AD-A083 242   AIR FORCE FLIGHT DYNAMICS LAB WRIGHT-PATTERSON AFB OH   F/G 20/4
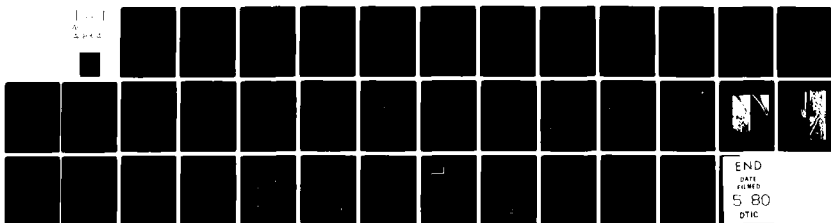   RESEARCH IN COMPUTATIONAL AERODYNAMICS.(U)
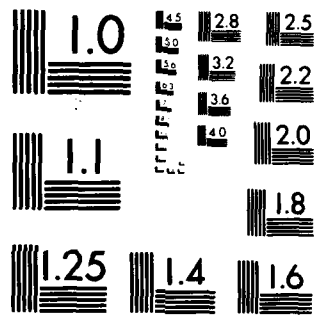   FEB 80   W L HANKEY
UNCLASSIFIED   AFFDL-TR-79-3127   NL

END
DATE
FILMED
5 80
DTIC

IIII 1.0

IIII 1.1

IIII 1.25

IIIII 1.4

||| 1.6

||| 1.8

||| 2.0

|| 2.2

|| 2.5

|| 2.8

| 3.2

3.6

4.0

MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

LEVEL II

②
B.S.

RESEARCH IN COMPUTATIONAL AERODYNAMICS

W. L. HANKEY

COMPUTATIONAL AERODYNAMICS GROUP
AERODYNAMICS AND AIRFRAME BRANCH
AEROMECHANICS DIVISION

FEBRUARY 1980

DTIC
SELECTED
APR 2 2 1980
E

AIR FORCE FLIGHT DYNAMICS LABORATORY √
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
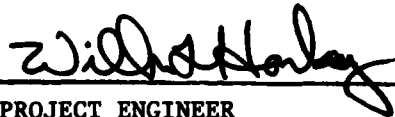WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

80   4 21 085

PROJECT ENGINEER

ERIC K. LINDBERG, Maj, USAF
Chief, Aerodynamics & Airframe Branch

FOR THE COMMANDER

PETER J. BUTKEWICZ, Colonel, USAF
Chief, Aeromechanics Division

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER AFFDL-TR-79-3127	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) RESEARCH IN COMPUTATIONAL AERODYNAMICS		5. TYPE OF REPORT & PERIOD COVERED FINAL REPORT, Jul 75 - Sep 79 6. PERFORMING ORG. REPORT NUMBER 2307N418
7. AUTHOR(s) W. L. HANKEY		8. CONTRACT OR GRANT NUMBER(s) IN4
9. PERFORMING ORGANIZATION NAME AND ADDRESS COMPUTATIONAL AERODYNAMICS GROUP AERODYNAMICS AND AIRFRAME BRANCH AEROMECHANICS DIVISION		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS 2307N418 38
11. CONTROLLING OFFICE NAME AND ADDRESS AEROMECHANICS DIVISION AIR FORCE FLIGHT DYNAMICS LABORATORY WRIGHT-PATTERSON AFB, OHIO		12. REPORT DATE February 1980 13. NUMBER OF PAGES 38
14. MONITORING AGENCY NAME & ADDRESS(If different from Controlling Office)		15. SECURITY CLASS. (of this report) Unclassified 15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Computational Aerodynamics
Viscous Flows
Navier-Stokes

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)
Experience obtained by the Computational Aerodynamics Group, WPAFB, in numerically solving viscous flow problems with the Navier-Stokes equations shall be addressed. Topics discussed include grid generation for arbitrary geometries, numerical instabilities, turbulence models, accuracy, efficiency, and smearing of discontinuities. Applications are reported for the method to solve shock wave-boundary layer interactions, 3D interactions, inlets and nozzle flow fields and unsteady viscous flows.

DD FORM 1473 1 JAN 73   EDITION OF 1 NOV 65 IS OBSOLETE

FOREWORD

This report covers the research in computational fluid dynamics accomplished in the USAF Flight Dynamics Laboratory during the period July 1975 to September 1979 under Work Unit 2307N418. These results are due primarily to the efforts of the following individuals under the direction of Dr. W. L. Hankey:

Dr. J. S. Shang
Maj. D. S. McRae
Dr. D. D. Knight

Capt J. K. Hodge
Maj G. S. Bluford
Lt J. D. Waskiewicz

Accession For		
NTIS GRA&I		✓
DDC TAB		☐
Unannounced		☐
Justification		
By		
Distribution/		
Availability Codes		
Dist	Avail and/or special	
A		

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

I. INTRODUCTION

In developing computer programs to numerically solve the Navier-Stokes equations, the purpose of the computation must be clearly kept in mind. In the Air Force Flight Dynamics Laboratory, our purpose is to provide design information on "non-linear" aerodynamic phenomenon for aircraft that perform throughout the flight corridor. This translates into requirements for a computer program which can solve the time averaged compressible Navier-Stokes equations (with a turbulence model) in three dimensions for generalized geometries. The intended application of the results then establishes the priorities in addressing critical issues.

In our investigations of viscous flows, several problem areas keep recurring. They are as follows:

        1) Grid generation for arbitrary geometry

        2) Numerical difficulties

        3) Turbulence models

        4) Accuracy and efficiency

        5) Smearing of discontinuities

A. GRID GENERATION FOR ARBITRARY GEOMETRY

It is generally accepted that viscous flow problems require a surface-oriented coordinate system. Also for arbitrary geometries, automation of a numerical transformation (as opposed to an analytic transformation) is necessary. In addition, some optimization of the distribution of grid points throughout the flow field is necessary to economically solve practical problems. Conceptually, this implies that higher order derivatives (in the transformed plane) of the primary dependent variable be minimized.[1]

The distribution of the grid points greatly influences the requirement of the number of field points necessary to achieve a desired accuracy. Considerably more attention is needed in this area to improve the economics of the viscous flow computations.

B. NUMERICAL DIFFICULTIES

This is a "catch all" term to cover the reasons a program "bombs out". Given a proven algorithm and an experienced user with a properly formulated problem, program failures are still common during the inital phase of the investigation. The problems are most frequently due to large truncation errors which eventually swamp the true solution. The cause of the problem is that the grid cannot truly be established until the flowfield is determined. A redistribution or increase in the number of grid points often permits success. Artfully changing the damping coefficients in the region of discontinuities has also been successful. In addition, alternate approaches for expressing the boundary conditions can have a dramatic effect on the success or failure of a problem. A requirement exists for a method in which the flowfield modifies its own numerical grid where needed (adaptive mesh generation). Also, additional program guidelines are needed to insure a more robust code.

C. TURBULENCE MODELS

In time-averaging the Navier-Stokes equations information is lost. Information must be re-inserted into the governing equations by resorting to experimental observation. The engineer needs empirically determined transport properties to proceed with the numerical computation. A large body of data exists for flat plate boundary layers and good correlations

2

have evolved which generally permit calculations to be performed that fit the data to within $\pm 10\%$ for skin friction and boundary layer thickness[2] (see Fig. 1). Unfortunately, the agreement for the pressure gradient case is not nearly as good. Higher order closure schemes have not greatly improved the prediction capability. There is a need for the measurement of turbulent Reynolds stresses under pressure gradient for a wide range of flow conditions to permit correlations comparable to the flat plate case. Without this data, progress in the field will be limited.

Many skeptics are pessimistic of our ability to compute turbulent flows in the near future. Turbulence is felt to be too complex and the progress has been slow in developing a thorough understanding. To counter these skeptics, an encouraging viewpoint is offered. First, the good design predictions of flat plate properties are possible without fully understanding the true mechanism of turbulence. Secondly, in some cases it may be possible to bracket the extremes of flows with pressure gradient by computing the frozen and equilibrium states[3], thereby, providing useable design information (Fig. 2). Thirdly, remarkable results are possible[4] in the prediction of gross turbulent properties by simply treating the eddy viscosity Reynolds number as a constant ($\frac{\rho U x}{\varepsilon} = Re_t = const$). Turbulence is limited and confined, and these approximate results are easy to compute; the difficulty is in reducing the error bounds to satisfy the scientist. Fourthly, in most applications, only displacement effects which influence the pressure distribution (separation point location) are significant. Skin friction and heat transfer, which require greater numerical resolution, are often of secondary importance. Therefore, there should be hope that it may

3

be possible to devise a turbulence model capable of producing useful design information for many engineering applications.

One last point concerning the future development of turbulence models. The models to date have been analytical and thus "analog" in nature. New models have an additional requirement to be compatible with numerical computation. We need something like "digital turbulence."

D. ACCURACY AND EFFICIENCY

Accuracy and efficiency should be addressed concurrently because of their interrelationship. Given a stable algorithm, the greatest control on spatial accuracy is the number and distribution of grid points. Figure 3 shows the error in drag coefficient vs number of grid points[10] in one coordinate direction in an airfoil flowfield. The computational time increases with N^2 (for a two dimensional problem) and hence it is very expensive to obtain the last few percent accuracy. The accuracy requirements of any design problem must be very carefully defined in order to avoid excessive computer cost.

Once satisfactory spatial accuracy is achieved, a convergence criteria must be selected which produces comparable accuracy. A time dependent approach is generally used to solve the Navier-Stokes equations in which the computation proceeds from an arbitrary initial condition until a steady state solution is achieved. In the past, several (maybe 5) characteristic times $(\frac{L}{U_\infty})$ has been sufficient for the initial transient to decay. However, based upon the analytical solution of an impulsively started flat plate, the error between the transient value and steady state decays as $t^{-\frac{1}{2}}$. This slow convergence rate implies that to cut the error in half, the computer time must be increased by a factor of four[5] (for the same Δt). (See

4

Fig. 4.) Another discouraging aspect is that for some flows, periodic values are legitimate steady state solutions. For example, subsonic airfoils near stall shed vorticies in a regular manner[6] (Fig. 5). Computations must be accomplished for many characteristic times to achieve mean and rms values for design application. Slow convergence could well be our most critical problem in our goal to economically produce aerodynamic design data.

