

UNCLASSIFIED

DTIC FILE COPY

2

SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1a. REPORT SECURITY CLASSIFICATION UNCLASSIFIED			1b. RESTRICTIVE MARKINGS	
2a. SECURITY CLASSIFICATION AUTHORITY MAR 12 1990			3. DISTRIBUTION / AVAILABILITY OF REPORT Approved for public release, distribution UNLIMITED	
2b.			5. MONITORING ORGANIZATION REPORT NUMBER(S) AFOSR-TR. 90-0296	
4. P AD-A219 621			7a. NAME OF MONITORING ORGANIZATION AFOSR/NA Bolling AFB DC 20332-6448	
6a. NAME OF PERFORMING ORGANIZATION S.G. RUBIN & P.K. KHOSLA DEPT. OF AERO ENG/ENG MECH.			7b. ADDRESS (City, State, and ZIP Code) AFOSR/NA Bolling AFB DC 20332-6448	
6c. ADDRESS (City, State, and ZIP Code) M.L. 343 UNIVERSITY OF CINCINNATI CINCINNATI, OH 45221-0343			9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER F49620-85-C-0027	
8a. NAME OF FUNDING SPONSORING ORGANIZATION Air Force Off. of Scientific Res.		8b. OFFICE SYMBOL (if applicable) NA		10. SOURCE OF FUNDING NUMBERS
8c. ADDRESS (City, State, and ZIP Code) Building 410 Bolling AFB, DC 20332		PROGRAM ELEMENT NO. 61102F	PROJECT NO. 2307	TASK NO. A1
11. TITLE (Include Security Classification) COMPOSITE REDUCED NAVIER STOKES PROCEDURES FOR FLOW PROBLEMS WITH STRONG PRESSURE INTERACTI				
12. PERSONAL AUTHOR(S) S.G. RUBIN AND P.K. KHOSLA				
13a. TYPE OF REPORT FINAL		13b. TIME COVERED FROM 2/1/85 TO 10/31/89		14. DATE OF REPORT (Year, Month, Day) 1/30/90
15. PAGE COUNT 39				
16. SUPPLEMENTARY NOTATION				
17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)	
FIELD	GROUP	SUB-GROUP	REDUCED NAVIER STOKES, THREE-DIMENSIONAL, SEPARATION LAMINAR FLOW BREAKDOWN, DIRECT SOLVER, MULTI GRID,	
	20.04			
19. ABSTRACT (Continue on reverse if necessary and identify by block number) The Reduced Navier Stokes (RNS) formulation for viscous-inviscid interacting flows with significant upstream or 'elliptic' effects has been applied for transient flow over airfoils at incidence, and steady two and three dimensional flows over cavity, wing and afterbody configurations. The solution technique applies uniformly over the entire Mach number range and allows for shock-boundary layer interaction, and for moderate regions of axial and secondary flow recirculation. For two dimensional problems with recirculation, it has been demonstrated that for laminar flows there exists a critical Reynolds number, that is geometry dependent, above which the solution exhibits a breakdown. This occurs in the region of recirculation and very close to the reattachment point. This phenomena is grid dependent and can be missed with insufficiently refined grids or when artificial viscosity is introduced. It has been shown that the pressure-split RNS procedure is in fact a special form of flux-vector splitting that has very favorable properties for sharp shock capturing. A sparse matrix direct solver procedure has been applied for both two				
20. DISTRIBUTION / AVAILABILITY OF ABSTRACT ☒ UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT. ☐ DTIC USERS			21. ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED	
22a. NAME OF RESPONSIBLE INDIVIDUAL L. SAKELL			22b. TELEPHONE (Include Area Code) (513) 356-7735	
			22c. OFFICE SYMBOL AFOSR/NA	

UNCLASSIFIED

dimensional transient flows, and for three dimensional steady flows with the RNS flux-split strategy. A uni-directional or semi-coarsening multigrid procedure has been further developed for viscous interacting flows, where significant grid stretching is required in order to adequately evaluate both thin viscous layers and large inviscid regions; with and without shock interaction.

Accession For	
NTIS GRA&I	☒
DTIC TAB	☐
Unannounced	☐
Justification	
By	
Distribution/	
Availability Codes	
Dist	Avail and/or Special
A-1	

98 05 12 12

FINAL REPORT

COMPOSITE REDUCED NAVIER-STOKES PROCEDURES

FOR FLOW PROBLEMS WITH STRONG PRESSURE INTERACTIONS

February 1, 1985 through January 31, 1990

Contract No. F49620-85-C-0027

Principal Investigators:

S.G. Rubin

P.K. Khosla

Department of Aerospace Engineering and Engineering Mechanics

University of Cincinnati

Cincinnati, Ohio 45221-0343

1. Introduction

The purpose of the current AFOSR research program is the development and application of primitive variable composite velocity and pressure flux-vector splitting formulations for more efficient numerical evaluation and prediction of viscous interacting flows than is generally possible with conventional time-dependent compressible Navier Stokes (NS) solvers. Three-dimensional separated flows, shock-shear layer interaction, high frequency laminar flow breakdown and transitional behavior and unsteady viscous/inviscid interactions are the primary topic areas.

A reduced form of the Navier Stokes equations, termed here RNS, is the foundation for both formulations. The RNS system is a composite of the full Euler and boundary layer/triple deck models. The flux-splitting or composite velocity procedures are designed to optimize the numerical representations of viscous and inviscid regions, respectively. These techniques can be viewed as a composite or single system 'matched interacting boundary layer-inviscid flow' solver or as a full elliptic version of parabolized Navier Stokes or PNS methodology. Both methods are applicable across the entire mach number range, i.e. from incompressible to hypersonic, and the same code has been applied at both ends of the spectrum. These formulations are applicable to flows with moderate regions of axial and secondary flow reversal, for capturing sharp shock waves and contact discontinuities, and for steady and transient behavior. The RNS system has previously been shown to accurately represent the full NS system for a large variety of flow problems and with a deferred corrector procedure full NS solutions can be recovered for this class of problems. The RNS procedure also allows for simplification of numerical boundary conditions and does not require the introduction of added artificial viscosity. This allows for fine mesh calculations that minimize numerical viscous effects and also allow for more efficient application of far field boundary conditions.

Several solution procedures for the discrete system of quasilinearized equations have been applied or are in development. For steady flow, space marching global relaxation techniques have been applied successfully for both the pressure variable flux-split and composite velocity formulations. For fully supersonic conditions or for very large free stream mach numbers, where subsonic viscous layers are very thin and where the effects of geometry do not have a significant influence on the axial flow behavior, a single pass will suffice to obtain the exact or a very good approximate solution. For subsonic or transonic flows, a multiple pass or full relaxation strategy is required.

For fine meshes, convergence rates can become quite slow; although, experience has shown that for most cases time step limitations of NS solvers are generally more severe. Convergence acceleration is a major element of this research program. To date, a unidirectional or semi-coarsening multi grid strategy, that is particularly effective when full multi grid methods fail, as with significant grid stretching, and a sparse matrix direct solver strategy, that is particularly effective for very strong interactions occurring in very local domains, where relaxation methods fail or stall, have been developed. These are currently being evaluated for three dimensional applications.

Current problem areas under investigation include three dimensional separated, subsonic and transonic flow over afterbody, corner wing-trough geometries, supersonic forebody, and shock interactive base flow, with cavity and roughness effects, and transient flow behavior associated with airfoils and supersonic inlets. The important aspects of adaptive grid generation and domain decomposition procedures, convergence acceleration and algorithm vectorization and parallelism complement the specific geometric computations.

The following topic areas have been considered during the past year. The references in brackets are given in section 2

Pressure Relaxation and Flux Vector Splitting [1,2,3,4,13]

Semi-coarsening Multi Grid/Laminar Flow Breakdown [1,5]

Transient Supersonic RNS Solutions for Inlet Unstart/Restart and Laminar Flow Breakdown [7,8,9,10,11]

Supersonic RNS Solutions/Sharp Shock Capturing/BL Interaction [7,8,9,10,11]

Sparse Matrix Solvers for Complex Viscous Interactions where Iterative Methods Fail [3,4,13,14]

Three Dimensional Separated Flow Pressure Variable Solutions [3,4,5,7,8,9,10,11]

Three Dimensional Separated Flow Composite Velocity Solutions [1,2]

During this reporting period, there have been numerous publications, presentations, dissertations and other interactions resulting from the research activity. These are also listed in Section 2. A review of progress associated with selected research investigations is presented in the following sections. A summary of research highlights, section 1.5, concludes this discussion. Supplementary lists of previous accomplishments, publications and student activity are given in the 1986, 1987, 1988 and 1989 Annual Reports.

1.1 Pressure Relaxation and Flux Vector Splitting

The present investigators have formulated composite pressure variable and composite velocity procedures for the computation of large Reynolds number (Re) viscous/inviscid interacting flows. These techniques have been applied as full potential and Euler solvers, but the application to viscous interacting flows has been the primary goal. This has led to development of global pressure relaxation procedures for a reduced form of the Navier Stokes equations termed RNS. This model is valid throughout the entire Mach number range and allows for upstream influence or 'ellipticity', whenever such effects are important. This could be viewed as a composite interacting boundary layer method, as a global PNS method, or more appropriately as a

form of flux vector splitting that more clearly defines the roles of convective and acoustic influences. These properties provide an important mechanism to efficiently and accurately investigate complex three-dimensional flow phenomena, including separation, shock interaction, and vortical interaction over the entire mach number range.

The governing equations have previously been described in detail, see references [1-12], and are written here for arbitrary non-orthogonal coordinates.

Pressure/Velocity Formulation

$$\frac{\partial}{\partial \xi} (\rho \sqrt{g} u) + \frac{\partial}{\partial \eta} (\rho \sqrt{g} v) + \frac{\partial}{\partial \zeta} (\rho \sqrt{g} w) = 0 \quad (1a)$$

ξ -Momentum Equation

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho \sqrt{g} u^2) + \frac{\partial}{\partial \eta} (\rho \sqrt{g} vu) + \frac{\partial}{\partial \zeta} (\rho \sqrt{g} wu) + \text{curvature terms} = \\ -g^{11} p_{\xi} - g^{12} p_{\eta} - g^{13} p_{\zeta} + \{\text{viscous terms}\} \end{aligned} \quad (1b)$$

ζ -Momentum Equation

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho \sqrt{g} wu) + \frac{\partial}{\partial \eta} (\rho \sqrt{g} vw) + \frac{\partial}{\partial \zeta} (\rho \sqrt{g} w^2) + \text{curvature terms} = \\ -g^{31} p_{\xi} - g^{32} p_{\eta} - g^{33} p_{\zeta} + \{\text{viscous terms}\} \end{aligned} \quad (1c)$$

Normal Momentum Equation

$$\begin{aligned} (\rho \sqrt{g} uv)_{\xi} + (\rho \sqrt{g} v^2)_{\eta} + (\rho \sqrt{g} vw)_{\zeta} + \text{curvature terms} = \\ -g^{21} p_{\xi} - g^{22} p_{\eta} - g^{23} p_{\zeta} \end{aligned} \quad (1d)$$

where

$$H = \frac{\gamma}{\gamma-1} \frac{p}{\rho} + \frac{1}{2} g^{ij} u_i u_j ; p = \rho RT$$

The equations (1) represent an asymptotic (RNS) approximation to the full Navier-Stokes systems and contain all of the terms that model the desired high Reynolds number flow physics. These equations differ from the thin-layer equations, wherein the normal viscous stresses are retained in the

three momentum equations. The RNS system includes all normal and secondary flow diffusion terms in the surface momentum equations, but neglects all diffusion in the surface normal momentum equation. These differences are quite significant as it has been shown by the present investigators that diffusion in the normal momentum equation is of the same order as the neglected axial diffusion effects; however, secondary flow diffusion is essential to accurately model three-dimensional separations.

Other investigators concerned with time dependent Euler, thin-layer or full Navier Stokes equations have introduced 'upwind' approximations that are associated with the movement of the physical forward and backward moving waves, i.e., flux-difference splitting, or with the movement of the discrete forward and backward moving particles, i.e., flux-vector splitting. In previous PNS investigations, the form of pressure/convective upwinding has resulted from characteristic or stability considerations. In these early PNS studies, the primary goal was to develop marching or initial value procedures for high Mach number flows. Elliptic effects were generally suppressed. For the present investigation, the Euler system has been re-examined to allow for fluxes in both the downstream and upstream directions. When the latter are neglected, the PNS methodology is recovered. When all terms are retained and this form of flux-vector splitting is applied to the RNS system, the global pressure relaxation procedure of the present investigators is recovered. These results represent a distinct variation of the flux-splitting methods presented by previous investigators.

This leads to the following discrete representation of the axial pressure gradient:

$$p_{\xi} = \omega_{i-1/2} \frac{(p_i - p_{i-1})}{\Delta \xi} + (1 - \omega_{i+1/2}) \frac{(p_{i+1} - p_i)}{\Delta \xi},$$

where $\omega = \text{Min}[\frac{\gamma M^2}{1+(\gamma-1)M_{\xi}^2}, 1]$ for constant stagnation enthalpy and $\omega = \text{min}[M^2, 1]$ for the full flux split energy equation. This technique has now

been extended to general non-orthogonal coordinates and the appropriate ω values are discussed in reference [10].

Global pressure relaxation for a pressure split form of the Euler or reduced Navier-Stokes (RNS) equations is then equivalent to a form of flux-vector splitting that satisfies the major eigenvalue and continuity constraints on the fluxes and flux derivatives. The upwinding is applied only in the axial or "streamline" direction. In keeping with the asymptotic form of the RNS system, two-point or trapezoidal discretization is used for appropriate normal gradients in order to allow for the accurate evaluation of shear layers and a consistent specification of far field boundary conditions. If the pressure gradient parameter $\omega(M)$ is given by its maximum value in subsonic regions, one of the eigenvalue is always zero and therefore with the second-order (ωp_x) discretization shock resolution is greatly improved. For regions of reversed flow convective upwinding is combined with the condition $\omega = 0$. This ensures that the fluxes, flux derivatives and eigenvalues remain continuous throughout the flow. This form of flux vector splitting is specifically designed to maintain a bias in the direction of the convective fluxes and therefore abandons the symmetry of the earlier forms of flux vector splitting. The upwinding leads to a relaxation method that is solely acoustic driven throughout subsonic regions, but also includes convective relaxation in regions of reversed flow without introducing large amounts of numerical dissipation. The procedure has now been applied to several problems that are discussed in detail in the papers referenced in section 2. Typical results for an afterbody configuration, supersonic over a cone-cylinder-flare configuration and supersonic flow over a ramp and flow in an inlet are shown in Section 3.

