to the aforementioned problem. This method is based on first principles in kinetic theory and satisfies all the conservation laws applicable to gas dynamics, surface and boundary interactions, and molecular collisions. It is a probabilistic technique which produces results that are the numerical equivalent of a solution for the time-dependent Boltzmann equation. The Direct Simulation Method is applicable to three-dimensional steady or unsteady flow with internal and external boundary conditions. Assumptions concerning a starting distribution function are not required and initial simplifying physical assumptions are not required.

Figure 1 presents results obtained by Hauser and Brock (ref. 2) with the Direct Simulation Method for the molecular density surrounding the shuttle configuration. These results are for the shuttle midplane and for the free-stream conditions denoted in figure 1. The normalized density values are those for the free-stream molecules that have been reflected from the shuttle surface (type 2 molecules).

References:

1. French, J. R. and Cruz, M. I., "Aerobraking and Aerocapture for Planetary Missions," Astronautics and Aeronautics, February 1980.

2. Hauser, J. E. and Brock, F. J., "Shuttle Flow Field Analysis Using the Direct Simulation Monte-Carlo Technique," Paper presented at the USAF/NASA International Spacecraft Contamination Conference, March 1978.

3. Bird, G. A., Molecular Gas Dynamics, Oxford University Press (London), 1976.

---

*SSD, Langley, 804-827-3770

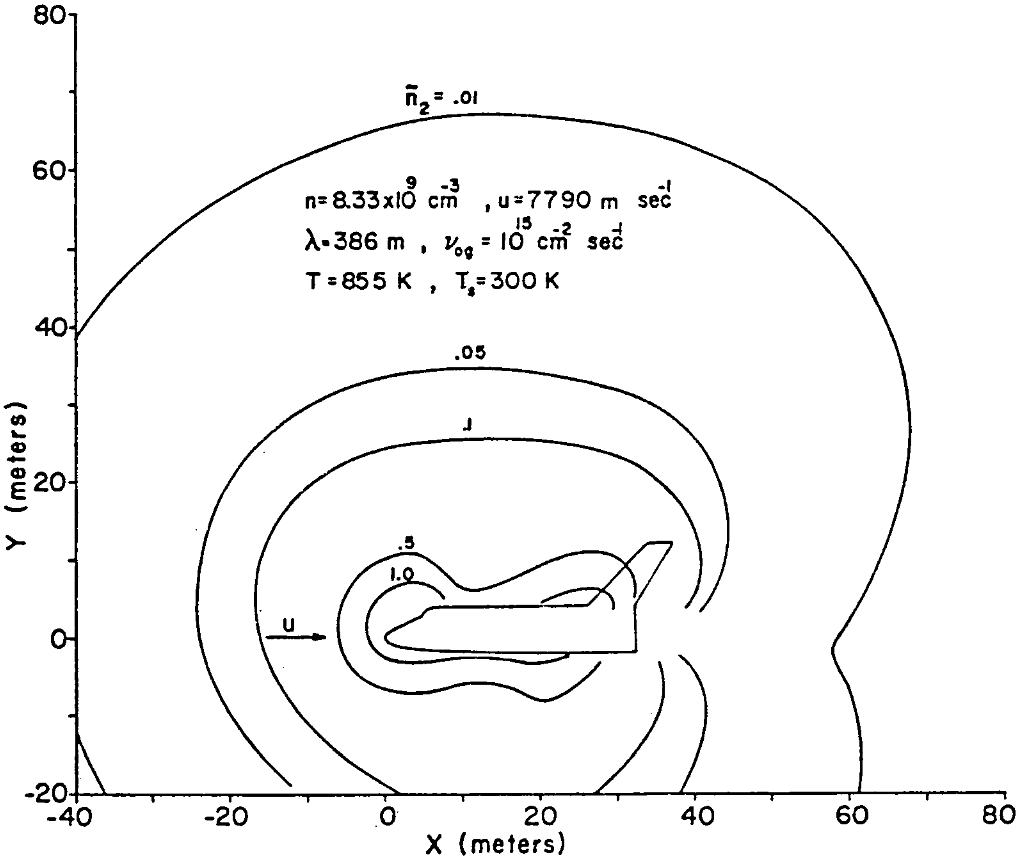


Fig. 1 The density distribution of reflected freestream molecules (type 2) in the Shuttle midplane, normalized by the undisturbed freestream density n=8.33 x 10^9 cm^{-3}. The freestream velocity u=7790 m sec^{-1}, the temperature T=855 K and the mean free path λ=386 m. The Shuttle surface temperature T_S=300 K, and the surface outgassing rate ν_{og}=10^{15} cm^{-2} sec^{-1}.

1. Report No. NASA TM-81877	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle A COMPENDIUM OF COMPUTATIONAL FLUID DYNAMICS AT THE LANGLEY RESEARCH CENTER		5. Report Date August 1980 6. Performing Organization Code
7. Author(s) Staff of the Langley Research Center		8. Performing Organization Report No.
9. Performing Organization Name and Address NASA Langley Research Center Hampton, VA 23665		10. Work Unit No. 505-31-83-02 11. Contract or Grant No.
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		13. Type of Report and Period Covered Technical Memorandum 14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

This document contains a set of summary presentations of research in progress at the Langley Research Center in Computational Fluid Dynamics (CFD). While this document does not include all or even most of the work going on in CFD at Langley, it does display in summary form the general scope and nature of research in this area.

Computational Fluid Dynamics is, of course, an important and substantial research activity at the Langley Research Center. No single organization has the primary responsibility for the CFD effort at the Center, but rather research activities in CFD are carried out within many different organizational units as an integral part of a wide variety of research programs.

The purpose of this compendium is to help identify, through numerous summary examples, the scope and general nature of the CFD effort at Langley. This document will help inform researchers in CFD and line management at Langley of the overall CFD effort. In addition to the in-house efforts and work at ICASE and JIAFS, out-of-house CFD work supported by Langley through industrial contracts and university grants are included. Researchers were encouraged to include summaries of work in preliminary and tentative states of development as well as current research approaching definitive results.

17. Key Words (Suggested by Author(s)) Computational Fluid Dynamics		18. Distribution Statement Unclassified - Unlimited Subject Category 34 64	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 273	22. Price* A12

* For sale by the National Technical Information Service, Springfield, Virginia 22161

NASA-Langley, 1980

LANGLEY RESEARCH CENTER
3 1176 01324 7896