

DTIC FILE COPY

2

AD-A206 364

Computational Study of
High Supersonic Flow Over Boattails with
Centered Propulsive Jets

NEAR TR 397

by

Robert E. Childs

DTIC
SELECTED
22 MAR 1989
S E D

This document has been approved
for public release and sale; its
distribution is unlimited.



NIELSEN ENGINEERING AND RESEARCH INC

Computational Study of
High Supersonic Flow Over Boattails with
Centered Propulsive Jets

NEAR TR 397

by

Robert E. Childs

Final Report

January 1989

Prepared under Contract DAAL03-86-C-0002

For

U.S. Army Research Office
P.O. Box 12211
Research Triangle Park, NC 27709-2211

DTIC
SELECTED
1989
S E D

This document has been approved
for public release and sale in
unlimited quantities.

NIELSEN ENGINEERING & RESEARCH, INC.
510 Clyde Avenue Mountain View, CA 94043-2287
Telephone: (415) 968-9457 • Facsimile: (415) 968-1410

UNCLASSIFIED

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		3. DISTRIBUTION/AVAILABILITY OF REPORT Approved for public release; distribution unlimited.			
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE					
4. PERFORMING ORGANIZATION REPORT NUMBER(S) NEAR TR 397		5. MONITORING ORGANIZATION REPORT NUMBER(S)			
6a. NAME OF PERFORMING ORGANIZATION Nielsen Engineering & Research, Inc.		6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION U.S. Army Research Office		
6c. ADDRESS (City, State, and ZIP Code) 510 Clyde Avenue Mountain View, CA 94043-2287		7b. ADDRESS (City, State, and ZIP Code) P.O. Box 12211 Research Triangle Park, NC 27709-2211			
8a. NAME OF FUNDING/SPONSORING ORGANIZATION USARO		8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER Contract No. DAAL03-86-C-0002		
8c. ADDRESS (City, State, and ZIP Code) P.O. Box 12211 Research Triangle Park, NC 27709-2211		10. SOURCE OF FUNDING NUMBERS			
		PROGRAM ELEMENT NO.	PROJECT NO. P-22354-EG	TASK NO.	WORK UNIT ACCESSION NO.
11. TITLE (Include Security Classification) Computational Study of High Supersonic Flow over Boattails with Centered Propulsive Jets					
12. PERSONAL AUTHOR(S) Robert E. Childs					
13a. TYPE OF REPORT Final		13b. TIME COVERED FROM 11/01/85 to 10/31/88	14. DATE OF REPORT (Year, Month, Day) 1989 January 31		15. PAGE COUNT
16. SUPPLEMENTARY NOTATION The views, opinions, and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position policy, or decision, unless so designated by other documentation.					
17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number) Afterbody flows, Base Drag; Turbulence Modeling; Computational Fluid Dynamics; Truncation error estimates.		
FIELD	GROUP	SUB-GROUP			
20	04				
01	01				
19. ABSTRACT (Continue on reverse if necessary and identify by block number) → The present study concerns the ability to achieve good accuracy in missile afterbody flow predictions based on the Reynolds-averaged Navier-Stokes equations. The problems of numerical accuracy and turbulence modeling are addressed. By employing grids which were well clustered and which have grid lines aligned with streamlines in shear layers, it was possible to obtain numerically accurate solutions and to obtain solutions at very high nozzle pressure ratios. The effects on turbulence of high Mach numbers and streamline curvature were examined. It was demonstrated that these phenomena have a significant effect on the turbulence and on the global flow characteristics such as base drag. Changes in base drag on the order of +/-20% for some afterbody flows can be attributed to Mach number and curvature effects. A modified form of the k-ε turbulence model which gives good accuracy for flows involving high Mach number and streamline curvature is presented. <i>Keywords:</i>					
20. DISTRIBUTION/AVAILABILITY OF ABSTRACT <input checked="" type="checkbox"/> UNCLASSIFIED/UNLIMITED <input type="checkbox"/> SAME AS RPT <input type="checkbox"/> DTIC USERS			21. ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED		
22a. NAME OF RESPONSIBLE INDIVIDUAL Dr. Thomas L. Doligalski			22b. TELEPHONE (Include Area Code) (919)549-0641	22c. OFFICE SYMBOL	

DD Form 1473, JUN 86

Previous editions are obsolete.