1.2 Direct Sparse Matrix Solvers for Fluid Flow Problems

Various iterative or factorized methods are commonly used for the solution of fluid flow problems. These techniques, although generally

efficient with respect to computer memory, tend to be sensitive to problem parameters and in many cases fail to converge or require gross under-relaxation or small time increments. Iterative methods generally follow a false transient to the steady state. This path can introduce erroneous behavior in certain problems. For example, in the computation of separated flows with large recirculation bubbles, high frequency phenomena near the reattachment point can be distorted due to false transients. Also, propagation of upstream influence and non-linear convergence can take place on different length and time scales, as determined by the iterative techniques and not by the physics. Gross under-relaxation or, equivalently, the use of small time steps, coupled with second or fourth order artificial dissipation, is quite often required to stabilize these calculation. Also, in certain cases physical high frequency modes will be suppressed. For flows with strong viscous/inviscid interaction, e.g., shock boundary-layer interaction, thin layers having triple-deck or similar scaling must be appropriately resolved in order to obtain a stable and accurate solution. This is either accomplished by using highly stretched grids or by employing a large number of grid points. In either case, many of the commonly employed iterative solution techniques become very slow or do not converge acceptably. On the other hand, if direct solution methods are employed, upstream influence is propagated instantaneously, even on finer or stretched grids. Of course, the propagation of non-linear information must be controlled in order to obtain the desired solution. The use of a direct solver improves many of the convergence difficulties; however, they can be quite slow and generally do require large amounts of computer memory. In view of the fact that increased memory is now available routinely on both large and desk top machines, the use of direct solvers becomes desirable and more feasible.

Recently many investigators, including the present authors, have examined the application of direct solvers for the computation of compressible

viscous and inviscid subsonic/transonic flows. It has generally been found that in two dimensions, direct solvers outperform iterative methods and in many 'difficult' cases compute solutions where the iterative methods generally stall or diverge. Some of this work has resulted in a number of publications and presentations. The direct solver has now been applied to the solution of three dimensional and two dimensional unsteady RNS equations. The general emphasis has been for the flow past afterbodies and for unstart and restart of supersonic inlets with a centerbody. For three dimensional flow, the direct solver has been employed as a cross plane solver with pressure relaxation in the axial direction. The resulting procedure is quite efficient and allows for the solution of problems on large grids. Additional strategies to render this procedure more efficient have also been investigated. In order to perform the LU decomposition in a minimum number of numerical operations, the YSMP (direct solver) performs symbolic factorization which is quite time consuming. This part of the solver has now almost been eliminated for steady three dimensional and two dimensional unsteady flow computations using the RNS formulation. This is achieved by noting that the matrix structure of the coefficient matrix at consecutive planes, or different time steps in an otherwise unseparated flow, remains unchanged. In these cases the symbolic factorization needs to be performed only once. The symbolic factorization is performed only when the matrix structure changes. This results in 40-50% savings in CPU time. In addition, the LU factorization must be updated only periodically. The LU factorization at one plane is saved and used to drive residuals at subsequent planes (typically 5) to zero. For unsteady flows on finer meshes, the storage requirements for the direct solver can be quite large. The two to four megaword memory usually allocated to each job on most supercomputers is exceeded for coarse meshes. In such cases, the computations are performed on finer meshes by using domain decomposition. The computational domain in the flow direction is divided into several

overlapping subdomains. Usually an overlap of 3 to 5 points is prescribed. This is found to be reasonable for implicit capturing of moving shocks. For flows without shocks, even a smaller overlap can be prescribed. On each subdomain the direct solver is employed. The resulting procedure becomes iterative; however, these iterations converge quite rapidly. Although convergence is dependent upon the size of Δt , 3-8 iterations are usually sufficient to converge the solution at a given time level. These iterations include the effort to converge all the non-linearities. The resulting procedure is found to be quite robust and captures moving shocks and shock boundary-layer interactions accurately. These modifications have been incorporated into the RNS codes and have now been applied for the computation of afterbody flows and for the unstart/restart of supersonic inlets.

1.3 Three Dimensional Subsonic/Transonic Composite Velocity Solutions

A composite velocity procedure for the three-dimensional reduced Navier-Stokes equations has been further developed. In the spirit of matched asymptotic expansions, the velocity components are written as a combination of multiplicative and additive composite of viscous like velocities (U, W) and pseudo-potential or inviscid velocities (Φ_x, Φ_y, Φ_z). The solution procedure is then consistent with both asymptotic inviscid flow and boundary layer theory. For transonic flows, the Enquist-Osher flux biasing scheme developed for the full potential equation is used. A quasi-conservation form of the governing equations is used in shock regions to capture the correct rotational shock; the standard non-conservation form of the equations used in non-shock regions. The consistent coupled strongly implicit procedure combined with a plane relaxation procedure is used to solve the discretized equations.

A general, non-orthogonal, curvilinear system of coordinates is presented. This allows for the use of appropriate 'streamline' coordinate

systems and places no restrictions on the type of grid. In the spirit of matched asymptotic expansions, the contravariant velocities are rewritten as

$$u = (U+1)(g^{11}\phi_{\xi} + g^{12}\phi_{\eta} + g^{13}\phi_{\zeta}) - (U+1)u_e \quad (2a)$$

$$v = (g^{21}\phi_{\xi} + g^{22}\phi_{\eta} + g^{23}\phi_{\zeta}) \quad (2b)$$

$$w = (U+1)(g^{31}\phi_{\xi} + g^{32}\phi_{\eta} + g^{33}\phi_{\zeta}) + W = (U+1)w_e + W \quad (2c)$$

The composite representations of u and w , the axial and crossflow velocity components, contain two types of terms, e.g., an irrotational 'pseudo' potential function and viscous velocities U and W . Since the change in v across the boundary layer is small, the normal velocity is determined solely by the 'pseudo' potential function. This particular composite form has the advantage that no additional unknowns are introduced and the boundary conditions remain easy to implement.

If definitions 2(a-c) are used in the RNS equations 1(a-d), the composite system is recovered, see references [2,13,16]. The governing composite velocity equations are differenced so that second order accuracy is obtained for all terms. First order accurate differencing for the streamwise derivative U_{ξ} and W_{ξ} is left as an option. The difference form of the continuity, ξ -momentum, and ζ -momentum equations is centered at (i,j,k) . All terms in the ξ -momentum, ζ -momentum, and continuity equations are differenced using central differences at half points. The derivatives of ϕ may then be central differenced at the half point locations. This provides the usual three point central difference for terms like $\phi_{\xi\xi}$, $\phi_{\eta\eta}$, and $\phi_{\zeta\zeta}$. In the η - ζ cross plane, the values of U and W at the half-points are determined by the average of the values at neighboring grid points. In order to provide the proper upwind bias consistent with the boundary layer marching character of U and W , the values of U and W at $i+1/2$ and $i-1/2$ are represented with upwind approximations as follows:

$$U_{i+1/2} = U_i + \frac{1}{2} \epsilon \sigma_i (U_i - U_{i-1})$$

$$U_{i-1/2} = U_{i-1} + \frac{1}{2} \epsilon \sigma_{i-1} (U_{i-1} - U_{i-2})$$

where $\epsilon = 0$ provides a first order accurate representation and $\epsilon = 1$ provides a second order accurate representation.

In separated flows, the representation of $W_{i+1/2}$ and $W_{i-1/2}$ must be modified to provide the proper upwinding for the W_ξ derivative in the separated flow region. This is required since W appears as an additive term in the composite representation for w rather than as a term multiplying an inviscid velocity as does U . Therefore the only upstream influence in separated regions on this term comes through upwinding of the W_ξ derivative.

The representations for $W_{i+1/2}$ and $W_{i-1/2}$ become:

$$\begin{aligned} W_{i+1/2} &= SM*[W_i + \frac{1}{2} \epsilon \sigma_i (W_i - W_{i-1})] + \\ &\quad SP*[W_{i+1} + \frac{\epsilon}{2\sigma_{i+1}} (W_{i+1} - W_{i+2})] \\ W_{i-1/2} &= SM*[W_{i-1} + \frac{1}{2} \epsilon \sigma_{i-1} (W_{i-1} - W_{i-2})] + \\ &\quad SP*[W_i + \frac{\epsilon}{2\sigma_i} (W_i - W_{i+1})] \end{aligned}$$

where

$$SM = \frac{1}{4} [2 + \text{SGN}(U_{i-1} + 1) + \text{SGN}(U_i + 1)]$$

$$SP = \frac{1}{4} [2 - \text{SGN}(U_{i-1} + 1) - \text{SGN}(U_i + 1)].$$

A backward difference for W_ξ results if $SM = 1$ and $SP = 0$ and a forward difference results when $SM = 0$ and $SP = 1$. At separation and reattachment points $SM = 1/2$ and $SP = 1/2$. This leads to a central difference for W_ξ . This provides for a smooth transition between backward and forward differencing at separation and reattachment points, which enhances the stability of the solution procedure in these regions.

The final form of the difference equations is solved using a standard plane relaxation procedure. This global relaxation procedure solves in the η - ξ cross plane assuming values from the previous global iteration for points ahead of the cross plane and using known values from the current global iteration for points from the previous marching planes. For the composite velocity equations, relaxation is only required for Φ since U and W may be marched for unseparated flows. In separated flows, additional relaxation on W is introduced due to the required upwinding of the W_{ξ} term.

In order to solve the difference equations in the cross plane in an efficient and robust manner a solution algorithm that is non-iterative, unconditionally stable, and spatially consistent is required. A solution algorithm that satisfies the above requirements is the consistent coupled strongly implicit procedure (CCSIP) developed by Khosla and Rubin.

1.4 Results

Transient Flows in an Inlet

In order to investigate the ability of the RNS formulation to capture sharp moving shocks and the associated shock boundary-layer interaction, flow through a supersonic inlet has been considered. The inlet has been modelled as a two-dimensional channel. The internal and coupled external flow fields have been computed for different back pressures. The solution has been obtained using a domain decomposition strategy, where each domain is solved by the sparse matrix direct solver. Both inviscid and viscous flows have been considered. Typical results are depicted in figures 1(a-d) and special flow features are discussed here. In the unstart mode, i.e. for a sufficiently large back pressure, a curved shock stands ahead of the inlet and the mass spillage occurs around the leading edge of the inlet. For viscous flows with reduced back pressure, a y-shaped or a lambda shock is formed near the leading edge of the inlet. At higher stagnation pressures, fluid tends to accumulate within the boundary layer and downstream of the

normal shock. This flow behavior is in agreement with flow visualization studies by Herman [] for flow in a shock tube. It should be emphasized that in the present case, the flow inside and outside of the inlet are coupled and computed simultaneously. The computations for an axisymmetric inlet with a centerbody are in progress. Some results have been presented in reference [10,11].

Supersonic flow past a cone-cylinder-flare

Axisymmetric laminar flow past the cone-cylinder-flare configuration at $M_\infty = 3$ have been computed. The results are shown in figures 2(a-c). In these calculations, the outer bow shock is fitted and imbedded recompression shocks are captured. It can be seen that these flows, with strong viscous/inviscid interaction, are captured quite well. It may be noticed that the global procedure converges much faster at higher mach numbers. Very recently additional results for flows over a cylindrical compression ramp at $M_\infty = 7$ and a variety of Reynolds numbers have also been obtained. These results are depicted in figures 3(a-e). In these computations, all shocks have been captured. Flow reversal due to shock-boundary-layer interaction in the compression corner is quite evident.

Three-Dimensional Flow:

Three dimensional computations for $M_\infty = .6$ laminar flow past past afterbody configurations of elliptical and hyper-elliptic cross-sections have also been considered. The results are shown in figures 4(a-d) and 5(a-d). Preliminary results indicate the existence of a vortex breakdown along the major axis of the ellipse. This will be further investigated with grid refinement studies in both the azimuthal and flow direction.

The results of transonic flow calculations on an afterbody configuration of elliptic cross-section, using composite velocity formulation are depicted in figures 6(a-e). These computations are for turbulent flow conditions and use to Baldwin Lomax eddy viscosity model.

Interactions

During the period of this annual report there has been interactions with several outside researchers; in addition, one Ph.D graduate has accepted employment with GE Aircraft Engines CFD branch. Dr. Bender, at General Dynamics, is continuing his investigation on the application of direct solvers RNS procedures for chemically reacting PNS solvers. There were many fruitful discussions about working on a joint project. Dr. Gordnier is still carrying out additional studies for composite velocity applications to afterbody flows. The PI's also had extensive discussions with Drs. L. Schutzenhofer, Helen Miagney and K. Tucker regarding code validations for applications to flows related to space shuttle program.

RNS procedures for internal flows are also under investigation at the NASA Lewis Research Center by T. Bensen, J. Adamczyk, and by D.R. Reddy at Sverdrup, a Lewis contractor. A number of meetings have taken place with these researchers. Several papers on supersonic inlet unstart/restart related to their interest have either appeared or are in the completion stage. Applications to turbomachinery are also in the completion stage. Applications to turbomachinery are also being considered.

Several researches from Lewis, WPAFB and AFIT have shown interest in working on RNS techniques towards their Ph.D theses. Mark Celestina of Sverdrup and Philip Morgan from WPAFB have already enrolled in our Ph.D program.

Roger Cohen who spent two years with us as a fullbright fellow, is currently finishing his Ph.D under the supervision of Prof. Fletcher at University of Sydney in Australia. Recently, he presented two papers on RNS direct solver solutions at the international conference in Brisbane and Tokyo. Roger is also investigating other re-ordering techniques and the application of Conjugate gradient type methods to the RNS direct solver approach.

H.C. Raven and M. Hockstoa of Maritime Research Institute in Netherlands are also continuing the successful application of the RNS techniques to the solutions for ship stern flow computations. For high mach number supersonic and hypersonic flows, the RNS formulation is still being applied by M. Barnett of UTRC and is also being considered by D.R. Reddy of Sverdrup.

S. Rubin participated in a workshop on modified Navier-Stokes procedures for efficient evaluation of thick interacting boundary-layer flows held at the Royal Institute in Stockholm, Sweden, May 1989. The participation was by invitation only.

Highlights of Research Progress

- (i) The reduced Navier-Stokes (RNS) formulation for the solution of viscous-inviscid interacting flows with significant upstream influence has been applied for the transient flows in a supersonic inlet with a centerbody, three dimensional afterbody flows, hypersonic axisymmetric flow over cone-cylinder flare configuration and a two dimensional ramp. The solution technique applies uniformly over the entire mach number range and allows for shock boundary-layer interaction, and for moderate region of axial and secondary flow recirculation. Furthermore, the formulation does not require the addition of explicit artificial viscosity.
- (ii) In order to apply the procedure on fine meshes, a domain decomposition strategy is developed. This strategy when combined with a direct solver on each subdomain leads to an efficient and robust solution algorithm. Flows with moving shocks, recirculation regions lambda shocks are computed by this technique.
- (iii) The pressure-split flux vector procedure is now extended to arbitrary non-orthogonal coordinates.