Paramount to all of these issues is the fact that a good finite difference algorithm must be used to solve the governing equations. Considerable success has been achieved with MacCormack's method[7] to solve supersonic viscous flows. MacCormack's explicit method possesses many desirable features with the exception of efficiency. The CFL stability limit requires small time steps where small spatial steps are required to resolve viscous regions. To relieve this restriction, implicit methods have been developed which are conceptually unconditionally stable. However, our experience shows a gain in efficiency only in the viscous region. Accuracy (not stability) requirements in the inviscid region can be achieved only for the CFL time step. Hence the hybrid method[5] (explicit in the inviscid and implicit in the viscous region) is at present probably the most efficient method available.

E. SMEARING OF DISCONTINUTIES

In examining viscous flow problems, two scale lengths appear. One is the mean free path, $\lambda \sim \frac{\nu}{V}$; the other, which is introduced through the boundary conditions, is a characteristic geometric length, L. One can also derive another scale length, $\delta \sim \sqrt{L \lambda}$; which is a combination of the previously mentioned two lengths. In numerically solving any viscous

flow problem, the grid size, Δy, should be sufficiently small to accurately resolve these three scale lengths (L, δ, λ). This, of course, is impossible to achieve in nearly any practical problem today. Slip lines, shock waves and leading edge flows are examples where the characteristic lengths are too small to be honored. As a consequence, these discontinuities are incorrectly computed. Large errors exist in the immediate vicinity of these regions and numerical smearing results. Based on both wind tunnel and computational experience, it is believed that these local errors near singularities do not totally invalidate the global results. Figure 6 shows a Navier-Stokes computation[9] of a high speed inlet flow indicating good agreement with experiment with the exception of the shock jump and the entropy layer generated by the cowl lip leading edge. More effort is required to minimize the smearing of these discontinuities.

F. APPLICATIONS

The principal application areas in computational fluid dynamics during this reporting period can be generally grouped into four categories, i.e., (1) shock-turbulent boundary layer interactions (2) 3-D interactions (3) inlets and nozzle flowfields and (4) unsteady viscous flows. These research investigations shall now be discussed.

6

II. SHOCK WAVE-TURBULENT BOUNDARY LAYER INTERACTIONS

The general features of shock wave-boundary layer interactions were
identified by Dr. Hans Liepmann in 1940's. Analytical predictions were
accomplished in the 1960's by simultaneously coupling the laminar boundary
layer equations with simple supersonic wave theory, thereby successfully
describing the interaction process. This elementary method encountered
numerical difficulties when extended to turbulent boundary layers, hyper-
sonic flows or complex geometries. For that reason, the time dependent
Navier-Stokes equations with an algebraic turbulence model were utilized
to solve the turbulent flow case[11,12,13,14]. Nearly instantaneous success
resulted in that the numerical difficulties were overcome and the global
features of the interaction described (Fig 7 and 8). It was discovered,
however, that the equilibrium turbulence model was inadequate to accurately
predict the size of the separation bubble or the skin friction level in
the reattachment region. However, the feasibility to numerically compute
these flowfields was clearly demonstrated for the first time. Current
research is now being directed to improve the computational efficiency
and to improve the accuracy of the turbulence model for adverse pressure
gradients[15,16].

III. THREE DIMENSIONAL INTERACTING FLOWS

Early work with the Navier-Stokes equations was for simple 2D geometries. In order to advance the method for practical design applications an effort was undertaken to solve 3D configurations. Two approaches were taken. First, conical (quasi-three dimensional) flows were examined and secondly the full three dimensional Navier-Stokes equations were computed.

Conical flows possess three velocity components as a function of only two spatial variables, and hence introduce some features encountered in true 3D flowfields. Only supersonic inviscid flows over conically symmetric bodies are exactly conical. However, in regions where the radial derivatives of the viscous terms are small compared with the transverse derivatives a "locally-conical" approximation can be performed (Fig. 9). In this manner the flowfields over cones[17,18] and delta wings[19,20] at angles of attack were computed. Excellent agreement was obtained for these conical flows when compared with experiment (Fig. 10 and 11).

Efforts in computing fully 3D flows concentrated on axial corner, shock wave-boundary layer interactions primarily because of the availability of detailed experimental flowfield data. To prove feasibility comparisons must be performed with experimental results which have sufficient detail to outline the essential flow features. The axial corner problem possessed such data for both laminar and turbulent supersonic and hypersonic conditions. Computations for these cases indeed proved the feasibility of the approach[21,22,23] (Fig. 12). For the cases computed, the insensitivity of the turbulence model for 3D flows was observed (as opposed to the opposite for 2D flows). The cases investigated may not have been severe enough to display a large discrepancy, however.

8

The need for improved computer efficiency was clearly displayed in computing these cases and hence the research on 3D Navier-Stokes was directed toward the use of vector processing computers. The 3D time dependent Navier-Stokes program was converted to operate on the CRAY-1 computer with the assistance of the University of Michigan staff under the direction of Professor D. Calahan. On September 1978 a vectorized version ran at a speed of 128 times the CDC 6600 computer[24] (Figure 13). This accomplishment was a major step towards the achievement of practical 3D flowfield computations for design application (Fig 14).

IV. INLETS AND NOZZLE FLOWFIELDS

With the foundation laid to compute viscous flows with shock waves for simple shapes, efforts focused on refining the code to solve practical configurations with complex geometries. Computers possess insufficient storage to solve the flowfields for complete configurations, however, air-craft component flows can be resolved. Components requiring considerable design attention are inlets and nozzles. Attention was, therefore, focused in this area.

Following the guidelines developed by Dr. Joe Thompson, a numerical transformation from a body-oriented coordinate system in the physical plane to a unit square (with constant step size) in the transformed plane permits a completely general approach for resolving any configuration[25,26,27,28].

The overall objective of this approach is to establish a versatile generalized method, simplify the prescription of the boundary conditions, and minimize truncation error. The first coupling of the compressible Navier-Stokes equations with Thompson's transformation was accomplished for a chemical laser diffuser by Knight and Hankey[29]. Others quickly followed so that this approach is presently the most accepted proce-dure[30,31,32,33] (Fig 6). In the future, the method must be refined to include adaptive mesh generation, local orthogonality and automated smooth-ing of the transformation derivatives.

General grid generation, stable shock capturing methods, effective turbulence models and stable boundary conditions are necessary features requiring attention in computing inlets and nozzle flowfields.

V. UNSTEADY VISCOUS FLOWS

Solutions to the Navier-Stokes equations are obtained by utilizing the time-dependent terms and computing from free stream initial conditions until steady state results are achieved. Therefore, this approach has the capability of analyzing some unsteady flows (Fig 15). A class presently under investigation is the self-excited flow (i.e., buzz, buffet, resonance). These flows are generally the result of an unstable separated shear layer (Fig 16) which grows until a "limit cycle" is approached[34,35]. If the reduced frequency parameter is less than unity it is believed that accurate solutions are possible utilizing the Reynolds-averaged Navier-Stokes equations. Good agreement with experimental results has been obtained for a pressure oscillation in an open cavity[36] (Fig 17). This area shows considerable promise for future payoff in analyzing unsteady flow problems of interest to the Air Force.

VI. CONCLUSION

Although additional research is required, we believe outstanding
advances have been accomplished in Computational Aerodynamics. The ground
work has been laid in that stable algorithms exist, mesh generators for
fairly arbitrary shapes have been used, turbulence models are available
that produce global results, and the use of vector processors has demon-
strated a rapid speed up of the codes. Although the feasibility has been
displayed, much improvement is needed in all areas before the practically
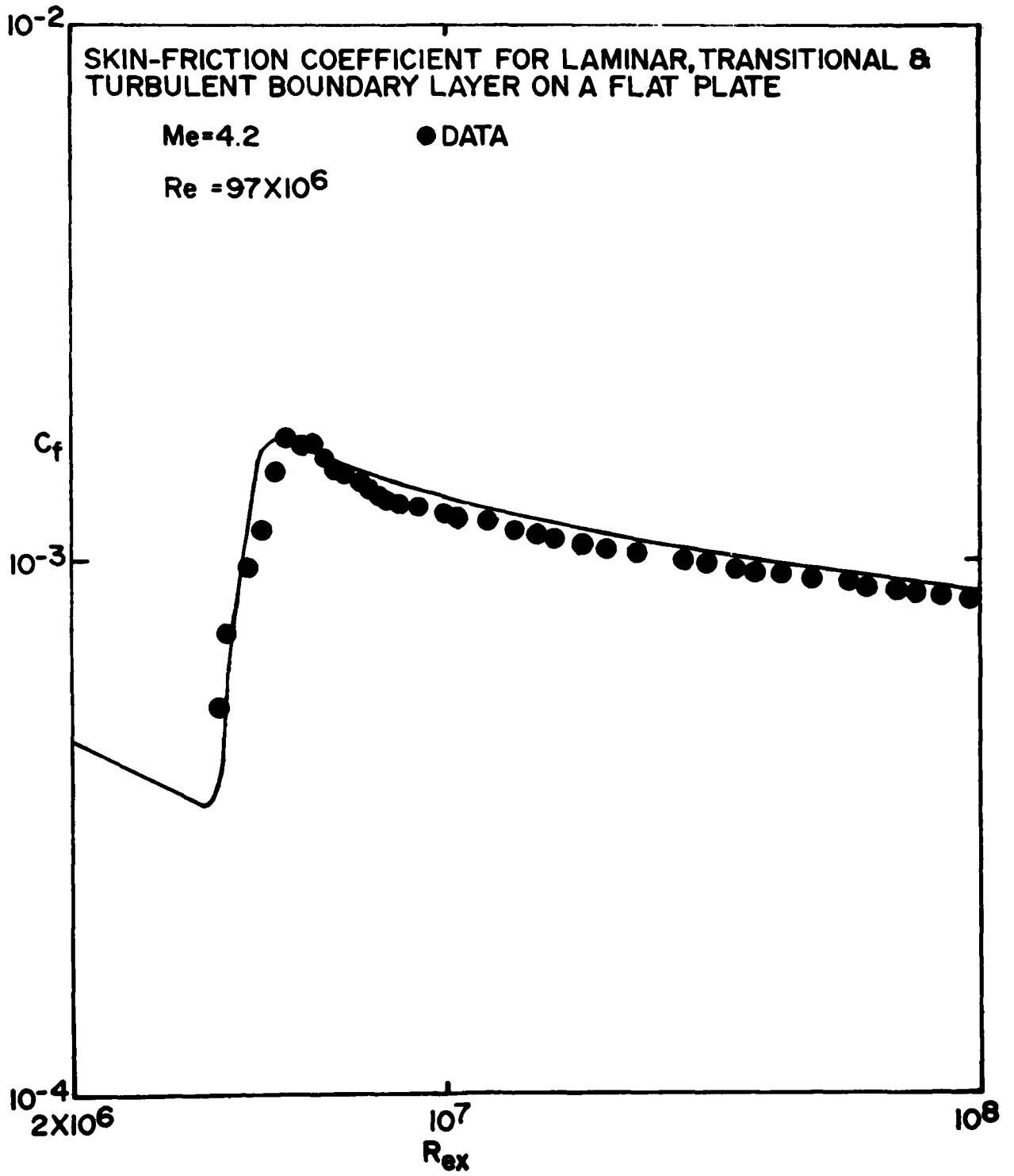of computational fluid dynamics is achieved.