SECURITY CLASSIFICATION OF THIS PAGE

UNCLASSIFIED

TABLE OF CONTENTS

SUMMARY	iv
INTRODUCTION AND OBJECTIVES	1
BRIEF CHRONOLOGY OF RESEARCH EFFORT	2
RESEARCH RESULTS	4
Discretization Accuracy	4
Turbulence Modeling	6
The k-ε Turbulence Model	6
Mach Number Effects	7
Modification for Streamline Curvature	8
Modeling Effects in Afterbody Flows	10
SUMMARY OF ACCOMPLISHMENTS	12
Numerical Accuracy	12
Turbulence Modeling	12
RECOMMENDATIONS FOR AN ACCURATE PREDICTION METHOD	13
RECOMMENDED FUTURE RESEARCH	13
PUBLICATIONS	14
TECHNICAL PERSONNEL	14
REFERENCES	15
FIGURES	16

Approved For	
NARS (0001)	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By _____	
Distribution/ _____	
Availability Codes	
Dist	Avail and/or Special
A-1	

SUMMARY

Flow in the afterbody region is important to the performance of many flight vehicles, including missiles and ordnance projectiles. Previous studies have shown that flow prediction methods based on the Navier-Stokes equations are often inaccurate for these flows. The present study concerns the ability to achieve good accuracy in afterbody flow predictions based on the Reynolds-averaged Navier-Stokes equations. The problems of numerical accuracy and turbulence modeling are addressed.

A method for estimating truncation errors was used to determine the errors associated with numerical solution of the governing equations on a finite volume grid. A method of adapting the grid to the computed solution to reduce numerical error was used. By employing grids which are well clustered and which have grid lines aligned with streamlines in shear layers, it was possible to obtain numerically accurate solutions and to obtain solutions at very high nozzle pressure ratios. The effects on turbulence of high Mach numbers and streamline curvature were examined. It was demonstrated that these phenomena have a significant effect on the turbulence and on the global flow characteristics such as base drag. Changes in base drag on the order of $\pm 20\%$ for some afterbody flows can be attributed to Mach number and curvature effects. A modified form of the $k-\epsilon$ turbulence model which gives reasonably good accuracy for flows involving high Mach number and streamline curvature is presented.

Several conclusions arise from the work. (1) By employing solution-adapted grids, it is possible to obtain axisymmetric solutions in which the grid-dependence is negligible for most purposes. (2) By employing grids in which grid lines are aligned with streamlines at the nozzle exit, the computational difficulties at high nozzle pressure ratios can be eliminated. (3) The effects on turbulence of high Mach numbers and streamline curvature can be significant in afterbody flows. (4) The $k-\epsilon$ model with modifications for Mach number and curvature gives reasonably good accuracy for the afterbody flows studied in this work. (5) Because of fortuitous error cancellation, the standard $k-\epsilon$ model may give reasonably good base pressures in some cases; however, this apparent accuracy will not, in general, extend to details of the flow, such as the velocity field, or to a wide range of cases. A similar conclusion is probably applicable to other turbulence models which have not been designed to account for the complex turbulence phenomena which occur in afterbody flows.

INTRODUCTION AND OBJECTIVES

Numerical flow prediction methods based on the Euler and Navier-Stokes equations have become important design and analysis tools in the Army and elsewhere. They are cost effective and can provide insight into flow phenomena which are difficult to obtain by experimental means. These methods can be used to study higher Mach numbers, Reynolds numbers, or enthalpy conditions than can be achieved in ground based test facilities. They are being relied upon to an increasing extent in the design and analysis of a wide variety of flight vehicles. However, there is a danger in this approach. Flow prediction methods are not always as accurate as they need to be. For some flows these methods are exceedingly inaccurate. The flow in the afterbody region of missiles is one case in which prediction methods have been shown to be inaccurate in past studies.^{1,2}

In 1985 Petrie and Walker¹ presented the results of a blind test evaluation for bluff-base afterbody flows of several prediction methods. The boundary conditions needed to perform the calculations were provided from the experiment, but the experimental results were withheld until the computational results had been submitted. None of the computational methods predicted the base drag with less than 30% error, and the error was substantially greater for some methods. These errors were attributed to inadequate resolution (coarse grids) and to deficiencies in the turbulence models which were used.

Since the release of Petrie and Walker's data, several calculations have been able to achieve reasonable agreement with the measured base pressure. However, these studies have not demonstrated that the grid-independent solutions agree well with experiment. Such a demonstration is necessary because numerical errors can mask errors in the turbulence modeling and vice versa. Fortuitous error cancellation is not an acceptable means of achieving good prediction accuracy for a wide variety of flows, although it may give the appearance that prediction methods are accurate in some calculations.

The objectives of the present work are to identify the specific sources of errors in Navier-Stokes calculations of afterbody flows and to develop means of reducing or eliminating these errors. Based on past experience, this work focuses on truncation errors, that is, grid-dependence in the computed flow fields, and on turbulence modeling for the curved and high Mach number shear layers which occur in afterbody flows.