2. AFOSR Publications, Presentations, Related Activity and Interaction
1/89-1/90

A. Publications and Proceedings

1. Gordnier, R.E. and Rubin, S.G.: "Transonic Flow Solutions Using a Composite Velocity Procedure for the RNS Equations", Computers and Fluids, 17, 1, pp. 85-98, Feb. 1989
2. Gordnier, R.E. and Rubin, S.G.: "Three Dimensional Composite Velocity Solutions for Subsonic/Transonic Flows", Proceedings of 4th symposium on Numerical and Physical Aspects of Aerodynamic Flows, Long Beach, Ca., Springer-Verlag Pub., Dec. 1989
3. Khosla, P.K., Rubin, S.G. and Himansu, A.: "Three Dimensional RNS Separated Flow Calculations", AIAA 29th Aerospace Sciences Meeting, Reno, Ne., AIAA Paper No. 89-0300, Jan. 1989
4. Cohen, R. and Khosla, P.K.: "Three Dimensional RNS Solutions for Subsonic Separated and Non Separated Flows", Int'l J. Numerical Methods in Fluids, Vol. 9, pp. 1087-1098, 1989
5. Rubin, S. and Himansu, A.: "Convergence Properties of High Reynolds Number Separated Flow Calculations, Int'l J. Num. Methods in Fluids, 9, pp. 1395-1411, 1989.

B. Presentations, Seminars and Other Student Activity

6. Rosenbaum, D. and Rubin S.G." Global Pressure Relaxation for Laminar 2-D Internal Flow, Accepted for Int. J. for Numerical Methods in Fluids.
7. Pordal, H.S., Khosla, P.K. and Rubin, S.G., "Inviscid Steady/Unsteady Flow Calculations," Presented at CATC Meeting in Brisbane, Australia, July 1989. Also to appear the Proceedings to be published by Academic Press.
8. Pordal, H.S., Khosla, P.K. and Rubin, S.G., "Inviscid Steady/Unsteady Flow Calculations," submitted to Computers & Fluids.
9. Pordal, H.S., Khosla, P.K. and Rubin, S.G., "Unsteady Flow in Planar Inlets," submitted to Int. Conf. on Unsteady Flows.
10. Pordal, H.S., Khosla, P.K. and Rubin, S.G., "Unsteady Viscous and Inviscid Supersonic Flow Calculations in Axisymmetric Inlets," AIAA Conference, Reno, AIAA-0585, 1990.
11. Pordal, H.S., Khosla, P.K. and Rubin, S.G., "Supersonic Turbulent Flows in Inlets," accepted for CFD Symposium on Aeropropulsion at NASA Lewis.
12. Hussain, Almahroos, Khosla, P.K. and Rubin, S.G., "Subsonic/Transonic Flow Calculations Over 3-D Afterbodies," accepted for CFD Symposium on Aeropropulsion at NASA Lewis.
13. Gordnier, R.E. and Rubin, S.G.: Three Dimensional Composite Velocity Solutions for Subsonic/Transonic Flow Over Afterbodies, AIAA-89-1837, 1989.
14. Discussions with NASA Marshall CFD Group, March 1989.

15. Rubin, S.G., Presentation of Review of our Work on Workshop on Thick Boundary-Layers at Royal Institute in Stockholm, Sweden, May, 1989.
16. Gordnier, R.E.: "Composite RNS Solutions for Afterbody Configurations", Ph.D. dissertation, Jan. 1989, now at AFWAL, WPAFB, Dayton, OH.
17. Liang, T.: "RNS Procedures for Supersonic Viscous Interaction", Ph.D. dissertations, Jan. 1990, Now at G.E. Evendale, OH.
18. Hagenmaier, M.: G.E. Fellow and Summer Research Assistant working on adaptive methods for high speed RNS formulations, M.S. student

C. Committees and Assignments

S.G. Rubin (1988-89):

Member of the Advisory Committee for (Case Institute/NASA Lewis) Institute for Computational Methods in Propulsion (ICOMP)

Consultant on the NASA (OAST) Aerospace Research and Technology Subcommittee (ARTS) of the Aeronautics Advisory Committee (AAC)

Member of NASA AAC Review Committee on Supersonic Cruise Airplane Drag Reduction

Editor-in-Chief, Int'l Journal, Computers and Fluids, Pergamon Press.

P.K. Khosla (1988-89):

Member of Editorial Advisory Board, Int'l Journal, Computers and Fluids

D. Student Graduates (2/1/85 - 10/31/89)

- | | |
|---------------------------------|------------------------|
| 1. H.T. Lai, Ph.D 1985 | (Sverdrup, Cleveland) |
| 2. S.V. Ramakrishnan, Ph.D 1988 | (Rockwell) |
| 3. Eric Bender, Ph.D 1988 | (General Dynamics) |
| 4. Raymond Gordnier, Ph.D 1988 | (WRDC) |
| 5. D. Rosenbaum, Ph.D 1988 | (Garrett Air Research) |
| 6. T. Liang, Ph.D 1989 | (General Electric) |
| 7. H. Pordal, M.S. 1986 | (U.C. Ph.D Student) |
| 8. A. Himansu, M.S. 1986 | (U.C. Ph.D Student) |
| 9. M. Hagenmaier, M.S. 1990 | (U.C. Ph.D Student) |