Figure 1. Comparison of Skin Friction
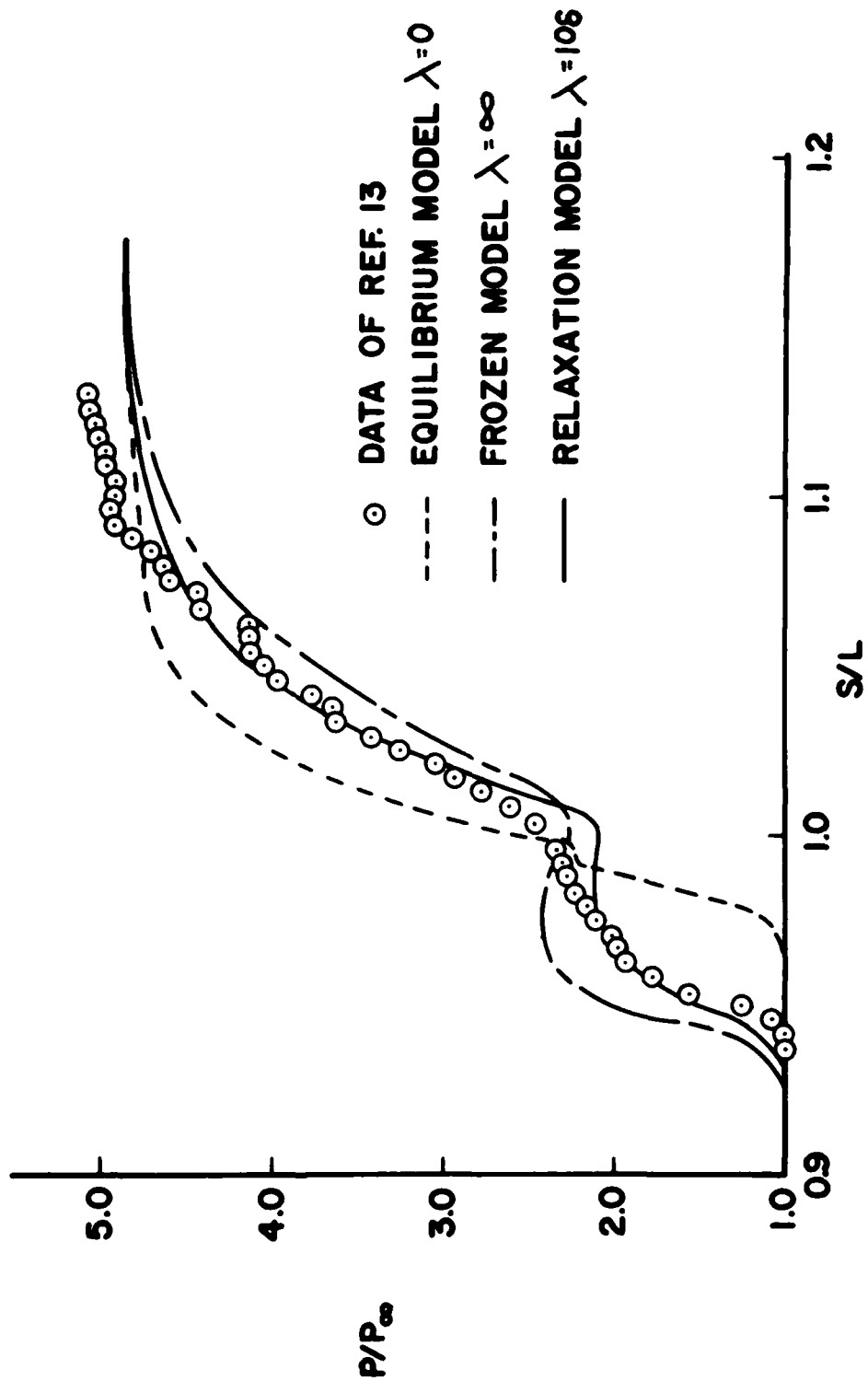Values for a Flat Plate
Boundary Layer

13

Figure 2.   Comparison of Frozen and
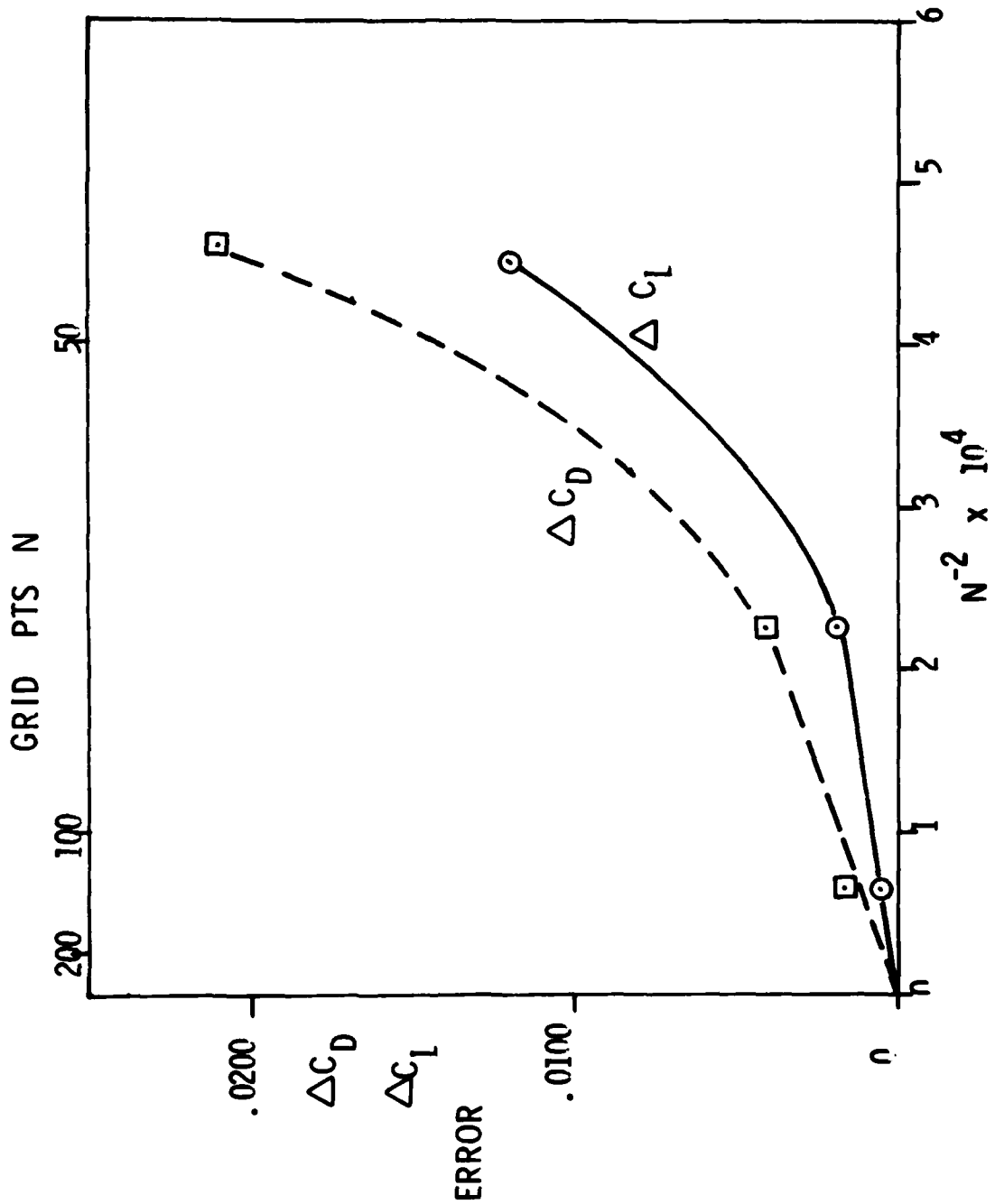Equilibrium Turbulence Models
for Compression Ramp

14

Figure 3. Error in Force Coefficients
vs. Number of Grid Points for
Inviscid Flow Over an Airfoil