BRIEF CHRONOLOGY OF RESEARCH EFFORT

The research effort consisted of several distinct phases. The initial part of the work involved the acquisition and development of some basic numerical methods, a grid generation scheme and a Navier-Stokes solver with a two-equation turbulence model.

The second stage of the work involved an initial assessment of the extent to which computed solutions depended on the grid and on the turbulence model. In this early work, an uncertainty on the order of 12% in predicted base drag could be attributed directly to turbulence modeling and grid-dependence for the low pressure ratio case of Petrie and Walker. Serious difficulties with the numerical solution procedure were encountered at moderate nozzle pressure ratios (NPRs), the ratio of the static nozzle exit pressure to the freestream pressure.

It was also discovered that the standard $k-\epsilon$ model gave a reasonably good estimate of measured base pressures for the data of Petrie and Walker.¹ This result was somewhat unexpected because it appeared to contradict a basic premise of the original proposal, that "standard" models (in this case the standard $k-\epsilon$ model³) give poor accuracy for many of the phenomena which occur in base flows. Furthermore, a modification to the $k-\epsilon$ model designed to account for the effects of high Mach number degraded the accuracy of the prediction of base drag. However, this Mach number modification had been validated against experimental results for high speed shear layers. It was concluded at that time that the Mach number modification was performing properly, and, therefore, that using this one improvement in the model exposed other errors which had not yet been identified. The results of this work were presented at the AIAA 19th Fluid Dynamics, Plasma Dynamics and Lasers Conference, June 1987.⁴

The next step in the research was to address errors associated with numerical discretization of the Navier-Stokes and turbulence model equations. There were two important reasons for confronting this problem at that time. One was that numerical errors can be studied without having an accurate turbulence model. A truncation error analysis and grid refinement studies were used to assess numerical errors, instead of comparing calculations against experimental data. Conversely, it is essential to have numerically accurate solutions before trying to identify problems in turbulence models. Secondly, the numerical problem at the nozzle exit would cause the method to diverge at high NPRs, and this was an unacceptable situation.

A truncation error analysis method based on Richardson extrapolation was used to identify sources of large numerical errors. This method is described in the Results section below. The truncation error was found to be largest in the free shear layers in the base region and at the nozzle exit where the solution procedure would diverge at high NPRs. Consideration of the terms in the Navier-Stokes equations and the results from the truncation error analysis lead to the conclusion that aligning the grid lines with the flow direction in regions of solution gradients could significantly reduce the truncation error. This concept was tested, and it was found to be successful. The problems at high NPRs were completely eliminated, and the accuracy in free shear layers was improved. By aligning the grid lines with the streamlines and refining the grids in shear layers, it was possible to obtain solutions in which the

residual uncertainty in the computed base pressure was estimated to be less than 2% of the freestream pressure. These solutions are nearly grid-independent for most design or analysis purposes. The results of this work were presented at the AIAA 26th Aerospace Sciences Meeting, January 1988.⁵

The last stage of the research effort was again directed at the issue of turbulence modeling. While the inclusion of the Mach number modification had decreased the global accuracy in earlier work, it was felt that a modification of this type could not be omitted from a turbulence model for high speed flow. A modification to the $k-\epsilon$ model to account for the effects of streamline curvature was developed and included in the model. This modification yielded significant changes in the computed shear stresses and in the base pressure. The change in base pressure was roughly the same magnitude as that caused by the Mach number modification, but of the opposite sign. An appropriate model for afterbody flows should predict accurately at least Mach number and curvature effects. When both modifications are used in the $k-\epsilon$ model, the computed base pressures agreed well with the measured pressures for both cases reported by Petrie and Walker. The agreement between predicted and measured velocity fields was significantly better when the modified model was used than when the standard $k-\epsilon$ model was used. The results of this work were presented at the AIAA 27th Aerospace Sciences Meeting, January 1989.⁶

RESEARCH RESULTS

Numerical flow prediction methods can be viewed as consisting of two parts, the mathematical equations used to approximate fluid physics and the numerical methods used to obtain approximate solutions to these equations. For afterbody flows, the approximations in both of these parts can introduce significant errors in the predicted flow fields. The dominant errors are due to the modeling of the effects of turbulence and to the numerical discretization used to obtain solutions. These topics are addressed below. The goals of the work are to determine the specific sources of errors and to develop means of reducing these errors.

The present work has been based on a finite volume method⁷ and the $k-\epsilon$ turbulence model. The $k-\epsilon$ model is used for the basis of this work because it is, perhaps, the best all-purpose model currently available. However, the present work could have been performed with other models or differencing schemes, and most of the basic conclusions of the work would be essentially unchanged.