10.plt ** contour plot

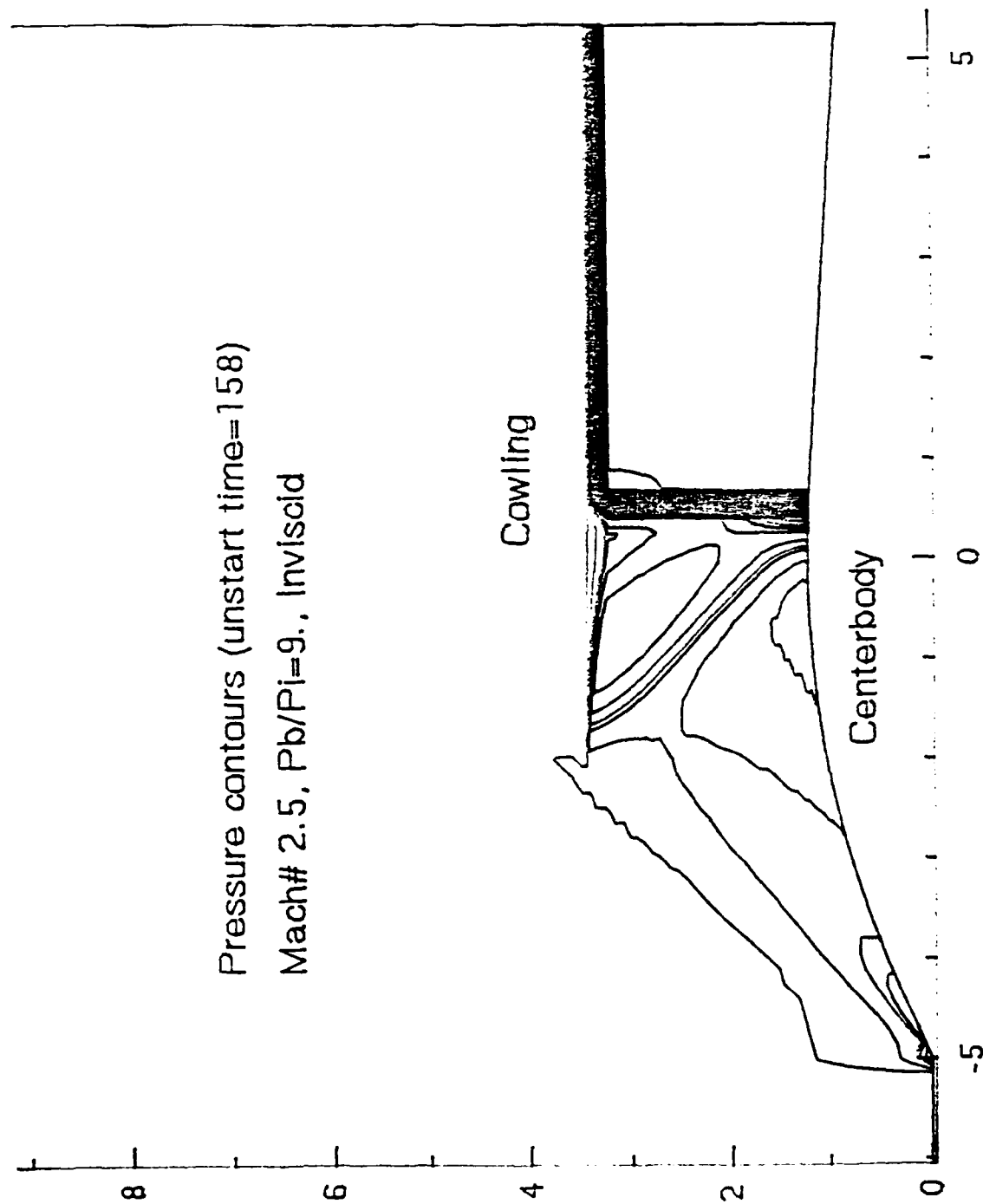


Fig. 1a. Inviscid pressure contours - shock inside the inlet

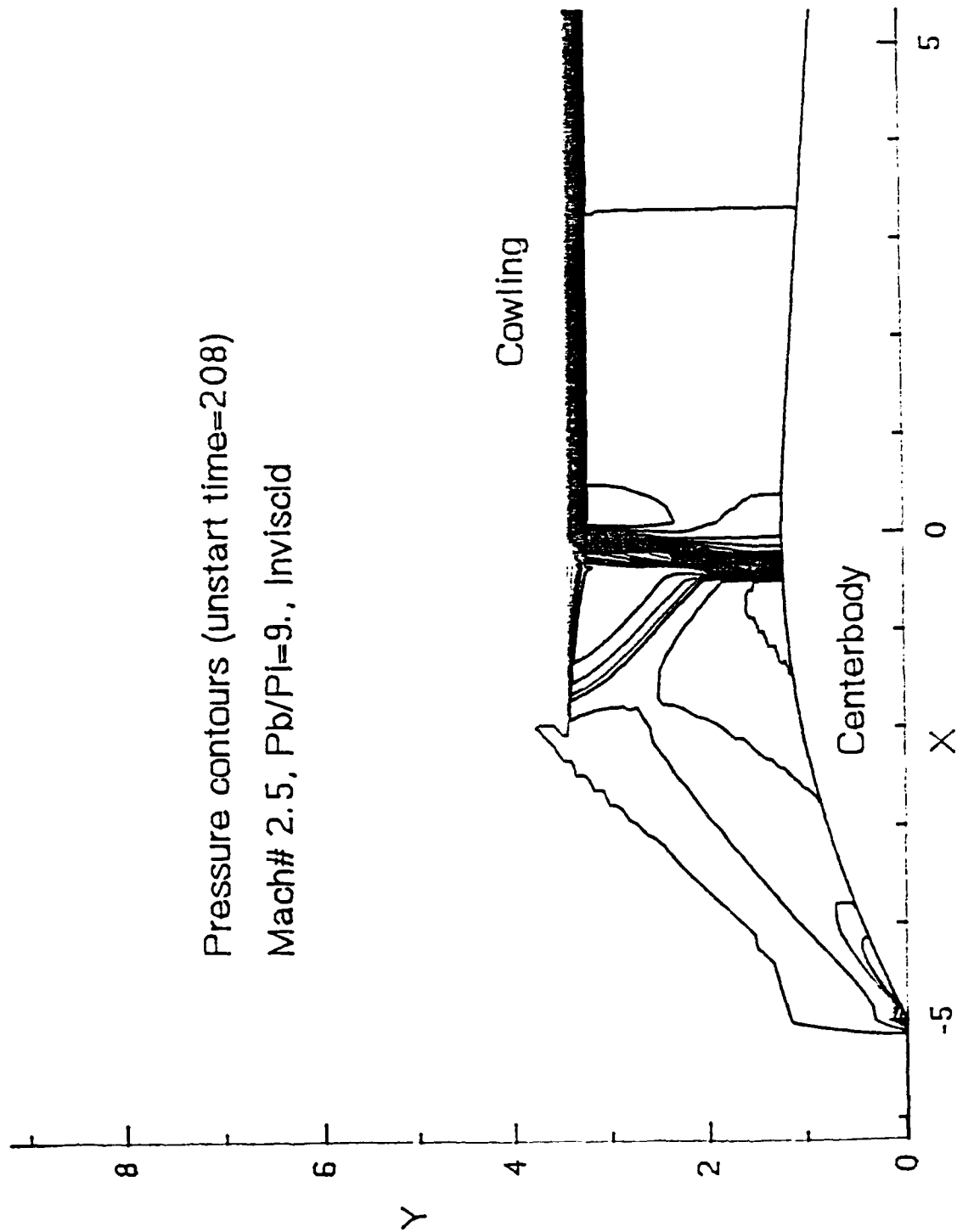


Fig. 1b. Inviscid pressure contours - shock closer to the cowl lip

plt

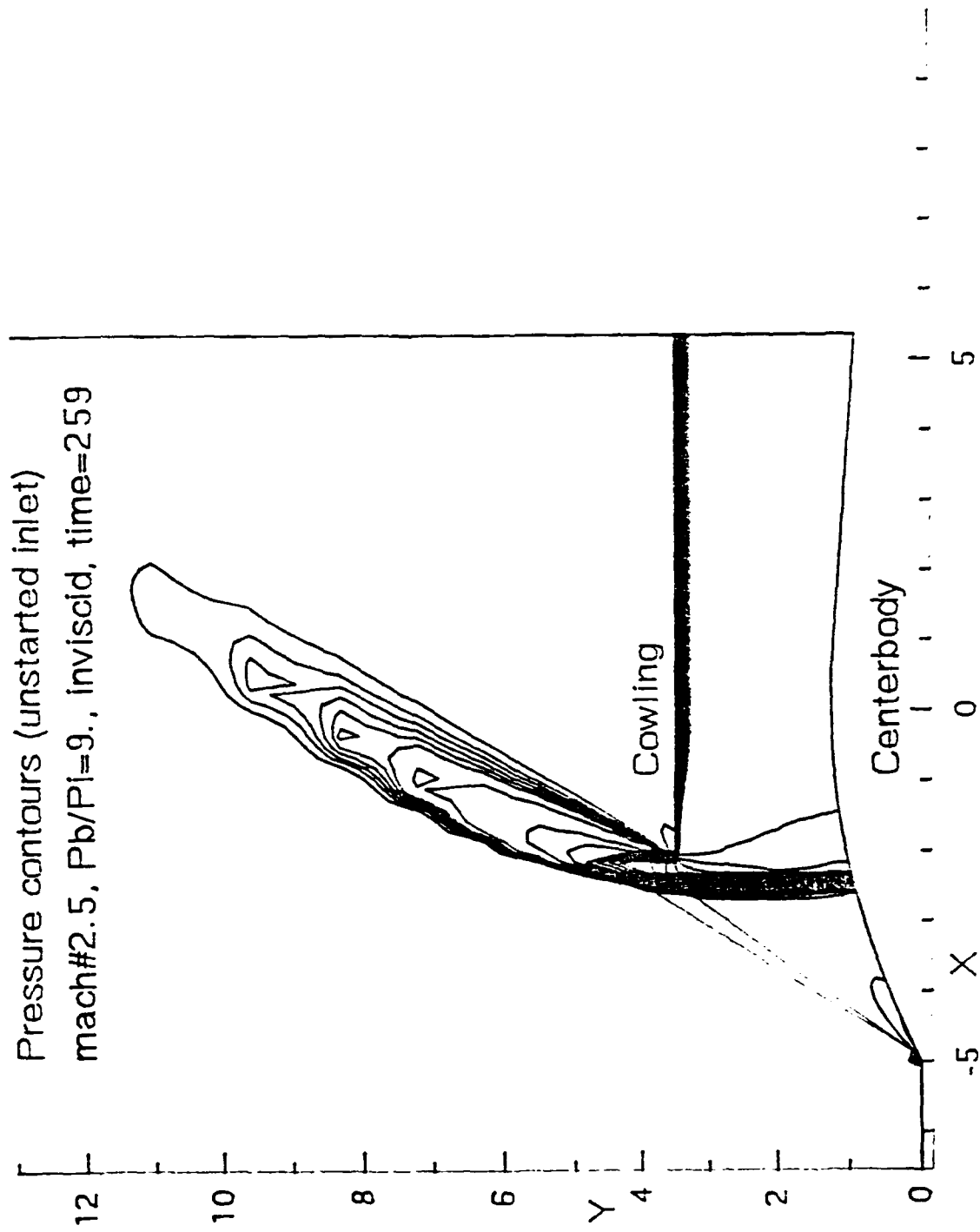


Fig. 1c. Inviscid pressure contours - unstart of the inlet

20.plt ** contour plot

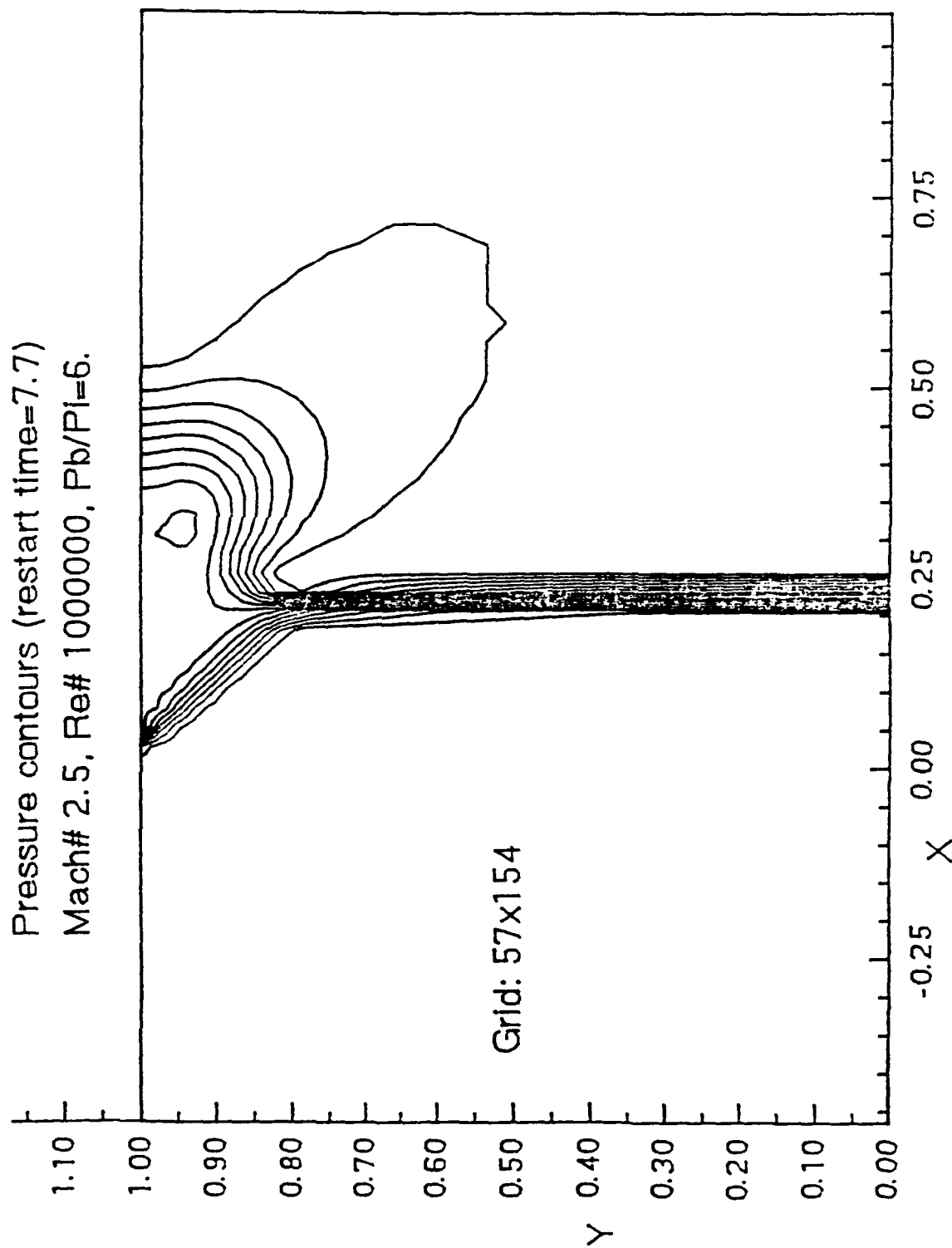