15
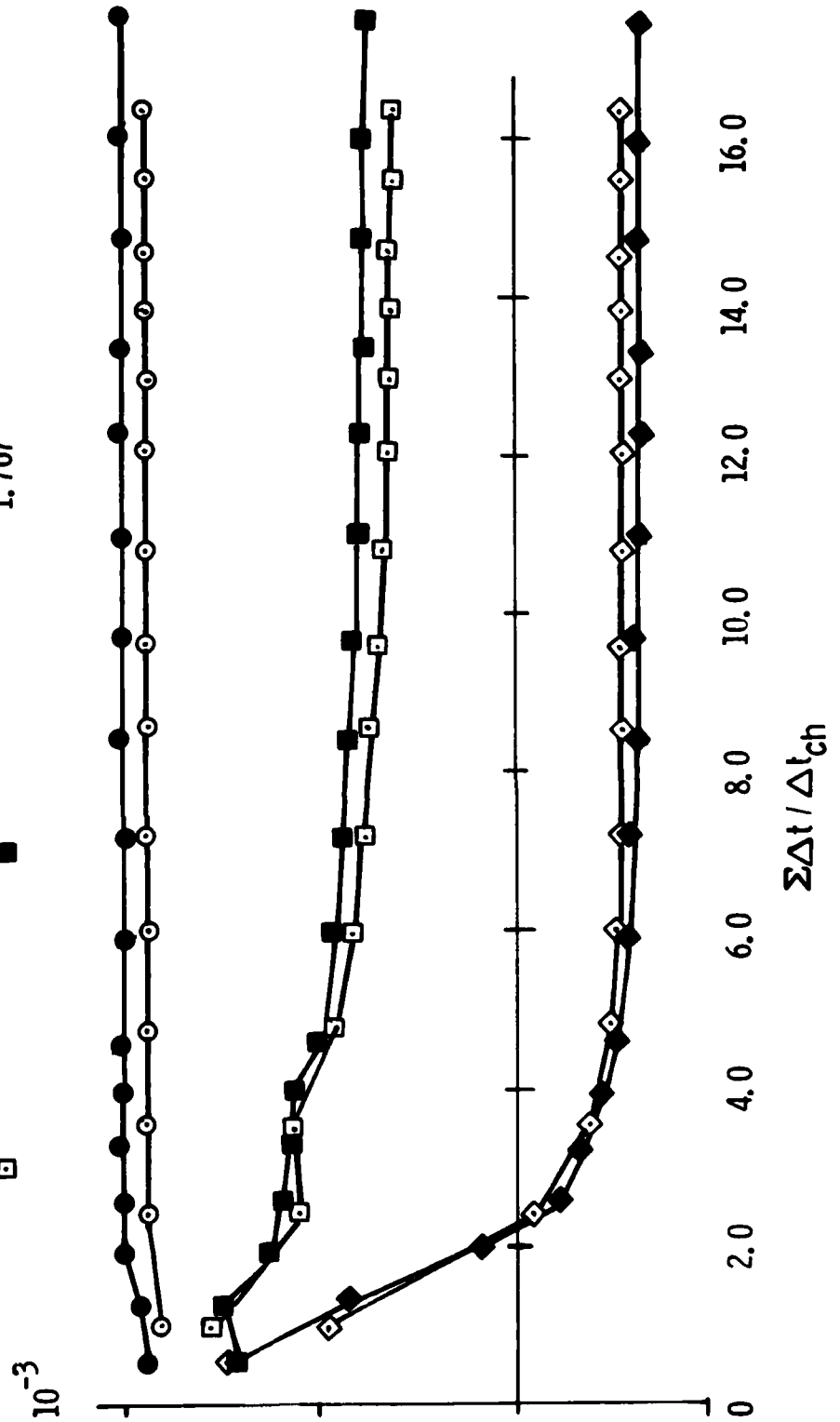
Figure 4. Convergence Characteristics for
Shock Wave Impingement

16

PREDICTED DRAG AND LIFT TIME HISTORY
NACA 6412 ANGLE OF ATTACK = 10 DEG
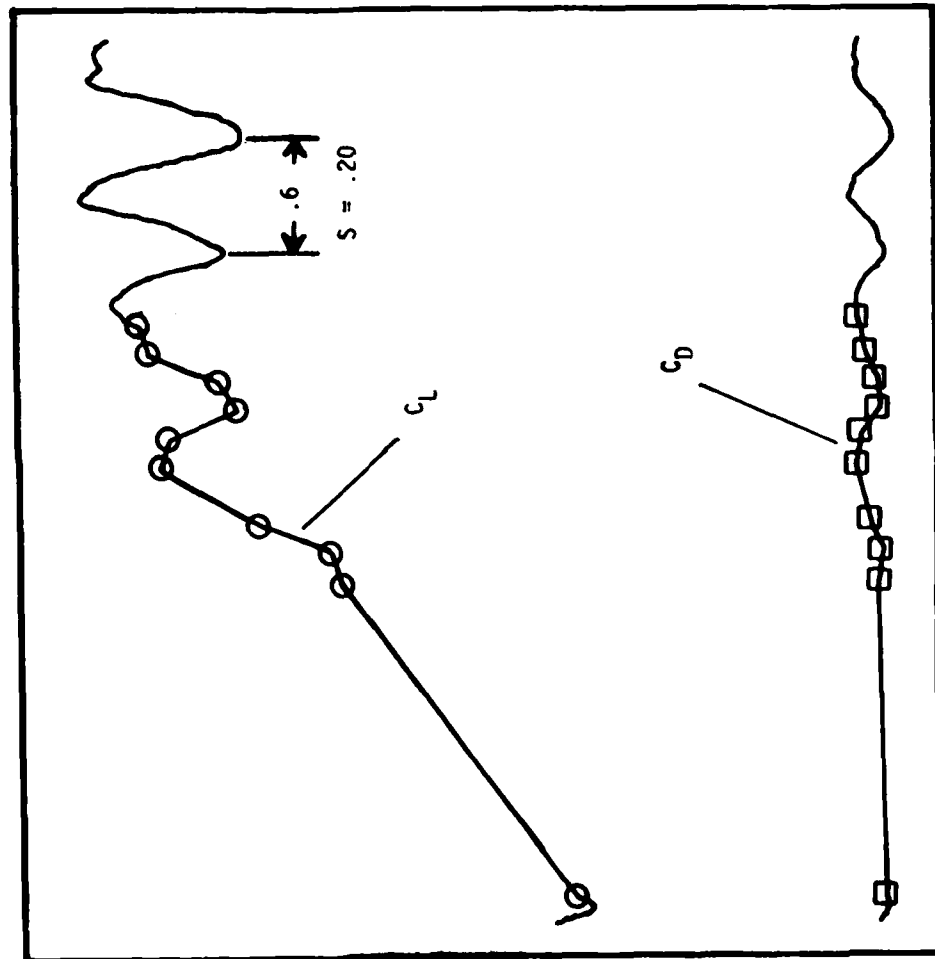### REYNOLDS NUMBER 167000

NACA 6412

$\phi = 0°$

Re = 167000

C_L

C_D

$S = .20$

.6

Figure 5.   Time Dependent Variation of
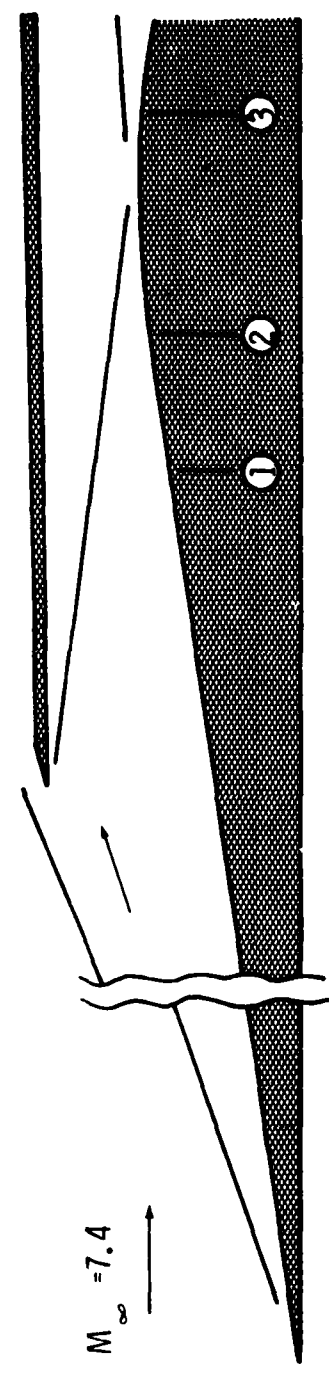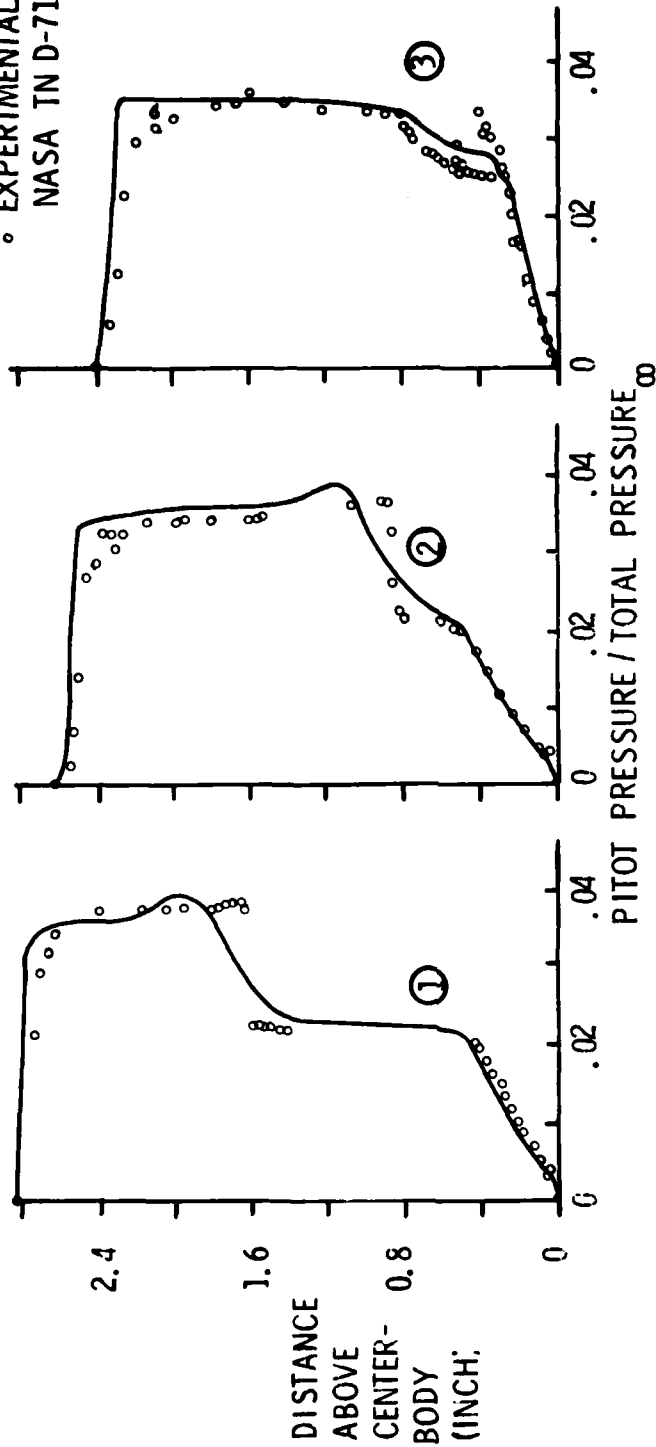Force Coefficients for an
Airfoil at Angle of Attack