Discretization Accuracy

Discretization errors are responsible for two significant problems in afterbody calculations. One problem is that the computed solutions can depend significantly on the computational grid. In the blind test by Petrie and Walker,¹ for example, Deiwert's⁸ method gave very different base pressures when solution-adapted and non-adapted grids were used. Secondly, expansion fans exist at the edge of the high pressure propulsive jet and elsewhere which are difficult to resolve and which often cause large "undershoots" in pressure (see Figure 1). At moderate NPRs the undershoot can cause the pressure to become negative and terminate a calculation. Both problems are caused by truncation error.

The Navier-Stokes equations consist of terms like $\partial F/\partial x$, in which F represents the flux of mass, momentum, or energy due to inviscid and viscous transport. Truncation error results when the discrete approximation, $\delta F/\delta x$, of the gradient is inaccurate. Second order discretization is not especially accurate for computing $\delta F/\delta x$. For example, a simple analysis reveals that second order central differencing can represent only 15% of all Fourier modes which can be supported on a grid with 95% or better accuracy. The accuracy for high wave number components approaches zero. In other words, many grid points may be needed to accurately discretize a complex flow field containing several shear layers, and thin shear layers may require extreme grid clustering. An understanding of these truncation errors is needed.

An analysis method based on Richardson extrapolation has been used to study the dependence of truncation error on the grid and on the flow field. The use of this analysis technique was instrumental in understanding the problems at high NPRs and with global grid-dependence in the solution. The truncation error analysis method is presented below.

The steady state Navier-Stokes equation can be expressed in operator form as

$$L(q) = 0 \quad (1)$$

in which L represents the Navier-Stokes equations and q is the solution. The discretized form of these equation is

$$L_h(q) + h^m E_h(q) = 0 \quad (2)$$

in which h denotes some characteristic grid spacing, L_h represents the discretized Navier-Stokes equations, and $h^m E_h(q)$ is the truncation error which depends on the grid and the solution. The order of accuracy of the discretization scheme is m , so that $m = 2$ for the present second order method. If the grid spacing is doubled from h to $2h$, and the resulting differential operator L_{2h} is applied to the solution, the following equation is obtained.

$$L_{2h}(q) + (2h)^2 E_h(q) = 0 \quad (3)$$

By applying an analysis which is more detailed than warranted here (see Caruso and Childs⁵ for details), an explicit expression for estimating the truncation error in a computed solution is obtained from the difference between Equations (2) and (3).

$$h^2 E_h(q_h) \sim L_{2h}(q_h) \quad (4)$$

in which q_h is the solution of the discretized equations on the grid with characteristic grid spacing h . The operation described by Eq. (4) is easily performed, and the result provides a means of assessing the truncation error $h^2 E_h$ associated with the solution q_h . It is used in a qualitative sense to evaluate the effects of grid clustering and quality on the accuracy of the solution.

The truncation error analysis method was applied to afterbody flow solutions, and the errors were largest in shear layers where the grid lines were poorly aligned with the streamlines. These errors can often be reduced by simply increasing the number of grid points in the regions of high error. However, there is a way to reduce the truncation error without adding more points.

The fluxes, such as ρu or $\rho u^2 + p$, are continuous along streamlines, even through shocks, due to inviscid conservation-law mechanisms. Normal to streamlines, the fluxes are continuous due to viscous mechanisms. In high Reynolds number flows the viscous mechanisms are considerably weaker than inviscid mechanisms, and gradients in the solution normal to streamlines can be significantly greater than gradients along them. However, the convective fluxes normal to streamlines are zero, because there is no velocity component in that direction. In many of the shear layers in the afterbody region, the flux gradients due to pressure and turbulence will also be small. Hence, the flux gradients normal to streamlines are small. By reducing the total fluxes, the errors in numerical calculation of the flux gradients is also reduced. Hence, aligning the grid lines with the local flow direction in regions of solution gradients can significantly improve the accuracy with which gradient terms in the Navier-Stokes equation are computed.

Aligning the grid lines with the local flow direction at the nozzle exit also appears to eliminate the problems associated with high NPRs. Calculations on properly aligned grids were run at NPR = 10 for the Petrie and Walker¹ afterbody configuration and at NPR = 300 for Orbital Science Corporation's Pegasus launch vehicle⁹ (work funded separately) with no difficulties. Thus, there is reasonable confidence that a means of eliminating the problem of divergent calculations at high NPRs has been found.