Fig. 1d. Two-dimensional parallel plate inlet-viscous flow

/20.plt ** vector plot

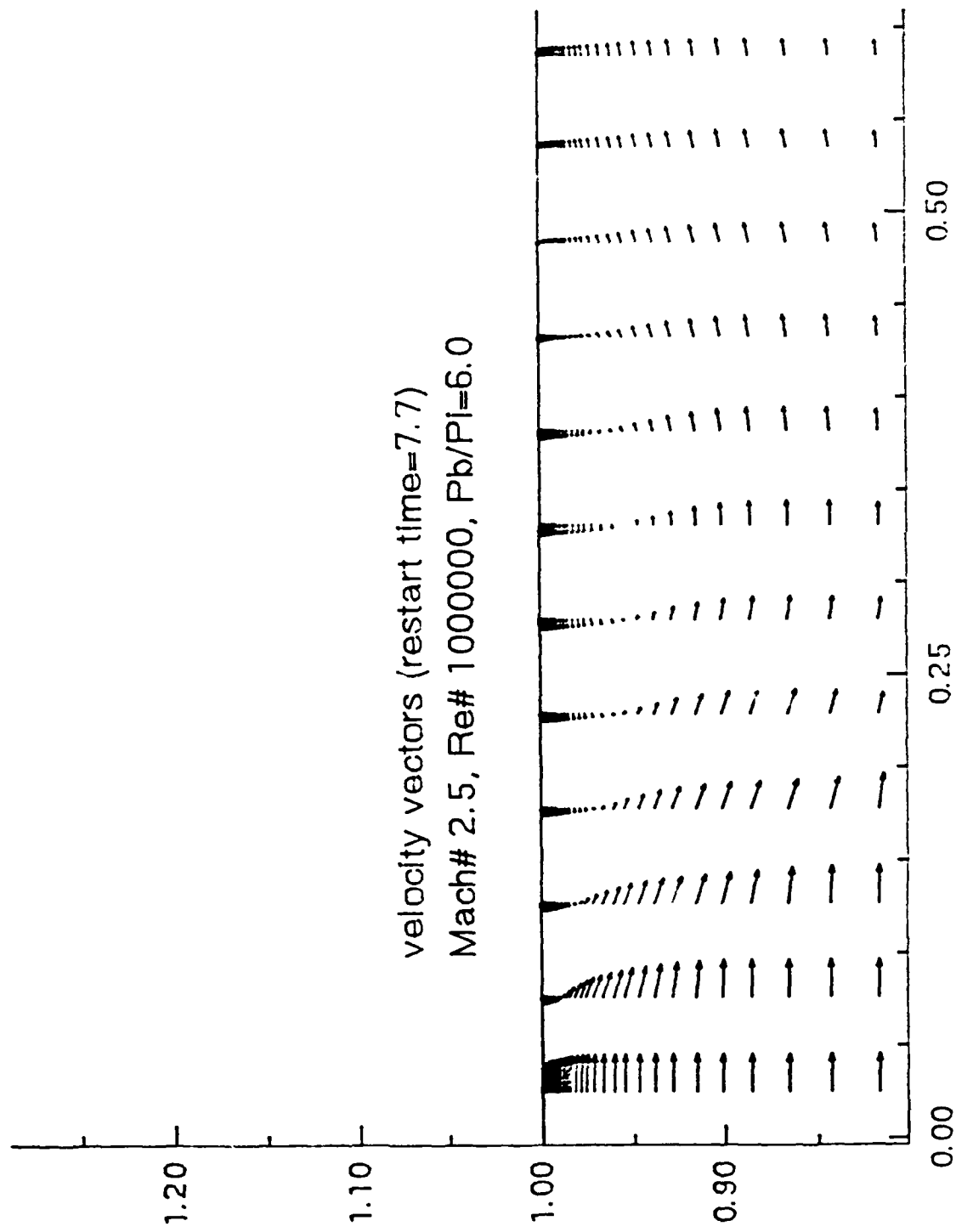


Fig. 1e. Velocity vector plot, two-dimensional Inlet

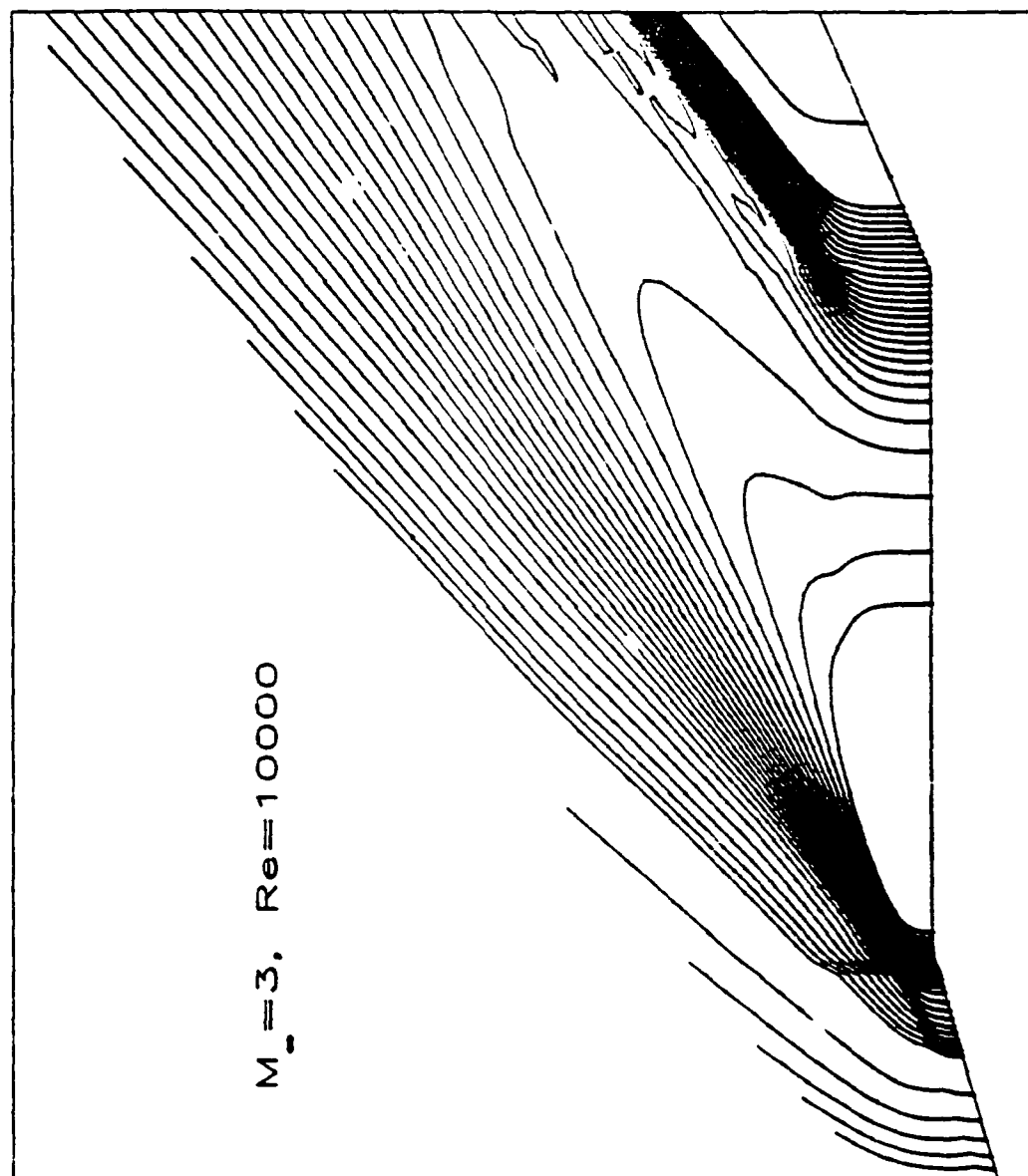


Fig 2a. Cone-Cylinder-Flare : Pressure Contours

Fig. 2b. Cone-Cylinder-Flare

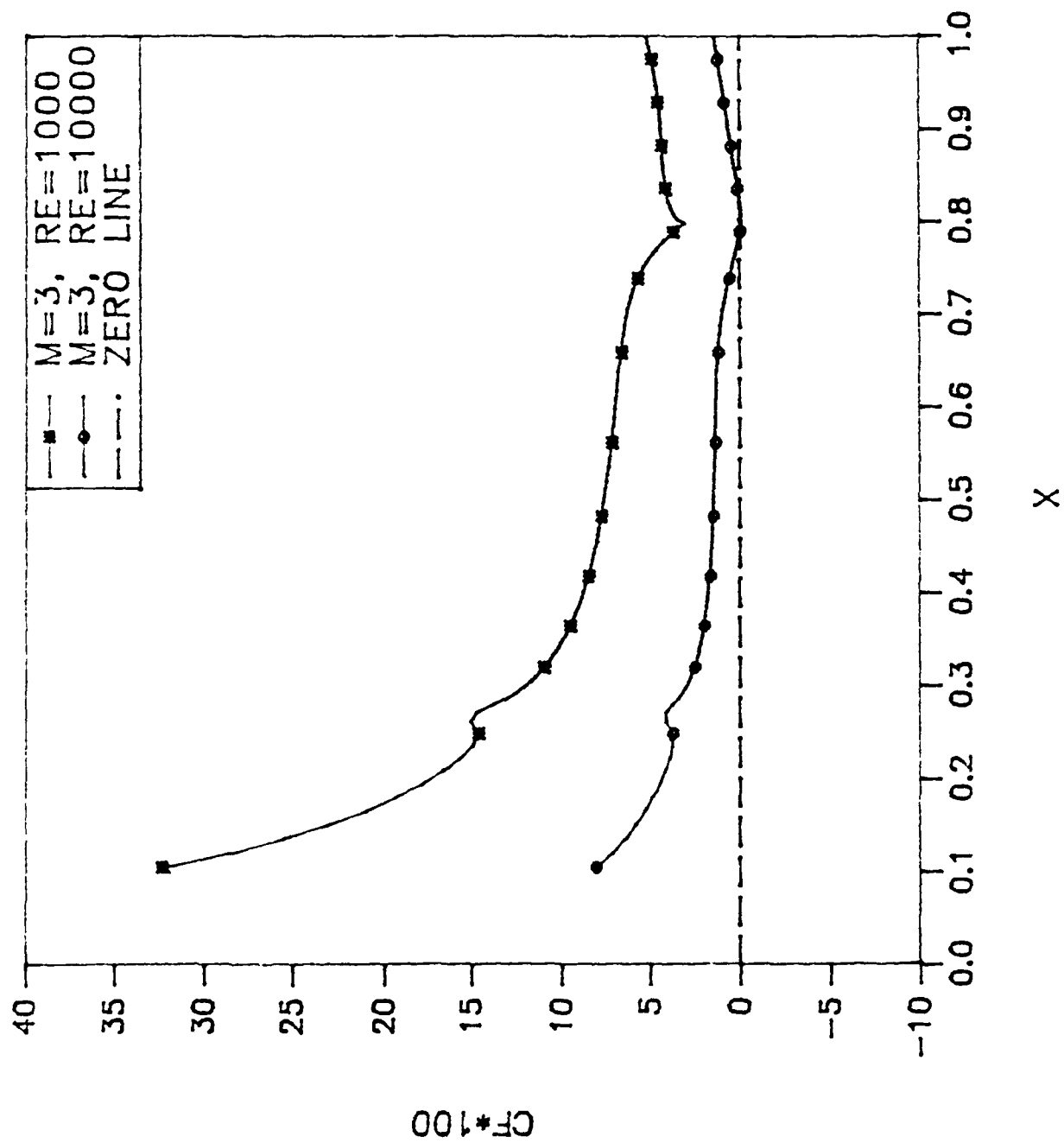
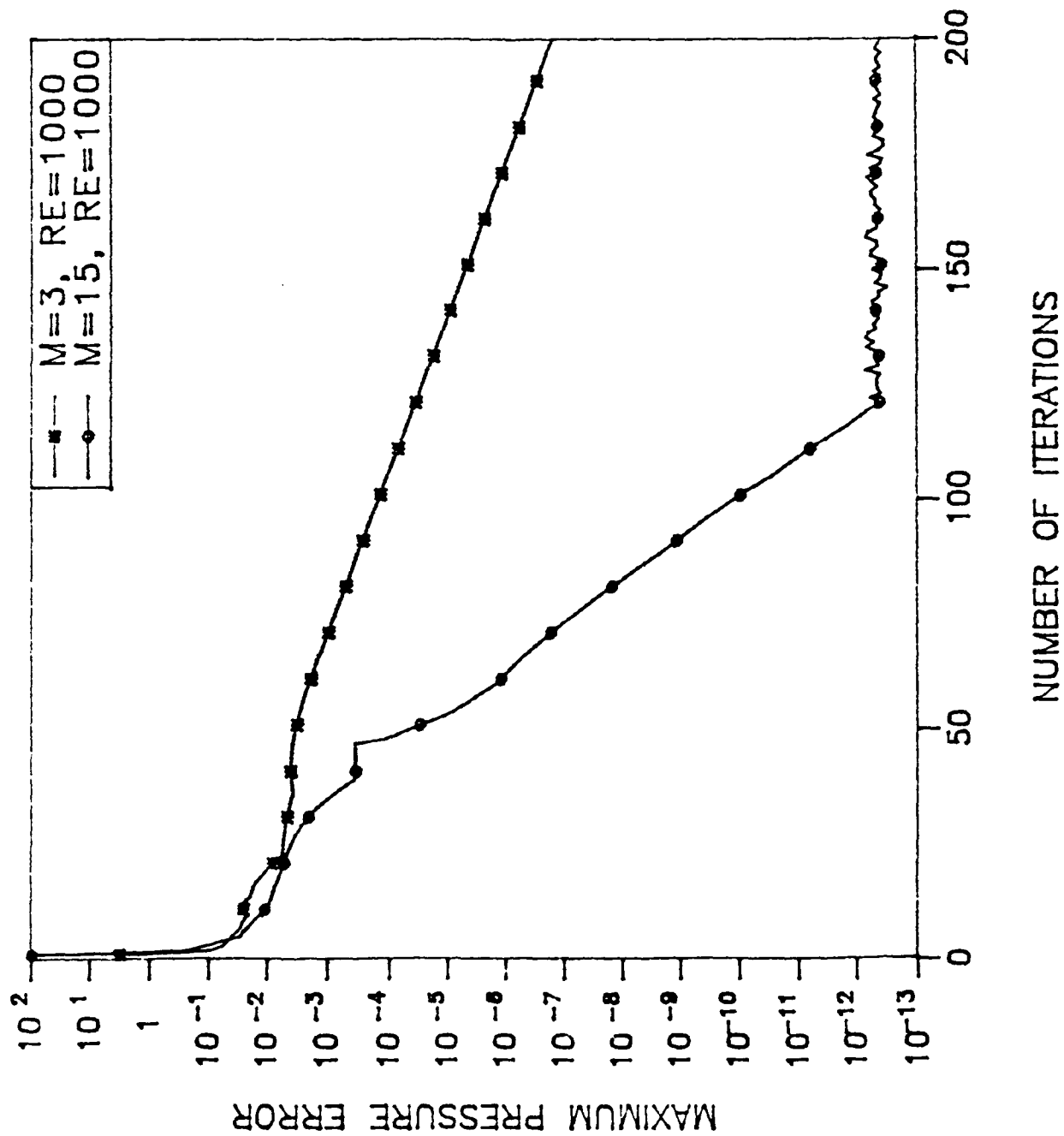