17

Figure 6. Pitot Pressure Distributions
for a Hypersonic Inlet

18

COMPARISON OF HOLOGRAPHIC INTERFEROGRAM
COMPUTED DENSITY CONTOURS
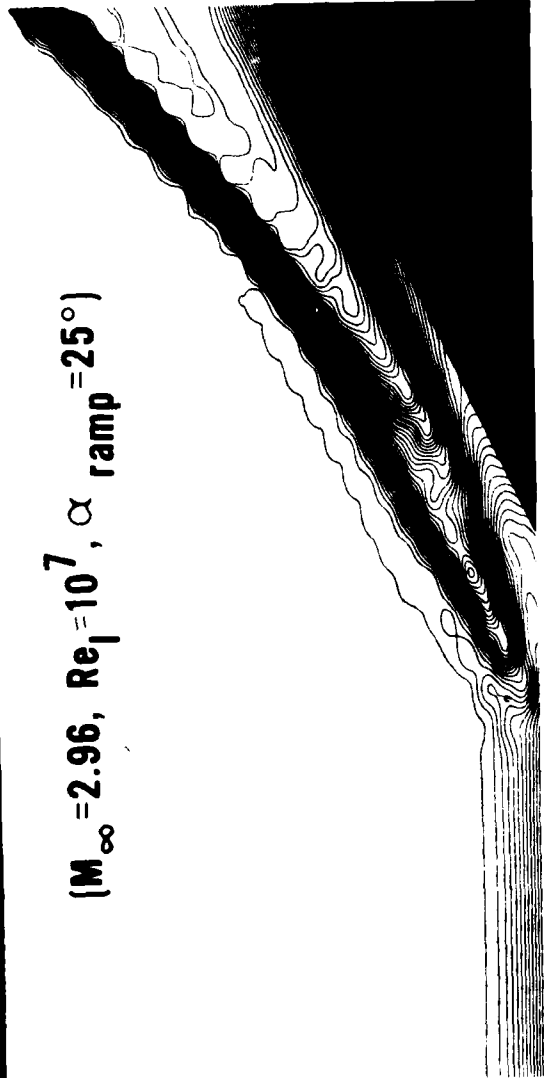
$(M_\infty = 2.96, \ Re_l = 10^7, \ \alpha_{ramp} = 25°)$

Figure 7. Comparison of Experimental
Interferogram for Compression
Ramp at M = 3

19

SHOCK - TURBULENT BOUNDARY LAYER INTERACTION WITH FLOW SEPARATION

Free Stream Mach Number = 3.0

Reynolds Number = 1.2 X 10^7

Incident Shock Angle = 27.6^0

COMPUTED RESULT (DENSITY CONTOUR)

EXPERIMENTAL INTERFEROGRAM

Figure 8.   Shock-Turbulent Boundary Layer
Interaction for Shock Impingement

20

DEFINITIONS

CONICAL SYMMETRY	SUPERSONIC INVISCID FLOW WITH CONICAL BOUNDARY CONDITIONS	$\dfrac{\partial}{\partial r} = 0$
"LOCAL CONICAL" SYMMETRY	$M_r - M\dfrac{d\delta^*}{dx} \sim \bar{x}$ $\bar{x} < 1$ CONICAL BOUNDARY CONDITIONS	$\dfrac{\partial}{\partial r} \ll \dfrac{\partial}{r\partial\phi}$ $\dfrac{\partial}{\partial r} \ll \dfrac{\partial}{r\partial\theta}$