The grid-dependence of global solution characteristics, such as base pressure, is also reduced by aligning the grid lines with streamlines in the dominant shear layers in the afterbody region. Figure 1 gives the base pressure for the NPR = 2.15 case of Petrie and Walker. The pressure undershoot is eliminated when the grid lines are properly aligned with the flow in the plume shear layer. As the grid is improved and refined the base pressure predicted with the standard k- ϵ model converges to a value slightly below the experimental data. From the results in Figure 1 and other results not presented here, it is estimated that the grid-independent base pressure predicted with this turbulence model will not differ from the present calculation on the finest grid by more than 2% of the freestream pressure. This level of accuracy is suitable for most design and analysis needs, and it greatly exceeds the accuracy that is expected from the turbulence model.

Turbulence Modeling

Many aspects of afterbody flows are predicted poorly by existing turbulence models. Few models can predict the effects on turbulence of the high Mach numbers, shear layer curvature, merging or "colliding" shear layers, and turbulence interactions with shocks or expansions which occur in many afterbody flows. It may be some time before a model is developed which can predict all of these effects. The Baldwin and Lomax¹⁰ model for thin shear layers is commonly used for missile and projectile afterbody predictions. However, it predicts the aforementioned phenomena either poorly or not at all.

The goals of this section are to demonstrate that the effects on turbulence of high Mach number and streamline curvature have a significant impact on the global nature of afterbody flows and to develop models which can predict these effects with reasonable accuracy. This work has been done with the k- ϵ turbulence model, which is presented below.

The k- ϵ Turbulence Model

The k- ϵ model³ consists of partial differential equations for the turbulence kinetic energy k and its dissipation rate ϵ . The turbulent stresses are computed from the eddy viscosity approximation

$$\tau_{ij} = \rho \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \delta_{ij} \frac{2}{3} \rho k \quad (5)$$

in which $\nu_t = c_\mu k^2 / \epsilon$ is the eddy viscosity. The k-equation is derived from the Reynolds-stress transport equations (RSTEs) with an approximation for

diffusive transport and is believed to be without major flaws. It can be written in Cartesian tensor notation as

$$\frac{D\rho k}{Dt} - \frac{\partial}{\partial x_j} \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_j} = P_k - \rho\epsilon \quad (6)$$

The ϵ -equation is

$$\frac{D\rho\epsilon}{Dt} - \frac{\partial}{\partial x_j} \frac{\nu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} = C_1 \frac{\epsilon}{k} P_k - C_2 \frac{\rho\epsilon^2}{k} \quad (7)$$

in which $P_k = \tau_{ij}(\partial U_i/\partial x_j + \partial U_j/\partial x_i)/2$ is the production of turbulence energy, and the model coefficients are $\sigma_k = 1.0$, $\sigma_\epsilon = 1.3$, and $C_1 = 1.45$. In low speed flow $C_\mu = 0.09$, and for uncurved flow $C_2 = 1.92$; these coefficients change in high speed and curved flow.

Mach Number Effects

Mach number effects on turbulence should be significant in high speed afterbody flows. The shear stress and spreading rate of a free shear layer decrease significantly as the difference in Mach number across the shear layer ΔM becomes large. The behavior of this flow was assessed by Bradshaw from data from several sources for the Stanford Conference on Complex Turbulent Flows (Ref. 11, p. 364). The spreading rate $d\delta/dx \approx 0.115$ for incompressible flow and drops to $d\delta/dx \approx 0.03$ for $\Delta M = 5$. For $\Delta M \sim 5$ the spreading rate is roughly 25% of the value which occurs at $\Delta M \sim 0$. Morkovin¹² argued that the change occurred as the velocity of turbulent fluctuations approached the sonic speed, and, thus, the parameter which controls Mach number effects is a Mach number of the turbulent fluctuations $1/2k/a$. Other work has suggested that compressibility effects on large scale turbulence structures may be a dominant mechanism. There is ongoing research in this area.

A model for Mach number effects was derived by Bonnet from theoretical arguments (Ref. 11, p.1408). Bonnet made assumptions to relate fluctuation of velocity and thermodynamic quantities and derived a Mach number correction to the pressure-strain closure model in a Reynolds-Stress transport model (RSTM) which gave good results for the high speed mixing layer.

In the present work, Bonnet's model is reduced to a form compatible with an eddy-viscosity model, such as the k - ϵ model. To do this, it is assumed that the flow is subject to homogeneous shear and that ratio of turbulence energies in the streamwise fluctuations to all fluctuations is 0.48, which is an average of this parameter for uniform shear flow and isotropic turbulence in incompressible flow. With these assumptions and the RSTM coefficients $C_1 = 1.8$ and $C_2 = 0.6$, the compressibility correction used in the present work is derived. It is a modification of C_μ given by

$$C_\mu = C_{\mu 0} (1 + C_{m1} k/a^2) / (1 + C_{m2} k/a^2)^2 \quad (8)$$

in which a is the acoustic speed, $C_{m1} = 8.4$ and $C_{m2} = 6.5$. As with the

standard k-ε model, $C_{\mu_0} = 0.09$. It is noteworthy that the only optimization against experimental data done in deriving Eq. (8) was that performed by Bonnet. Equation (8) has not been optimized specifically for afterbody flows.