Fig. 2c. Cone-Cylinder-Flare



p.plt ** Pressure Distribution

Mach = 7.0
adiabatic wall
Re = 66000

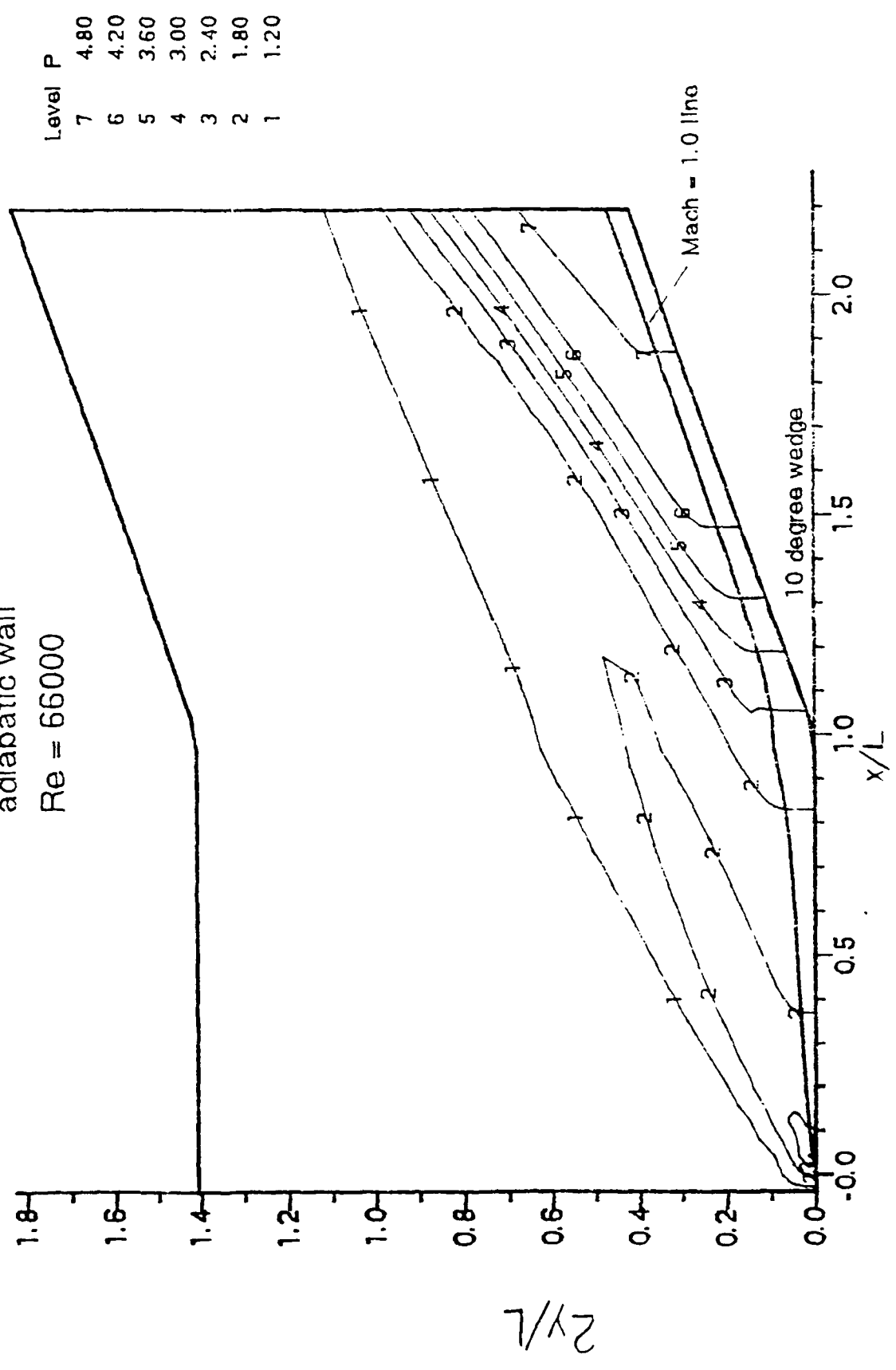


Fig. 3a. Pressure contours: 2-D ramp

vel7.plt ** Velocity Distribution

VELOCITY VECTOR PLOT

Mach = 7.0, adiabatic wall, Re = 66000

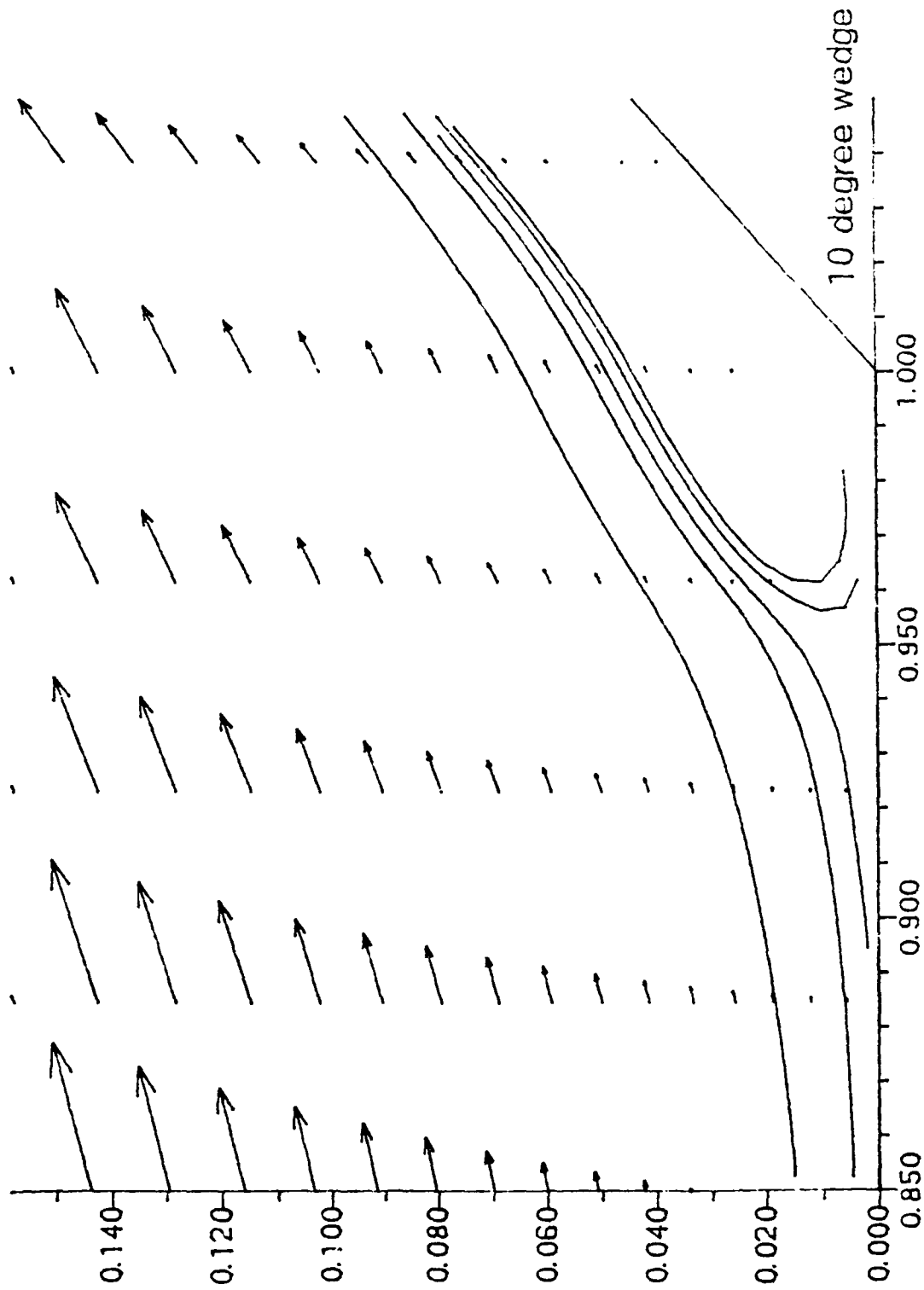


Fig. 3b. Velocity vectors: 2-D ramp. Note: y-dimension stretched by factor of 5

op7.plt **

WALL FRICTION AND PRESSURE

Mach = 7.0, adiabatic wall, $Re = 66000$, 10 degree wedge

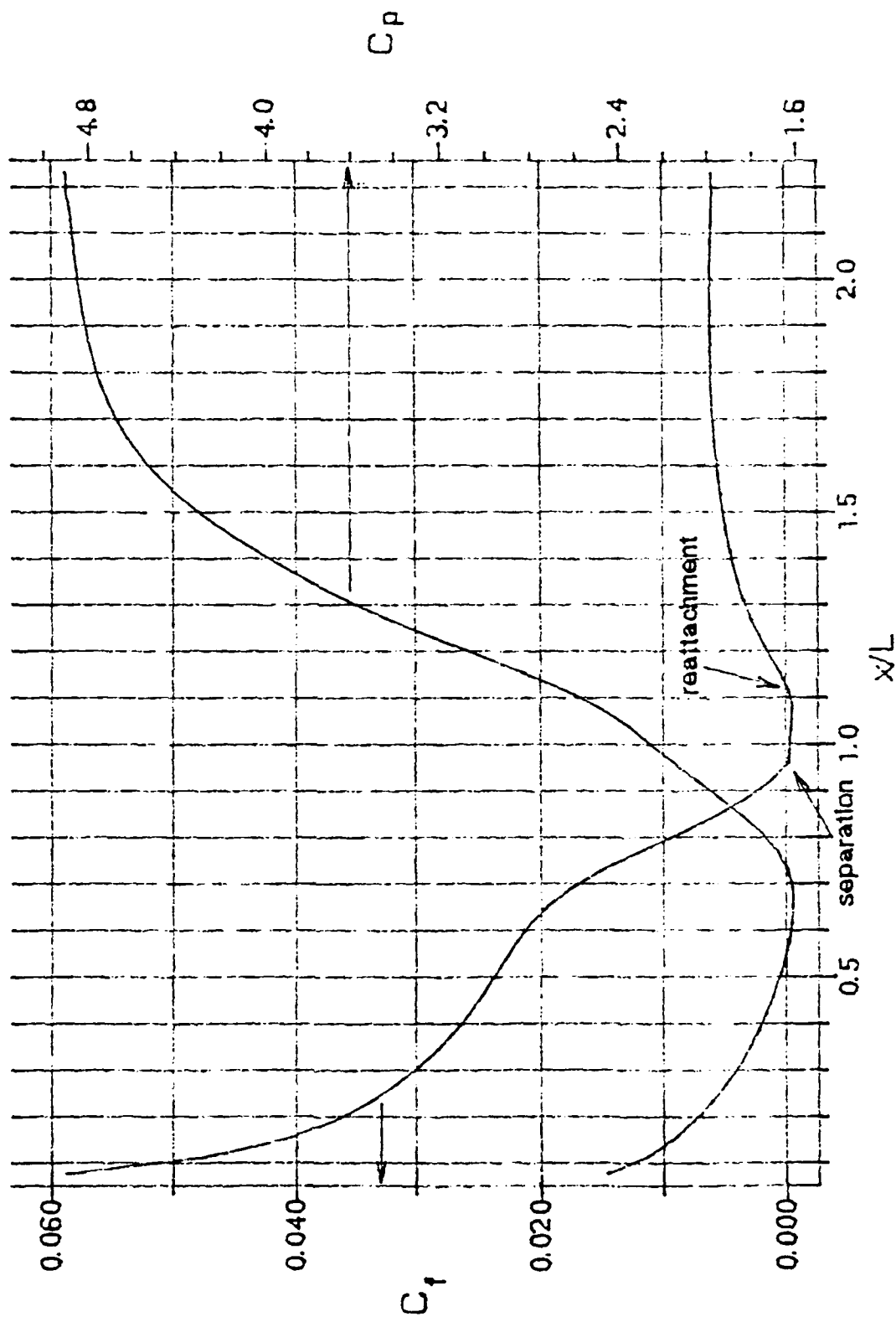


Fig. 3c. Skin friction & pressure distribution: 2-D ramp

pres1.plt ** contour plot

Pressure Contours for $\theta=0$. $M=0.6$

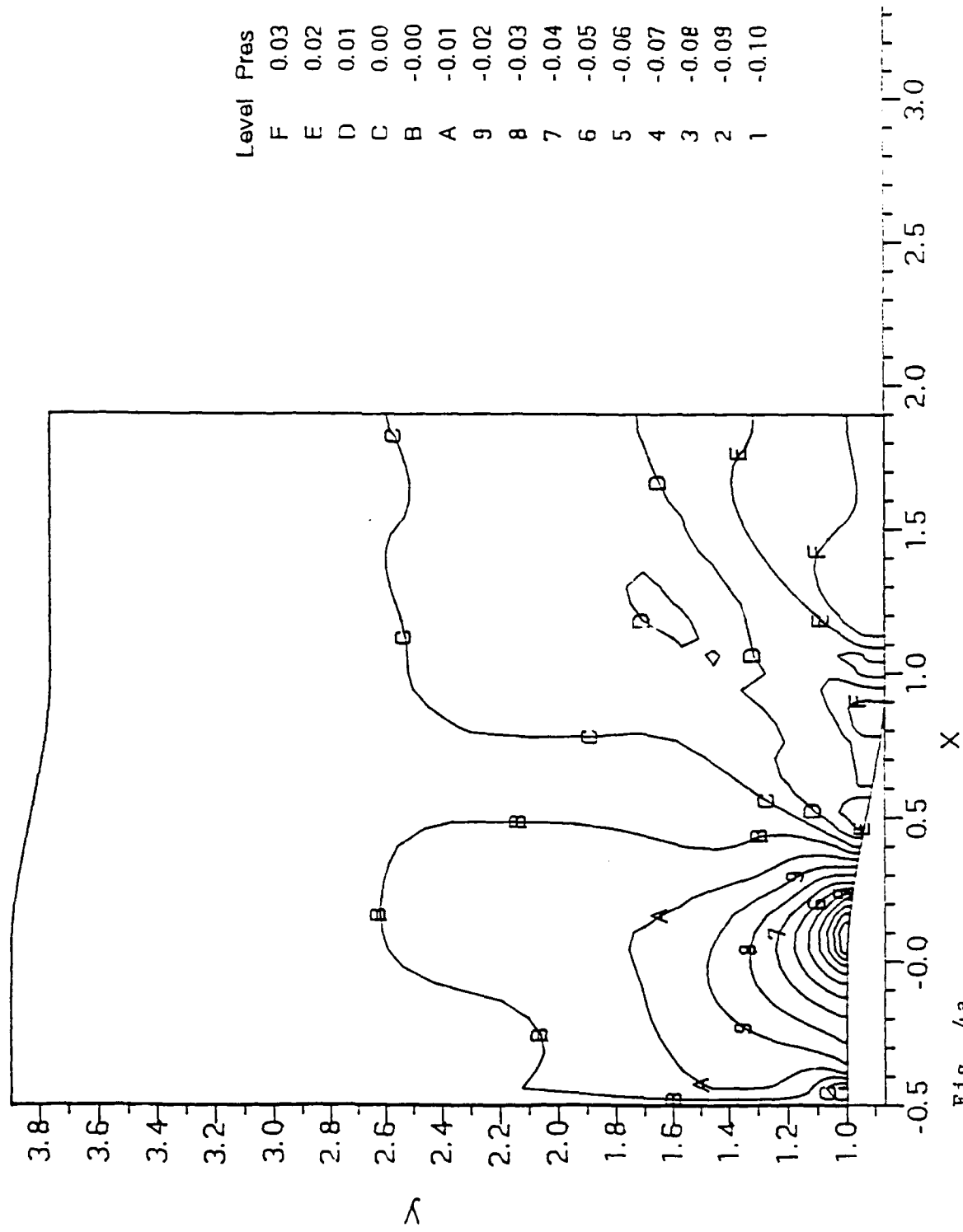


Fig. 4a.

pres2.plt ** contour plot

Pressure Contours for theta=45. M=0.6

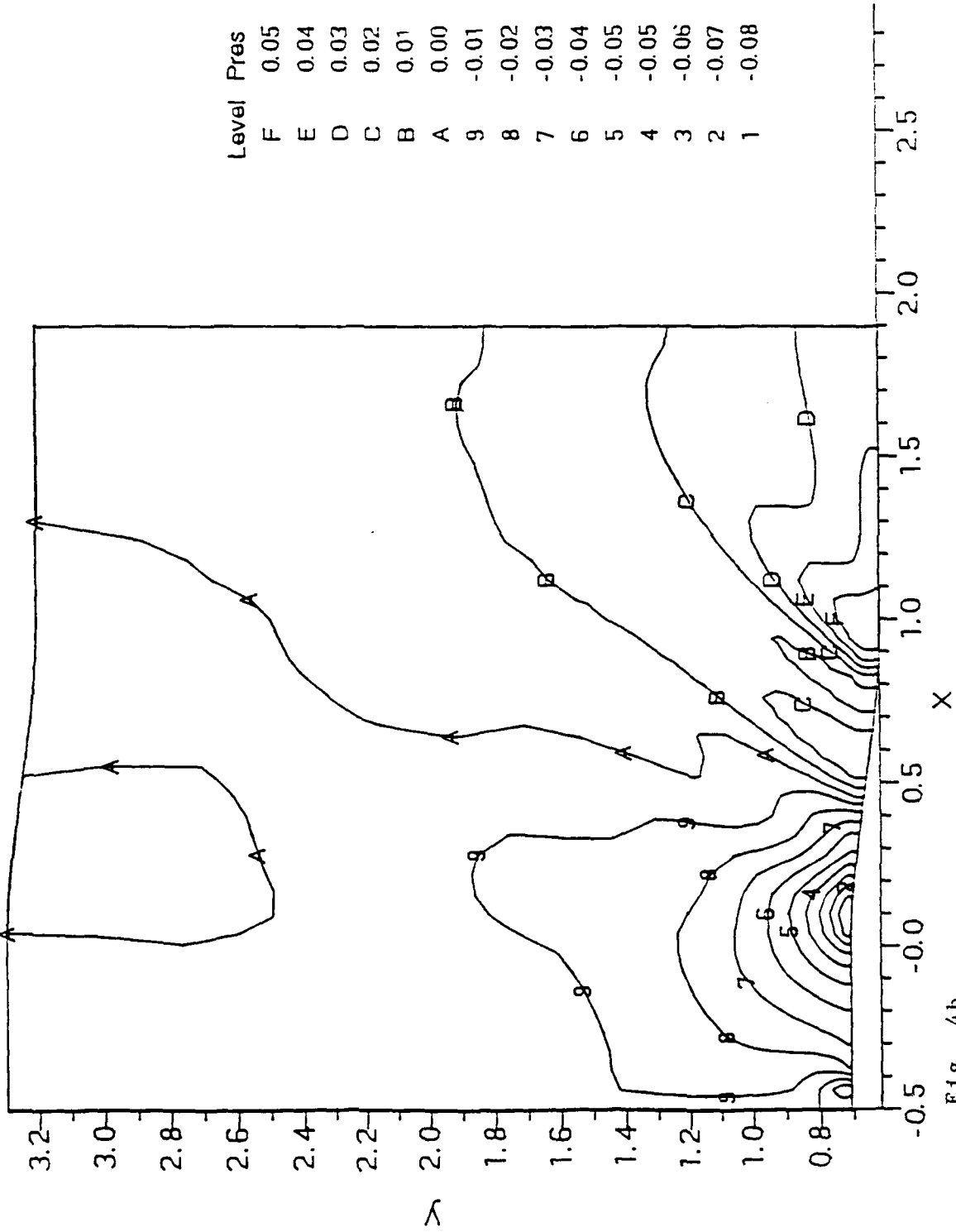


Fig. 4b.

pres3.plt ** contour plot

Pressure Contours for theta=90. M=0.6

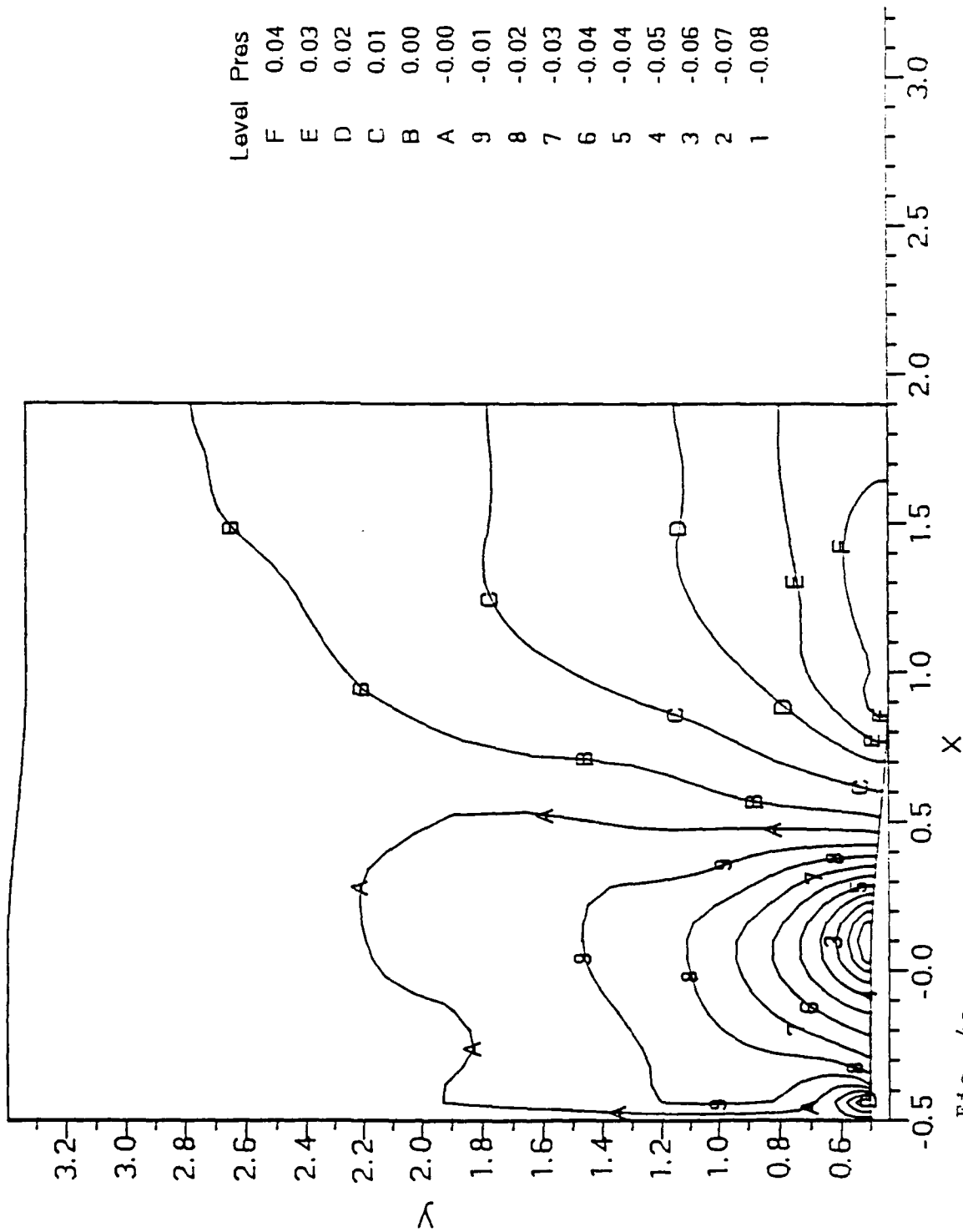


Fig. 4c.