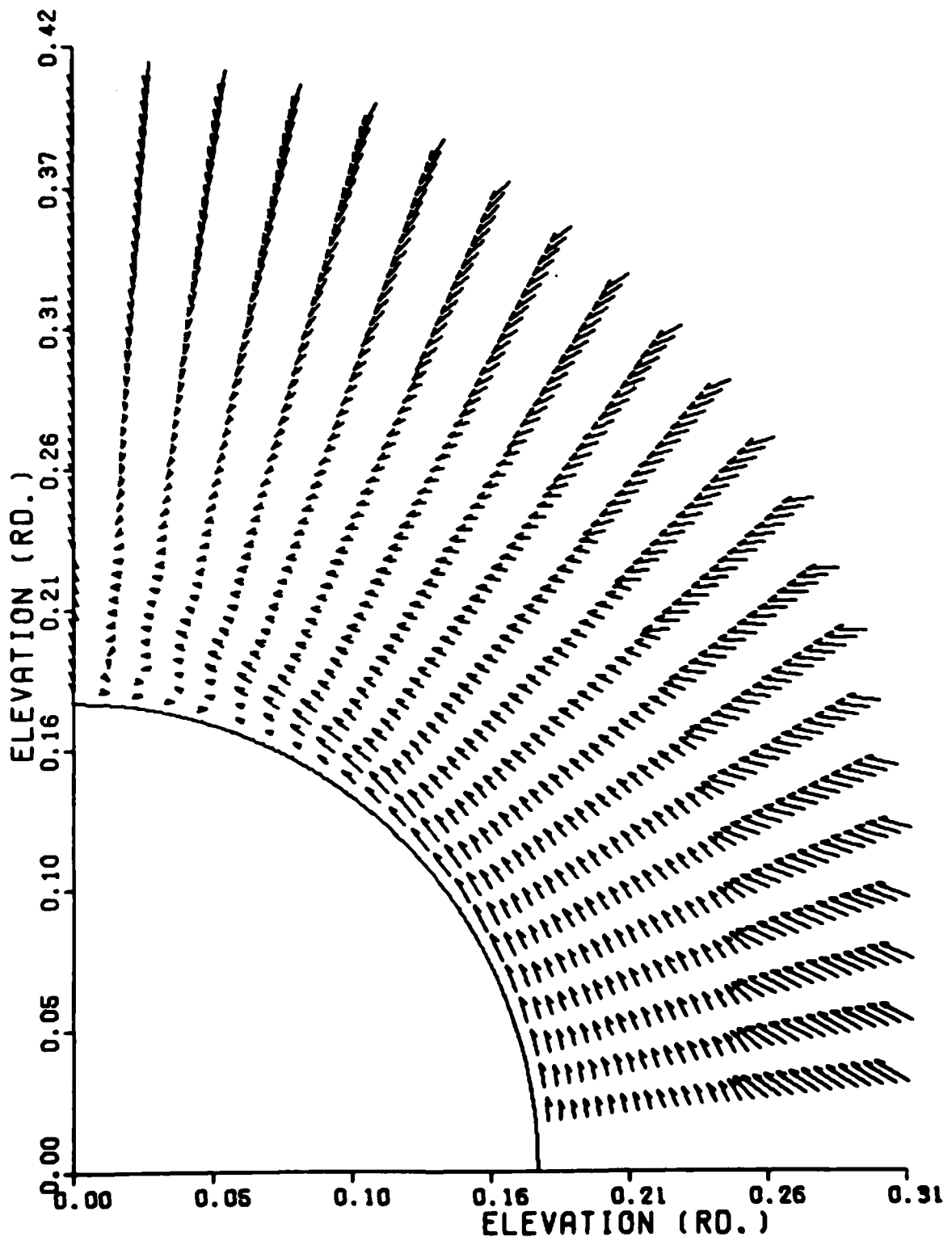Figure 9. Definition of Conical Symmetry

21

Figure 10. Velocity Vector Flow Field
for Cone at Angle of Attack

22
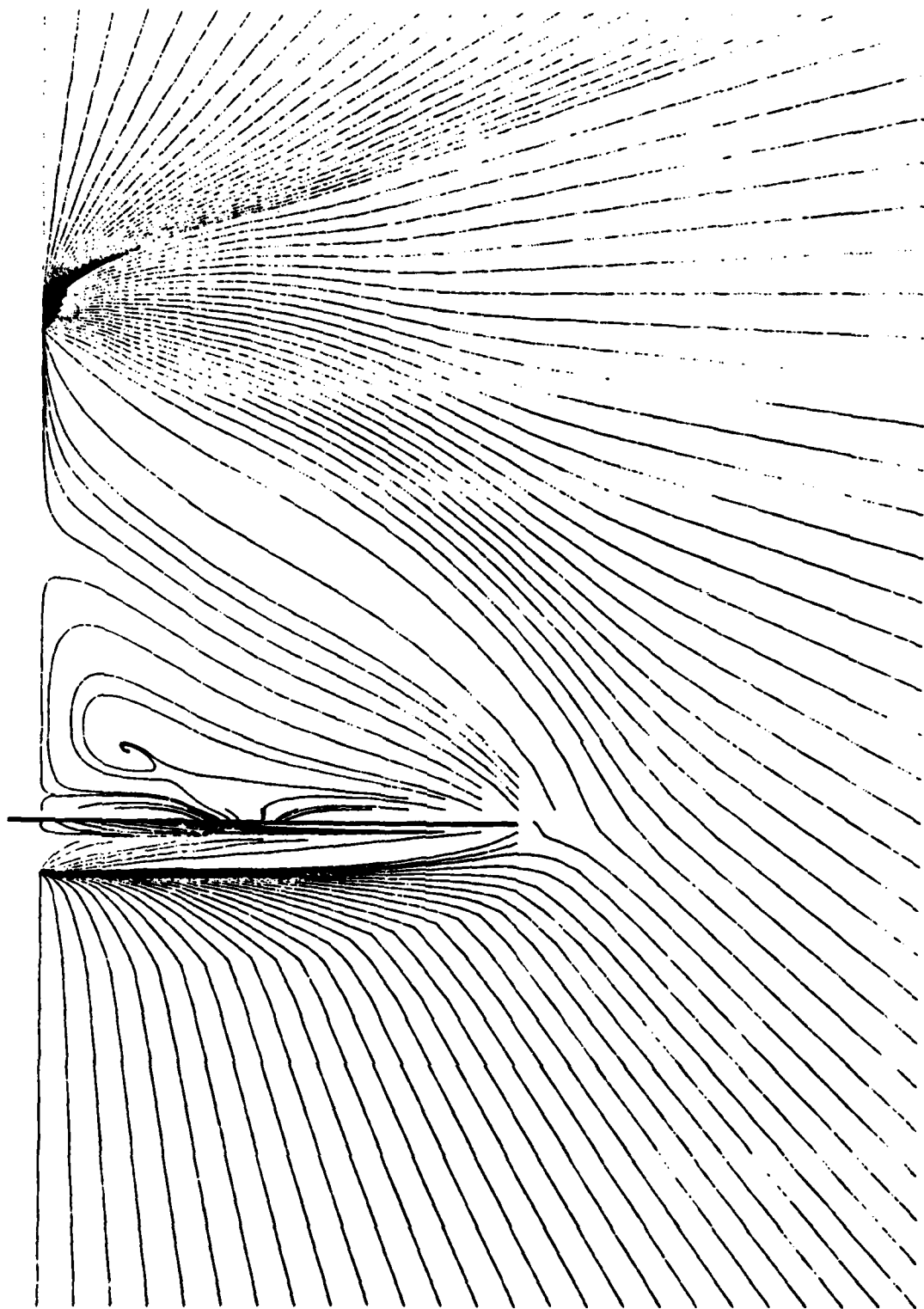
Figure 11. Cross Flow Streamlines for
            Hypersonic Flow Over Delta
            Wing α 15°

23

DENSITY CONTOUR FOR THE ENTIRE FLOW FIELD

$M\infty = 3.0$

$Re_L = 1.10 \times 10^6$

WEDGE ANGLE $9.48°$
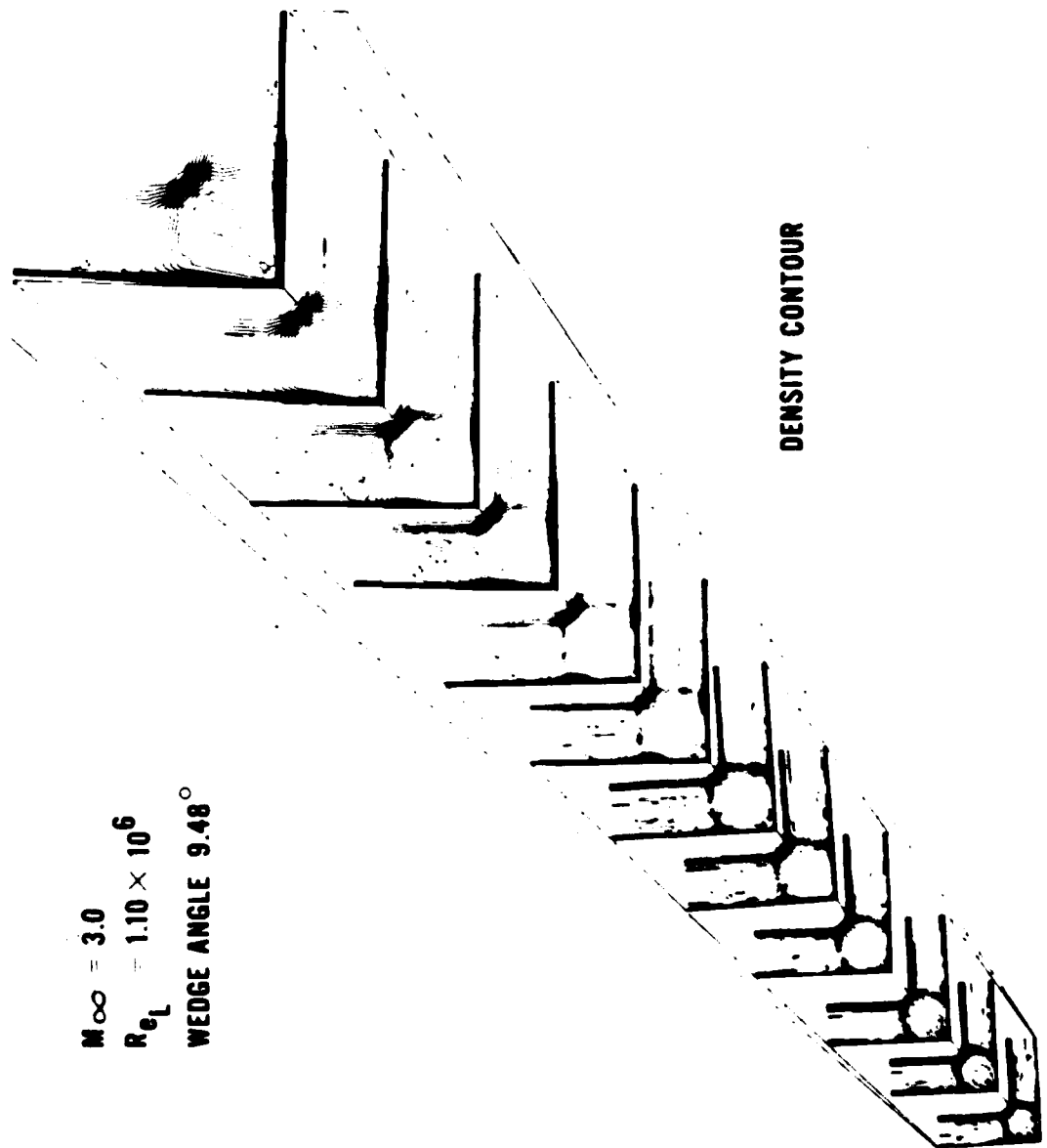
DENSITY CONTOUR

Figure 12.   Density Contours for Axial-
Corner Flow

24

COMPUTER	VERSION	UNIT PROCESSING RATE
CDC 6600	SCALAR	1.0
CDC 7600	SCALAR	5.2
CRAY 1	SCALAR	15.7
CRAY 1	VECTOR	127.7
CRAY 1	VECTOR (ASSEMBLY)	146.2
STAR 100	VECTOR (64 BIT)	49.9
STAR 100	VECTOR (32 BIT)	124.7

Figure 13.   Comparative Timing Results
for 3D Navier-Stokes Program

25

$M_\infty = 1.51$
$REY_\infty = 3.0 \times 10^7$
$\delta = 40\ mm$
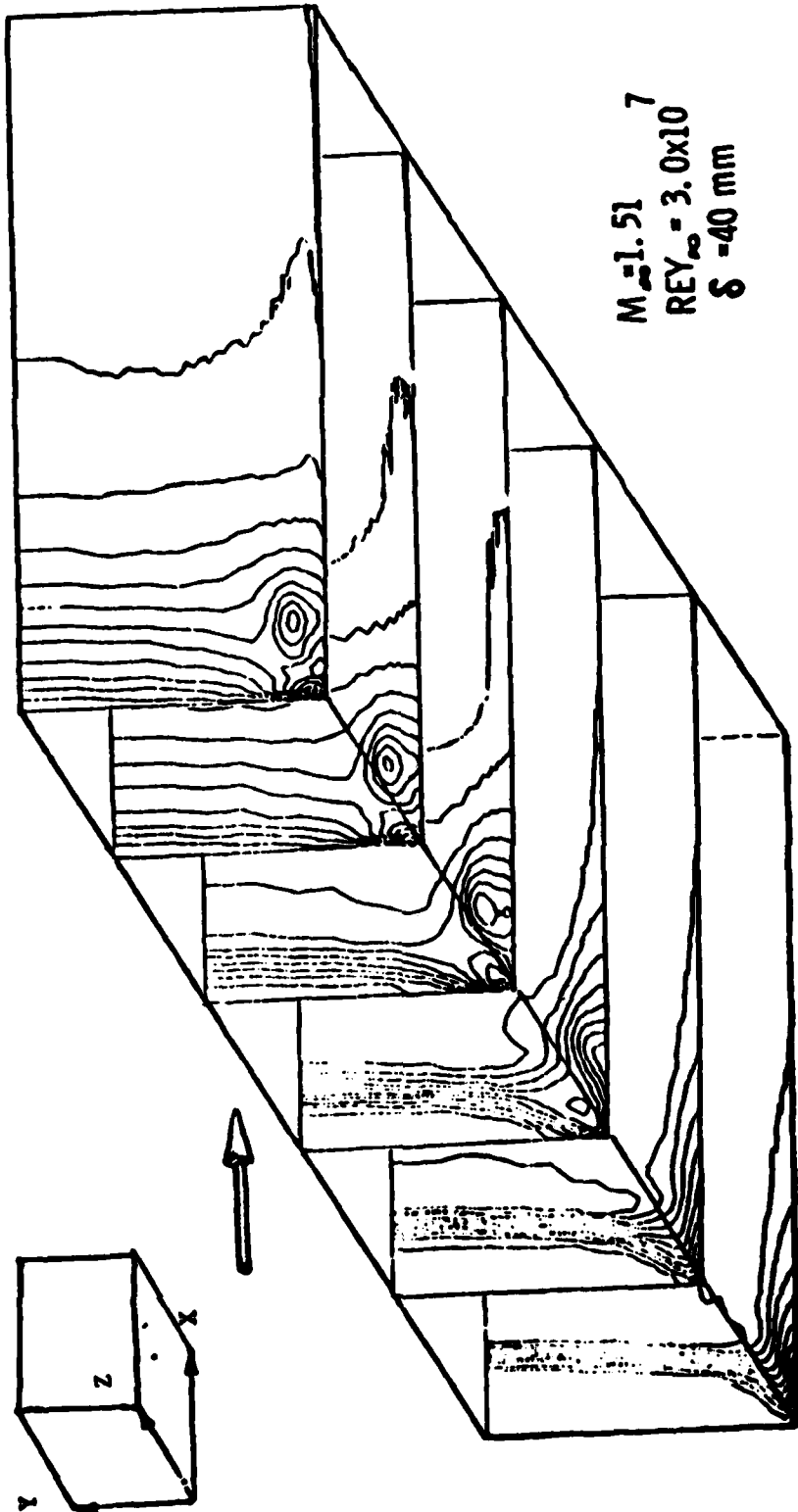
Figure 14. Perspective View of Density
Contours for a Normal Shock
in a Wind Tunnel Diffuser