Modification for Streamline Curvature

The effects of streamline curvature are also expected to modify significantly turbulence in the afterbody region of many vehicles. Using empirical correlations from Bradshaw¹³ it is possible to estimate that the shear stress may vary by +200%/-100% due to curvature effects in some regions of the missile base flows studied by Petrie and Walker.¹

Curvature effects result from at least two mechanisms. Additional rates of strain, introduced by curvature, alter the magnitude of turbulence production and affect turbulent stresses. This phenomenon is treated accurately if the production term P_k accounts for all strain terms. Curvature effects are also caused by the centrifugal "force" in a curved turbulent shear layer, which has a large effect on the correlation between radial and streamwise velocity fluctuations. It is the centrifugal force that produces Taylor-Goertler vortices in some curved flows and significantly reduces shear stress in others.

The effects of the centrifugal force in curved shear layers can be interpreted in terms of flow stability. The stability parameter of a line vortex was determined by Rayleigh¹⁴ to be $\beta = \partial \Gamma^2 / \partial r$, in which r is the streamline radius of curvature and $\Gamma = 2\pi r V_\theta$ is the circulation at a radial location. If $\beta > 0$ the flow is stabilized and disturbances (turbulence) are suppressed; if $\beta < 0$ the flow is unstable, and disturbances are amplified, and it is neutrally stable for $\beta = 0$. Properly interpreted, this stability parameter is also applicable in curved shear flows.

An alternate form of the stability parameter can be derived by using the Euler n-equation, for the pressure gradient normal to a curved streamline, to eliminate explicit dependence on the radius of curvature.

$$\beta^* = \frac{1}{\rho^2 V^2} \frac{\partial p}{\partial n} \frac{\partial p_t}{\partial n} \quad (9)$$

in which p is an effective pressure (thermodynamic pressure + $2\rho k/3$), $p_t = p + \rho V^2/2$ is the total pressure, V is the velocity magnitude, ρ is the density, and n is normal to streamlines. β^* differs from β by a scaling on radius cubed, but is otherwise similar. Eq. (9) and the slightly different form $\beta^* = \nabla p \cdot \nabla p_t / (\rho V)^2$ have been used in the present work. The latter form has the philosophical and computational advantage that it is independent of mean streamlines. The two forms give similar values of β^* in most regions but differ somewhat at shocks. This stability parameter is appropriate for compressible and incompressible flow. Eq. (9) is used for the results presented here.

A means of modeling curvature effects is needed, and it is important to have a model that mimics physical processes reasonably well. Some evidence suggests that curvature affects the pressure-strain correlation in the RSTEs.

Gillis and Johnston¹⁵ studied incompressible flow over a convex surface. After flowing over a short distance of curved surface, the turbulence energy was reduced moderately, but the shear stress decreased so much that it had the "wrong" sign (corresponding to a negative eddy-viscosity) in the outer part of the boundary layer. The production and "diffusion" mechanisms are not capable of producing this type of change, and the pressure-strain term is the only other viable mechanism. Also, the large change in shear stress accompanied by a significantly smaller change in energy, which could be due to dissipation, is consistent with the properties of the pressure-strain term. In a study of the VTOL upwash fountain by Childs and Nixon,¹⁶ the pressure-strain term was found to be a dominant term in the regions of high curvature and much less significant elsewhere. Hence, it is reasonable to suppose that streamline curvature affects the pressure-strain term and should be modeled accordingly.

Some theoretical guidance for developing a model can be obtained from the RSTEs. By equating the RSTE for shear stress to an equation for the rate of change of shear stress derived from Eq. (5), a dynamical equation for ϵ can be obtained. This new ϵ -equation is similar to Eq. (7); however, it contains a source term which depends on the pressure-strain term. The models used by Launder et al¹⁷ and Leschziner and Rodi¹⁸ also employ source terms in the ϵ -equation, and this approach is favored for the present work.

The model modification employed here is similar to that of Launder et al.¹⁷ For incompressible, thin, two-dimensional shear flows with weak turbulence the two are equivalent. A source term which is a function of the stability parameter is added to the ϵ -equation. The new source term can be incorporated in Eq. (7) by modifying C_2 as follows.

$$C_2 = C_{20} f_c \quad f_c = (1 - C_c \frac{k^2}{\epsilon} \beta^*) \quad (10)$$

in which $C_{20} = 1.92$ is the value of C_2 when streamline curvature is negligible, and C_c is the curvature model coefficient.