Skin-Friction Parameter for $M=0.6$

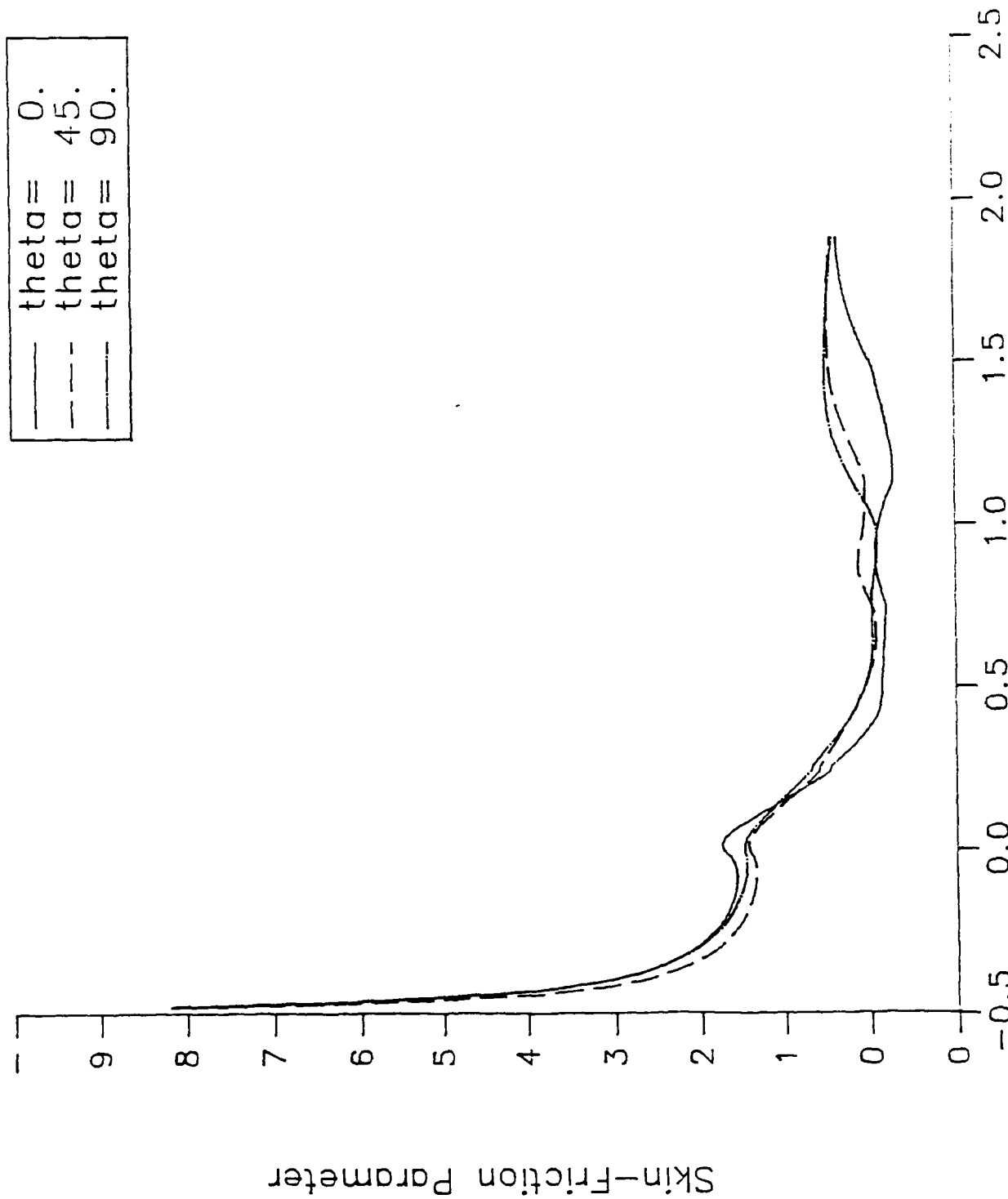


Fig. 4d.

Pressure Coefficient for Mach=0.3 at=0.86

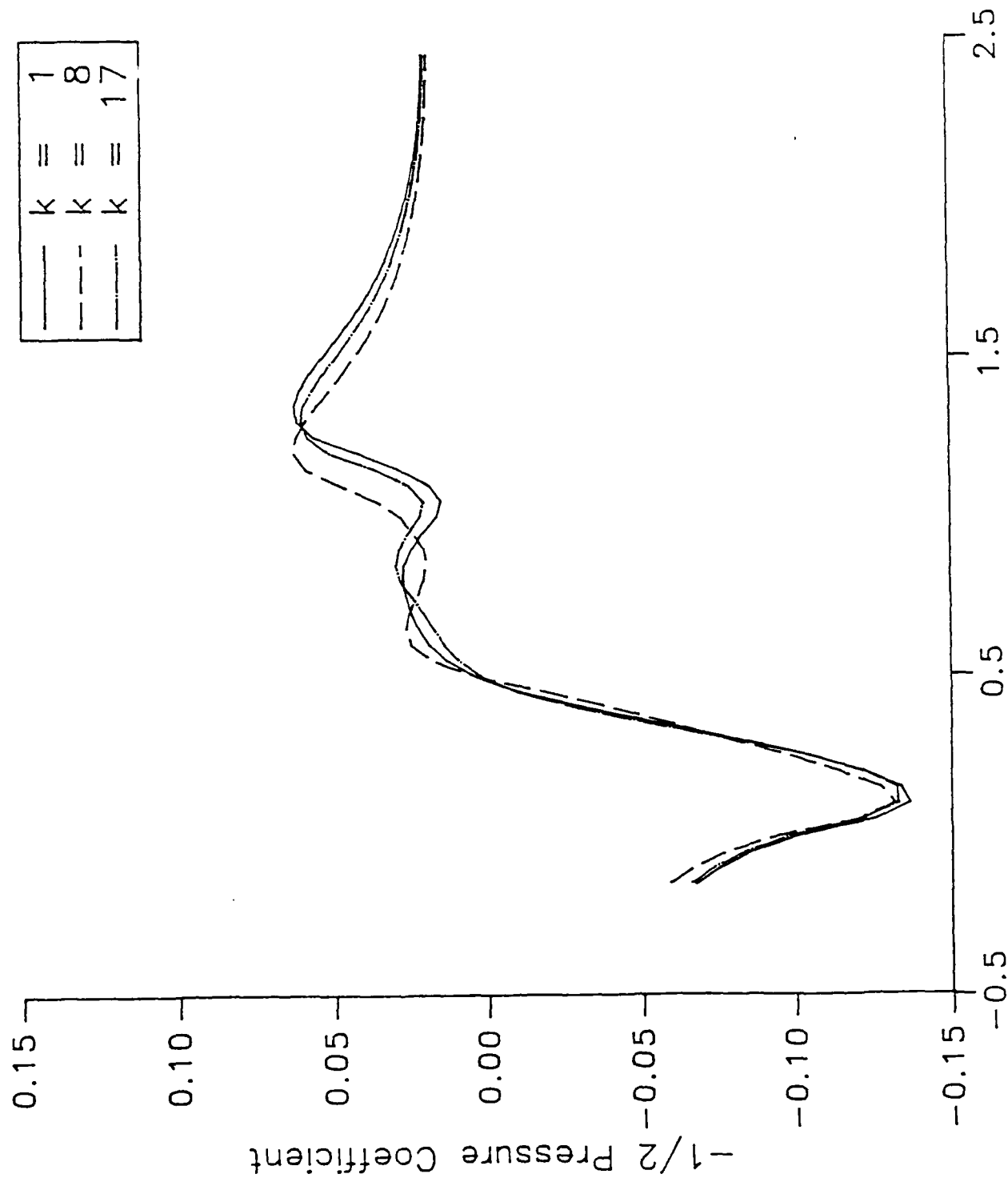


Fig. 5a. Afterbody configuration with hyperellipse cross-section - coefficient of pressure.

Skin-Friction Parameter for $M=0.3$ $Re=100000$.

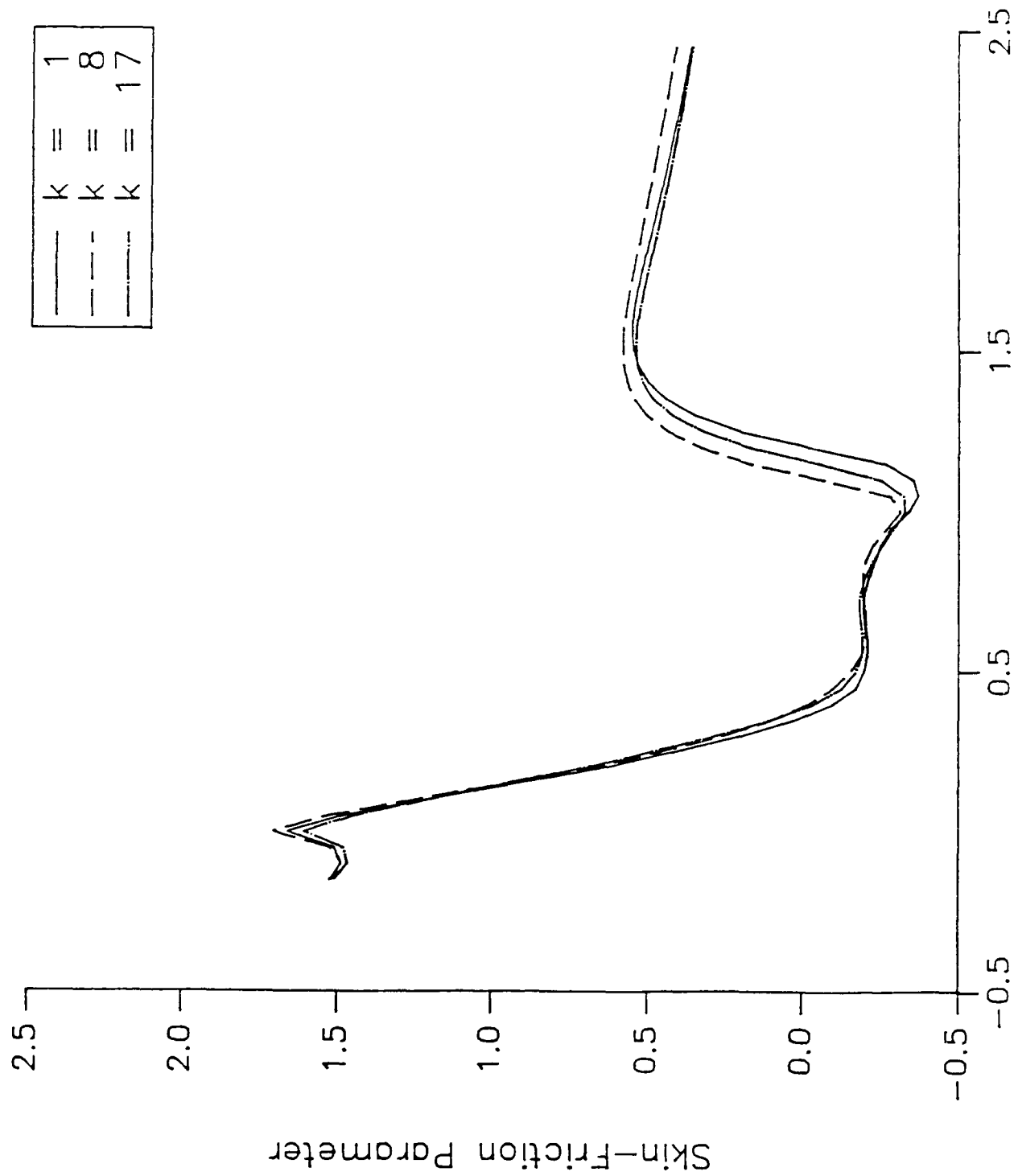


Fig. 5b. Afterbody configuration with hyperellipse cross-section - coefficient of skin friction.