NACA 6412 ANGLE OF ATTACK = 10 DEG
REYNOLDS NUMBER 167000
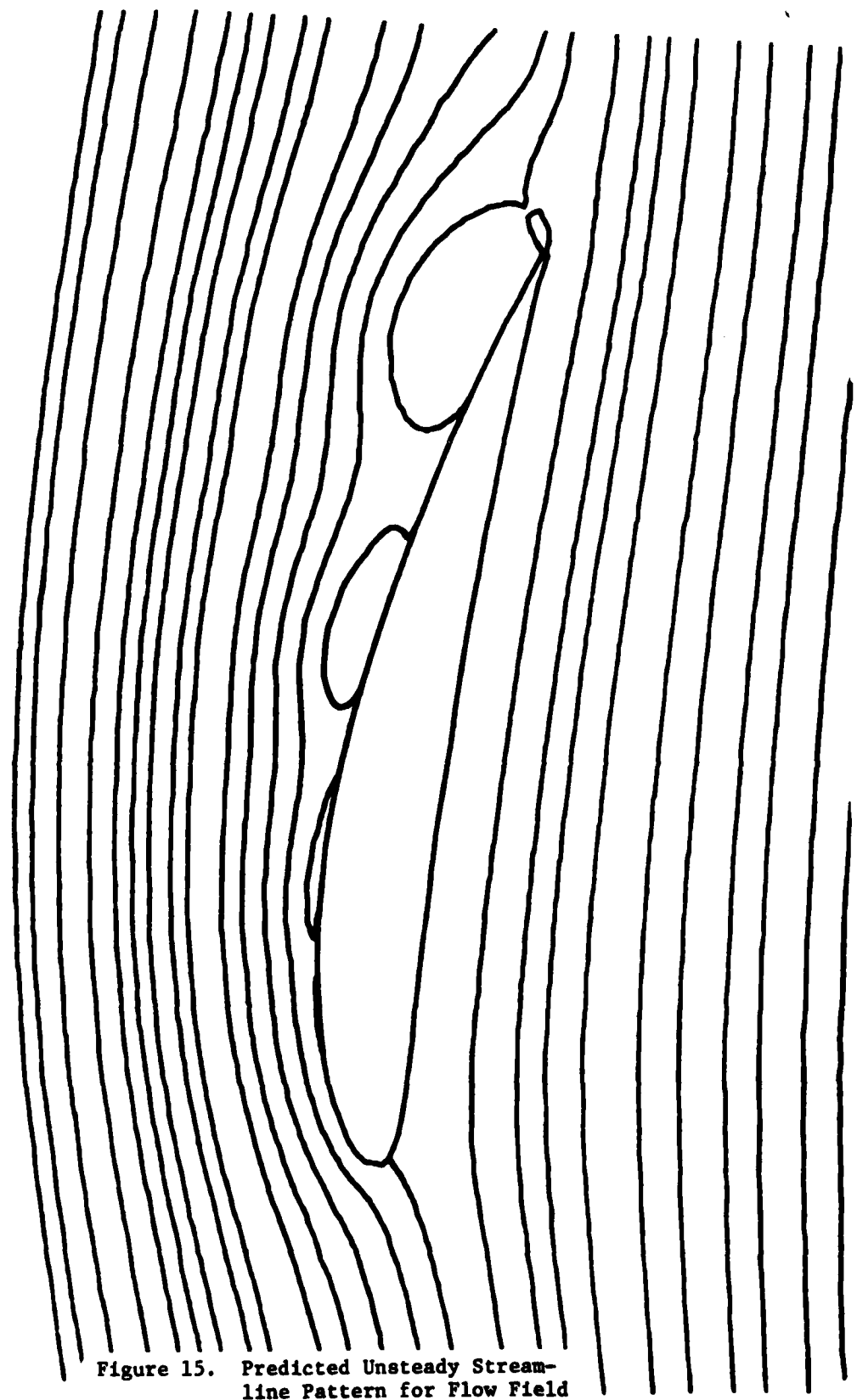
Figure 15.   Predicted Unsteady Stream-
line Pattern for Flow Field
Over an Airfoil

27
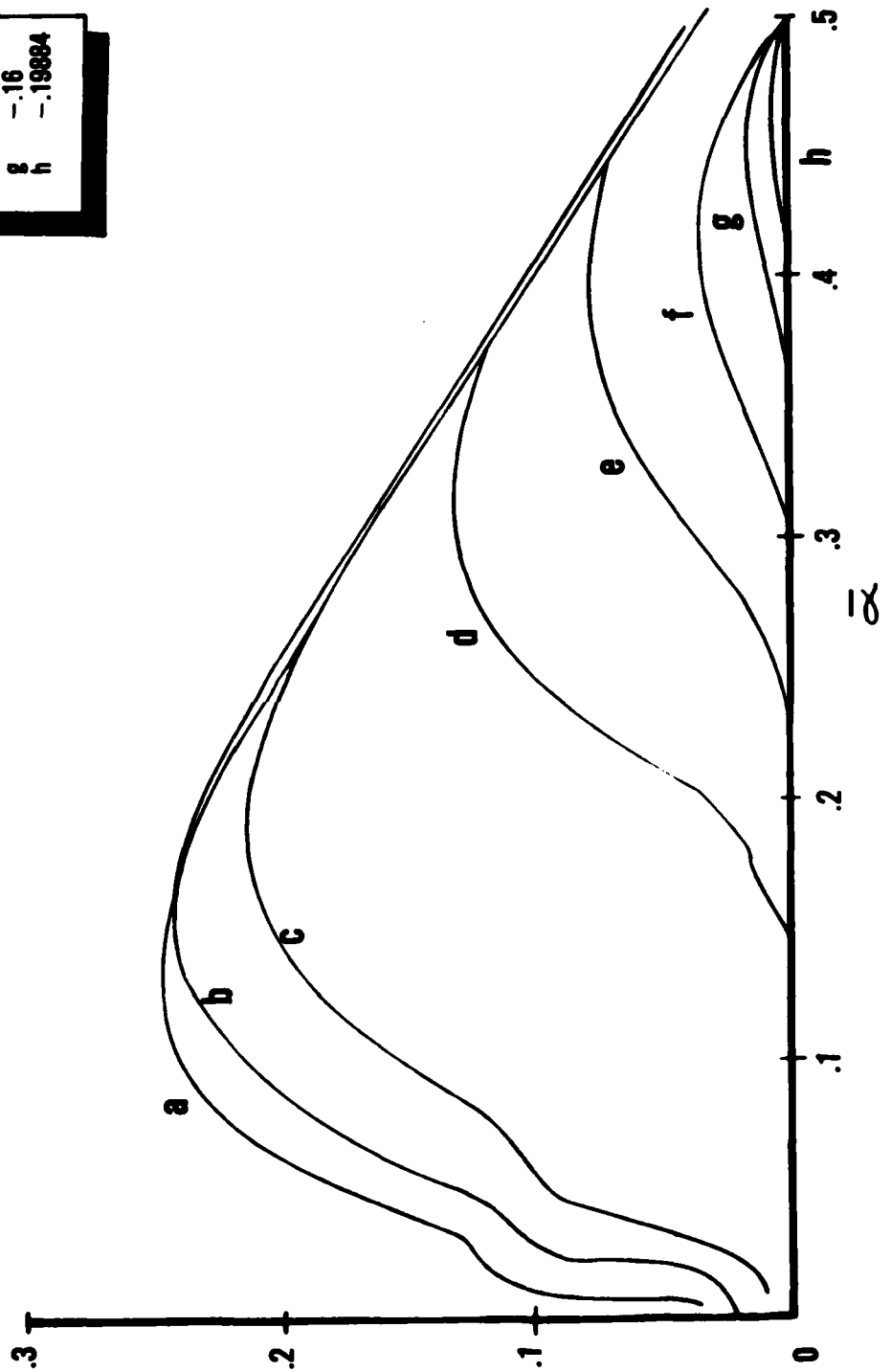
Figure 16. Unstable Wave Number Region
for Similar Separated Flows
(Lower Branch of Falkner-
Skan Boundary Layers)

28

$$e^{\frac{\alpha c_i L}{c}}$$

20

16

12

8

4

1

M = 0.85   δ = 0.15′
L = 3′   U∞ = 940 fps

1 = m
2
3
4
5
6
7
8

170

160

150

P(db)

140

130

120

110

0   100   200   300   400   500   600   700   800   900   IK   Hz

FREQUENCY

Figure 17. Comparison of Predicted and
Experimental Spectral Analysis

VII. REFERENCES

1. Ghia, U., Hodge, J.K., and Hankey, W. L., "Numerical Generation of Surface-Oriented Coordinates for Arbitrary Geometries - An Optimization Study," AFFDL-TR-77-117.

2. Shang, J. S., Hankey, W. L. and Dwoyer, D. L., "Numerical Analysis of Eddy Viscosity Models in Supersonic Boundary Layers," AIAA Journal, Vol. 11, Dec 1973, pp 1677-1683.

3. Shang, J. S. and Hankey, W. L., "Numerical Solutions for Supersonic Turbulent Flow Over a Compression Ramp, AIAA Journal, Vol. 13, No. 10, pp 1368-1374.