Launder selected $C_c = 0.2$ for computing flow over spinning cones. Rodi and Scheuerer¹⁹ tested Launder's model on three shear flows. With $C_c = 0.25$ the model gave good agreement for a stabilized boundary layer flow, slightly overpredicted curvature effects for a stabilized free shear flow, and overpredicted by roughly a factor of two the destabilizing curvature effects for the wall jet on a logarithmic spiral. Bradshaw¹³ has noted that stabilizing and destabilizing curvature affects turbulence quite differently and may need to be modeled differently. Based on the results of Launder et al and Rodi and Scheuerer, the following values for C_c are used in stabilized and destabilized curvature.

$$C_c = \begin{cases} 0.2 & \text{for } \beta > 0 \\ 0.1 & \text{for } \beta < 0 \end{cases} \quad (11)$$

As with the Mach number modification, the model coefficient was optimized by other researchers for simple flows not directly related to afterbody flows.

Arbitrary limits on f_c and on V in the denominator of Eq. (9) are imposed to suppress erratic model behavior at stagnation points and in regions of very high curvature, such as at oblique shocks: $0.1 \leq f_c \leq 2.0$ and $V \geq 0.1V_{ref}$ in the denominator of Equation (9) where the reference velocity is taken to be the freestream velocity. A suitable means of eliminating the need for these limits was sought but was not found. Predicted results can be sensitive to these limits. For example, the ϵ -equation is unstable if $f_c < 0$. Further work is needed in this area.

Modeling Effects in Afterbody Flows

The effects of the above modifications to the $k-\epsilon$ model on high speed afterbody flows are now examined. The results are intended to demonstrate that afterbody flows depend significantly on Mach number and curvature effects on turbulence, and that the proposed modifications to the $k-\epsilon$ model are reasonably accurate for these effects.

Four variations of the $k-\epsilon$ turbulence model are used in this work. They are the standard $k-\epsilon$ model above and that model with modifications for streamline curvature and Mach number effects on turbulence, which are presented below. Herein, they will be referred to by the following notation.

- STD $k-\epsilon$ model defined by Eqs. (5-7)
- M $k-\epsilon$ model with the Mach number modification, Eq. (8)
- C $k-\epsilon$ model with the curvature modification, Eqs. (9-11)
- MC $k-\epsilon$ model with Mach number and curvature modifications

The MC model should give significantly better predictions of high speed afterbody flows than the other models. However, results from all models are examined.

Given in Figure 2 are the base pressures for the $NPR = 2.15$ case of Petrie and Walker¹ computed with the four different models. Results from the STD and MC models are close to the experimental data, and based on these results alone it would be inappropriate to consider one model superior to the other. The M and C models predict the base pressure to be significantly higher and lower than the experimental data, respectively. The changes in base pressure due to predicted Mach number and curvature effects corresponds to variations of +22/-14% in base drag. The relatively good base pressure prediction of the STD model appears to result from fortuitous cancellation of large errors of opposite sign associated with the STD model's inability to predict accurately Mach number and curvature effects.

Figure 3 gives a comparison of the experimental shadowgraph and the Mach number contours predicted with the MC model. In the shadowgraph, the plume shear layer, the outer shear layer, Mach disk, and the recompression are visible, and agreement with the calculation is good. The Mach number contours also show a barrel shock which is not visible in the shadowgraph or velocity measurements. A comparison of measured and computed velocity vectors is presented in Figure 4, and generally good agreement is again seen. The reattachment location (where reverse flow ceases) is nearly correct, and the vortical flow adjacent to the base swirls with the proper sense and approximately the correct magnitude.

There are also some discrepancies between experiment and computation which may be significant. The location of the center of the recirculation is not predicted very well. The recirculation pattern can be critical to base heating rates and the effectiveness of modifications to the base shape for increasing vehicle performance. In the downstream-most line of velocity vectors, the predicted velocities display somewhat greater nonuniformity than does the experimental data. It should be noted that the agreement between computed and measured velocities is much better with the MC model than with the STD model (see Childs and Caruso⁴).

Calculations of the NPR = 6.44 case of Petrie and Walker have also been performed. Figure 5 gives the base pressures computed with the four different models and the trends are qualitatively similar to the NPR = 2.15 case. The difference between the STD and MC models is small, while the results from the M and C models differ significantly from the STD model. In this case the effects of the curvature modification are greater than in the NPR = 2.15 case. Again, the MC model gives a more accurate estimate of base pressure than the STD model. It is worth noting that the difference between the base pressures predicted with the STD and MC models is increasing as the NPR increases. Also, the accuracy of the STD model decreases as the NPR is increased. This suggests that the fortuitous error cancellation which occurs for the STD model in the NPR = 2.15 case will not occur for a wide range of afterbody flows.