Mach No. = 0.8 $Re = 1000000$
Afterbody Configuration

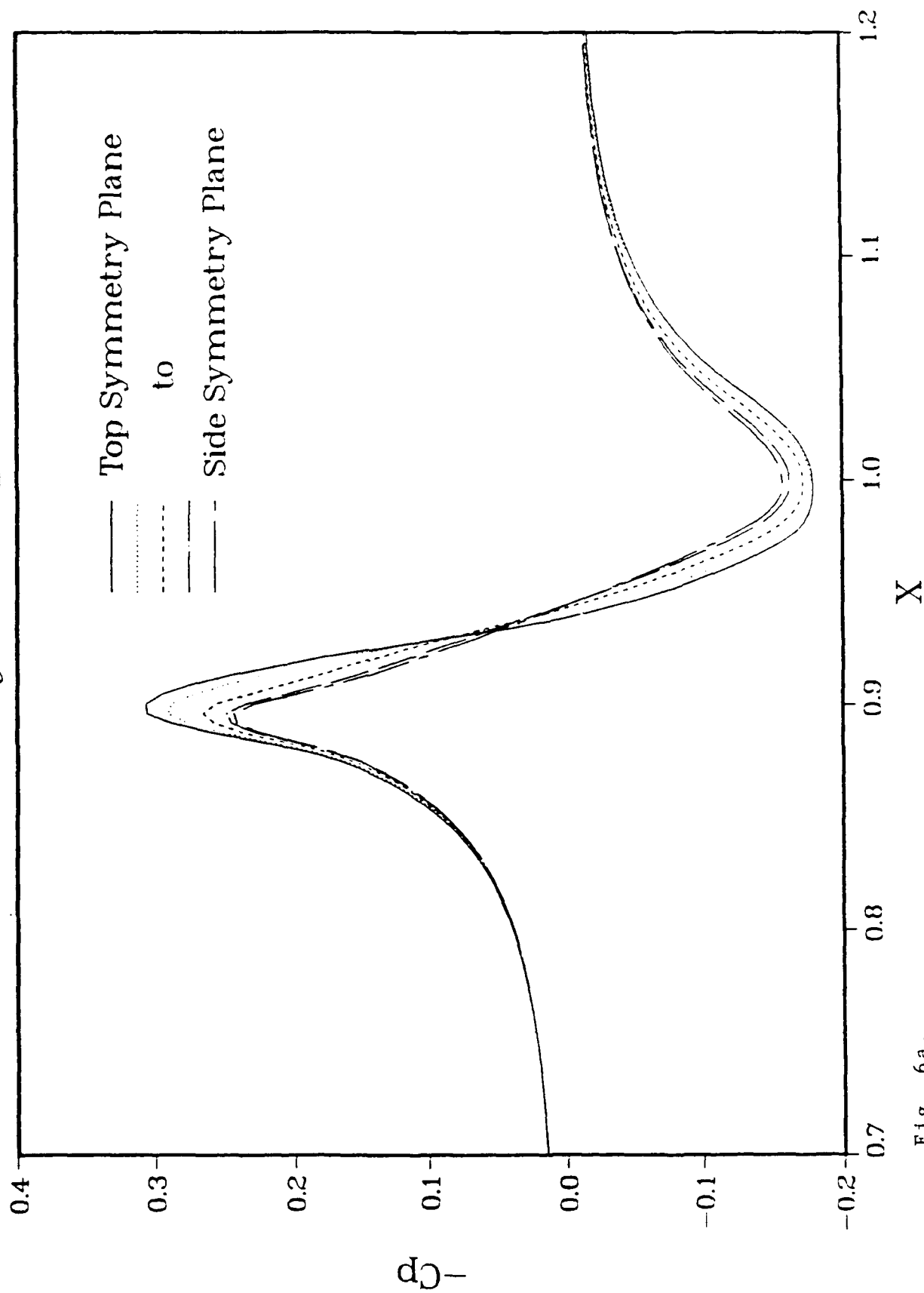


Fig. 6a.

Mach No. = 0.8 $Re = 1000000$
Afterbody Configuration

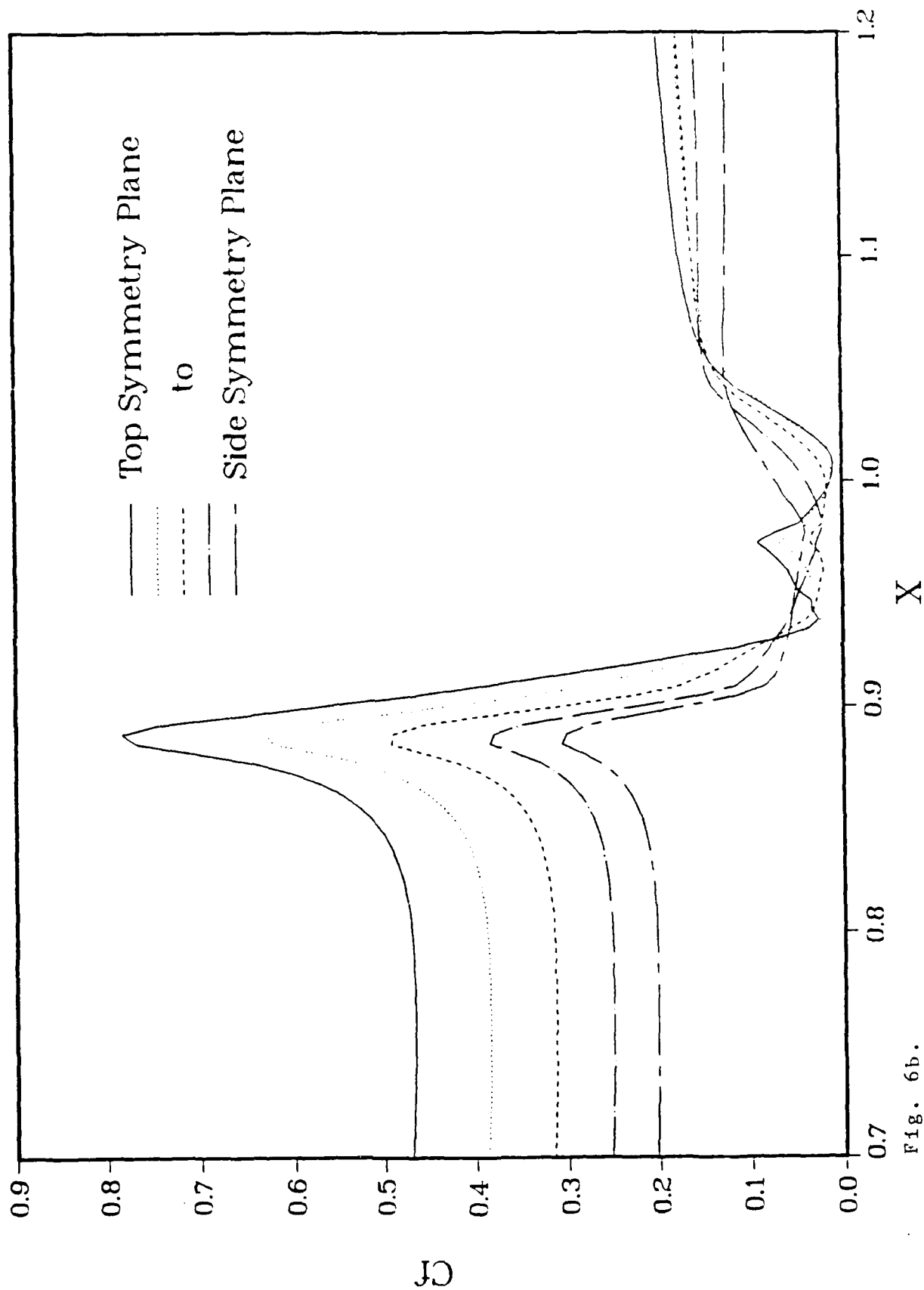


Fig. 6b.

Mach No. = 0.9 $Re = 1000000$ Afterbody Configuration

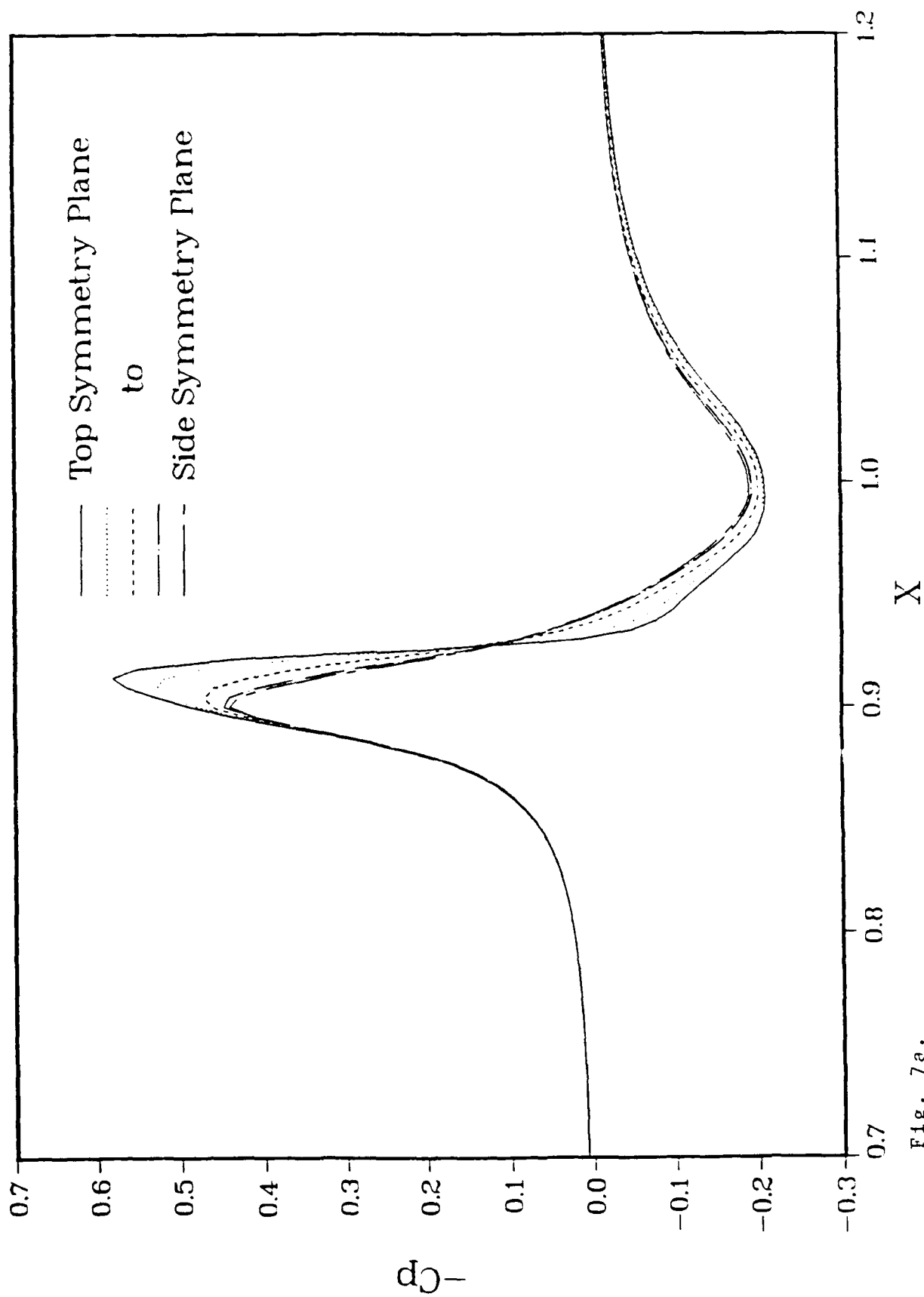


Fig. 7a.

Mach No. = 0.9 Re = 1000000
Afterbody Configuration

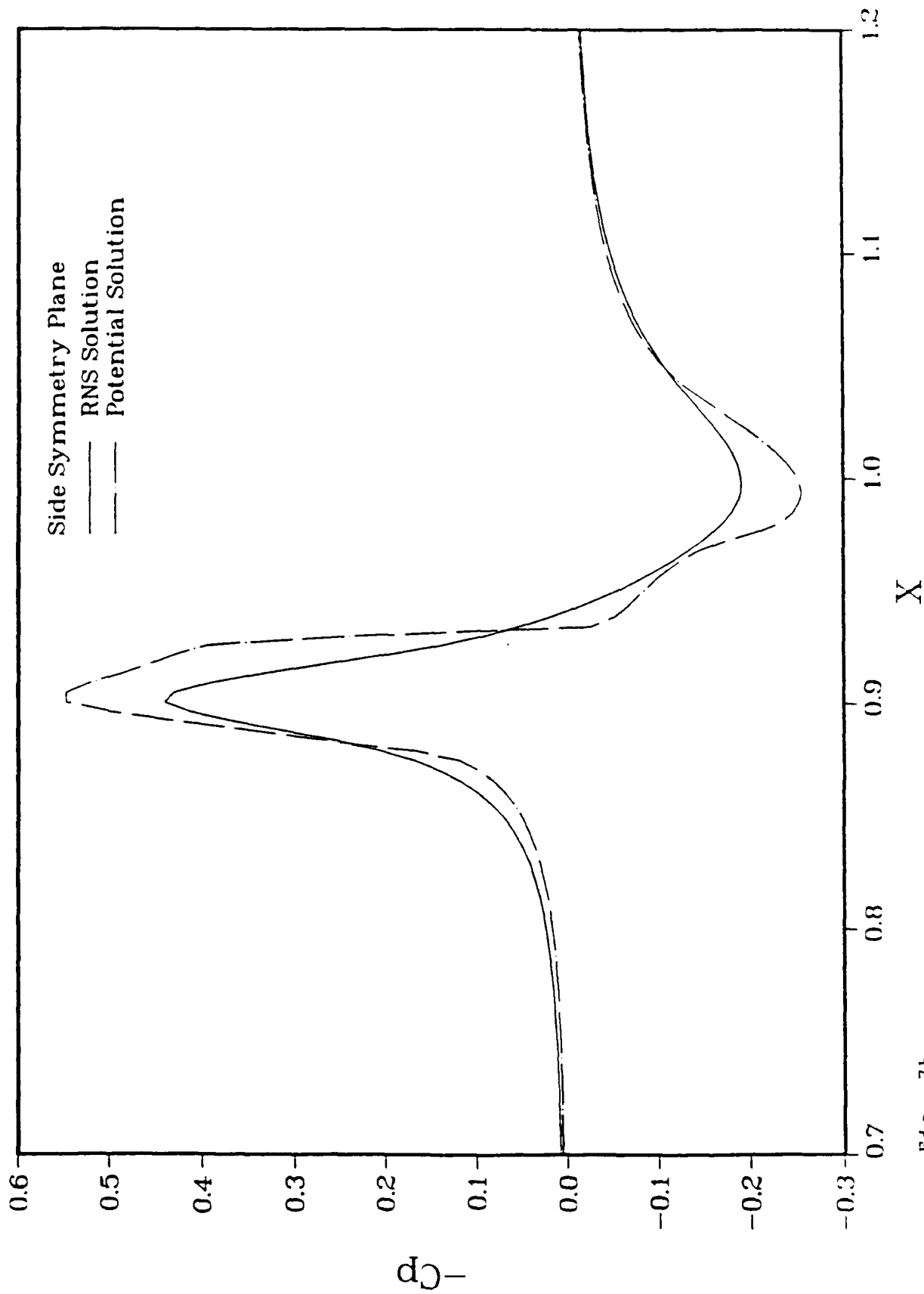


Fig. 7b.