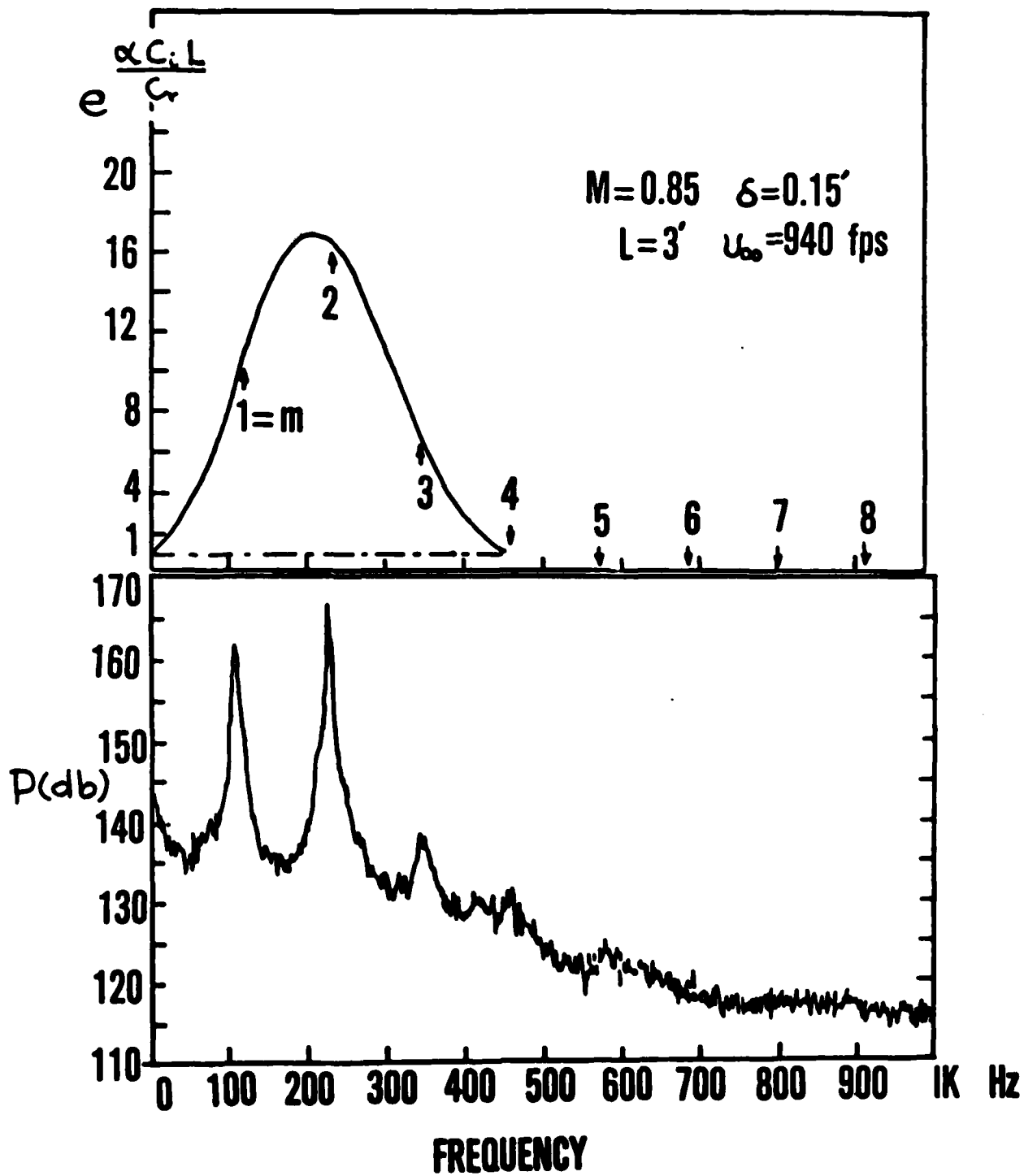4. Birch, S. F., "A Critical Reynolds Number Hypothesis and its Relation to Phenomenological Turbulene Models," Proceedings of the 1976 Heat Transfer and Fluid Mechanics Institute, Stanford University Press, 1976, pp 152-164.

5. Shang, J. S., "An Implicit-Explicit Method for Solving the Naiver-Stokes Equations," AIAA Journal, Vol. 16, No. 5, 1978 (Also AIAA paper 77-646).

6. Hodge, J. K., and Stone, A. L., "A Numerical Solution of the Navier-Stokes Equations in Body-Fitted Curvilinear Coordinates," AIAA Journal, Vol. 17, No. 5, May 1979.

7. MacCormack, R. W., "Numerical Solutions of the Interaction of a Shock Wave with a Laminar Boundary Layer," Lecture Notes in Physics, Vol. 8, Springer-Verlag, New York, 1971, pp 151-163.

8. MacCormack, R. W., "An Efficient Numerical Method for Solving the Time-Dependent Compressible Navier-Stokes Equations at High Reynolds Number," NASA TMX-73-129, July 1976.

9. Knight, D. D., "Numerical Simulation of Realistic High-Speed Inlets Using the Navier-Stokes Equations," AIAA Journal, Vol. 15, No. 11, Nov 1977.

10. Thompson, J. F., and Thames, F. C., "Numerical Solution of Potential Flow about Arbitrary Two-Dimensional Multiple Bodies," to be published in NASA CTR.

11. Shang, J. S., and Hankey, W. L., "Numerical Solution for Supersonic Turbulent Flow Over a Compression Ramp," AIAA Journal, Vol. 13, No. 10, Oct 1975, pp 1368-1374.

12. Shang, J. S., Hankey, W. L. and Law, H., "Numerical Simulation of Shock Wave-Turbulent Boundary Layer Interaction, AIAA Paper, 76-95, Jan 26-28, 1976.

13. J. S. Shang and Hankey, W. L., "Numerical Simulation of Shock Wave-Turbulent Boundary Layer Interaction," AIAA Journal, Vol. 14, No. 10, Oct 1976, pp 1451-1457.

14. Shea, J. R. III, "A Numerical Study of Transonic Normal Shock-Turbulent Boundary Layer Interactions," AIAA Paper, 78-1170, July 10-12, 1978.

15. Jobe, C. E. and Hankey, W. L., "Supersonic Turbulent Boundary Layer Calculations for Adverse Pressure Gradient Ramps," AIAA Abstract, January 1979.

16. Knight, D. D., "Turbulence-Model Predictions for a Flat Plate Boundary Layer," AIAA Journal, Vol. 13, No. 7, July 1975, pp 945-947.

17. McRae, D. S., "A Numerical Study of Supersonic Viscous Cone Flow at High Angle of Attack, AIAA Paper, 76-97, Jan 26-28, 1976.

18. McRae, D. S., "The Conically Symmetric Navier-Stokes Equations: Numerical Solution for Hypersonic Cone Flow at High Angle of Attack," AFFDL-TR-76-139, March 1977.

19. Bluford, G. S. Jr., "Navier-Stokes Solution of the Supersonic and Hypersonic Flow Field Around Planar Delta Wings," AIAA Paper, 78-1136, July 10-12, 1978.

20. Bluford, G. S. Jr., " A Numerical Solution of Supersonic and Hypersonic Viscous Flow Fields Around Thin Planar Delta Wings," AFFDL-TR-78-98.

21. Shang, J. S. and Hankey, W. L., "Numerical Solution of the Navier-Stokes Equations for a Three Dimensional Corner," AIAA Journal, Nov 1977, and AIAA Paper 77-169.

22. Shang, J. S. and Hankey, W. L., "Numerical Solution of Supersonic Interacting Turbulent Flow Along a Corner," AIAA Paper, 78-1210, July 10-12, 1978.

23. Shang, J. S., "Implicit-Explicit Method Solving the Navier-Stokes Equations, AIAA Journal, Vol. 16, No. 5, May 1978.

24. Shang, J. S. and Buning, P. G., "The Performance of a Vectorized Navier-Stokes Code on the CRAY-1 Computer," AIAA Paper, 79-1448, July 1979.

25. Ghia, K. N., Hankey, W. L., and J. K. Hodge, "Study of Incompressible Navier-Stokes in Primitive Variables Using Implicit Numerical Technique," AIAA Paper, July 1977.

26. Hodge, J. K. and Cooper, W. H., "Solution of Navier-Stokes Equations for Airfoils at Angle of Attack Near Stall," AFFDL-TM-77-94-FXM, Nov 1977.

27. Ghia, U., Hodge, J. K., and Hankey, W. L., "An Optimization Study for Generating Surface-Oriented Coordinates for Arbitrary Bodies in High-Re Flow," AFFDL-TR-77-117, Dec 1977.

28. Hodge, J. K. and Stone, A. L., "Numerical Solution for Airfoils Near Stall in Optimized Boundary-Fitted Curvilinear Coordinates," AIAA Paper, 78-284, Jan 16-18, 1978.

29. Knight, D. D. and Hankey, W. L., "Numerical Simulation of Nonchemically Reacting Radial Supersonic Diffusion Laser," AIAA Paper, 76-60, Jan 26-28, 1976.

30. Knight, D. D., "Numerical Simulation of Realistic High-Speed Inlets Using the Navier-Stokes Equations," AIAA Journal, Vol. 15, No. 11, Nov 1977.

31. Mikhail, A. G., Hankey, W. L., and Shang, J. S., "Computation of a Supersonic Flow Field Past an Axisymmetric Nozzle Boattail with Jet Exhaust," AIAA Paper, 78-993, July 25-27, 1978.

32. Mikhail, A. G., "Computation of a Supersonic Flow Field Past an Axisymmetric Nozzle Boattail with Jet Exhaust," AFFDL-TR-79-307&, Jan. 1979.

33. Waskiewicz, J. D. and Shang, J. S., "Numerical Simulation of Near Wakes Utilizing Relaxation Turbulence Model," AIAA Paper 79-0148, Jan 1979.

34. Roscoe, D. F. and Hankey, W. L., "The Stability of a Compressible Free Shear Layer," AIAA Paper, Feb 1979.

35. Verma, G. R., Hankey, W. L. and Scherr, S. J., "Stability Analysis of the Lower Branch Solutions of the Falkner-Skan Equations," AFFDL-TR, Aug 1979.

36. Hankey, W. L. and Shang, J. S., "The Numerical Solution of Pressure Oscillations in an Open Cavity," AIAA Paper, 79-0136, Jan 1979.