A comparison between the computed Mach number contours and the experimental shadowgraph is given in Figure 6, and there appears to be good agreement between the two. The computed and measured velocity vectors are shown in Figure 7 and, while there is generally good agreement, some discrepancies can be seen. The most significant of these may be in the magnitude of velocity and other flow details in the reverse flow region. There are also large differences in the area just downstream of the barrel shock and the Mach disk. The flow in this region is dominated by inviscid phenomena (except for the shock) and the present method should accurately predict the flow in this region. Measurements were made with an LDV, and errors due to particle lag may be the cause of these discrepancies.

The above results suggest that the MC model is reasonably accurate for the turbulence in the types of afterbody flows typified by the experiments of Petrie and Walker. The model predicts the level of pressure with good accuracy and gives the correct variation in pressure as the flow conditions are changed. The mean velocity fields are predicted reasonably well. Those discrepancies with experiment which are noted are probably due to inadequate turbulence modeling, in some cases, and due to uncertainty in the measurements in other cases.

SUMMARY OF ACCOMPLISHMENTS

Numerical Accuracy

- A truncation error analysis method was used to identify sources of numerical errors. This information was used to guide solution adaptive grid generation.
- Divergence of the solution procedure near the nozzle exit at high nozzle pressure ratios was eliminated by aligning grid lines with streamlines in the initial jet shear layer.
- Complex afterbody flow solutions were obtained in which the grid-dependence was negligible for most purposes.

Turbulence Modeling

- Existing modifications to the $k-\epsilon$ model to account for the effects on turbulence of Mach number and streamline curvature were improved upon or adapted to the $k-\epsilon$ model, and they were used in the present work. The calibration of these modifications was done by other researchers in generic flows not related to afterbody flows.
- The effects of Mach number and curvature on turbulence were shown to have significant effects on global afterbody characteristics, such as velocity field and base pressure.
- The $k-\epsilon$ model with modifications for Mach number and curvature effects gave good agreement with a wide range of experimental data in afterbody flows.
- It was demonstrated that the good predictions of base pressure for the cases reported by Petrie and Walker obtained with the standard $k-\epsilon$ model were caused by fortuitous error cancellation within the model. The results suggest that this fortuitous "accuracy" will not, in general, be achieved for other operating conditions or configurations.

RECOMMENDATIONS FOR AN ACCURATE PREDICTION METHOD

From this study it is possible to recommend which elements a flow prediction method should possess if it is to give accurate results for fundamentally correct reasons, that is, that all aspects of the flow prediction are treated with good accuracy.

1. A good grid generation method is needed. It should provide clustering of grid points in the shear layers and it should cause the grid lines to be aligned with streamlines in regions where the solution has gradients. Doing so increases the overall accuracy and can eliminate computational problems at the nozzle exit at high NPRs. However, clustering at shocks is not required, in general, because shocks are captured, not resolved. This is especially true if a good shock capturing algorithm is used.
2. A turbulence model which is reasonably accurate for high speed, curved, and merging shear layers and for the recirculating flow in the base region is needed. These types of flow phenomena cause significant changes in turbulent stresses, in the velocity field, and in the base pressure in some, and probably many, afterbody flows. Models which have not been specifically designed to be accurate in these flows will very likely be inadequate for many needs.
3. It is further recommended that an advanced two-equation turbulence model, such as the $k-\epsilon$ model with modifications for Mach number and streamline curvature effects or the $k-\omega$ model with similar modifications, be used. There is not currently any evidence that higher order models (such as Reynolds-stress models) are needed or will be more accurate than a good two-equation model. However, there is substantial evidence from References 1 and 2 that algebraic models, such as the Baldwin Lomax model, are inadequate for predictive calculations of complex afterbody flows which involve separated or reversed flow.

RECOMMENDED FUTURE RESEARCH

The present work has focused on a few of the dominant issues which affect calculations of simplified missile afterbody flows: grid generation to reduce truncation error and Mach number and curvature effects on turbulence modeling. In addition to the aspects which are addressed, actual missile afterbody flows generally involve combustion, high temperature, low density, particle-laden flow, and operate at angle of attack. The importance of other phenomena to prediction accuracy should be assessed critically.

There were some unresolved issues noted in this research effort which could be corrected in future work. Models for streamline curvature are ill-posed at stagnation points where the mean velocity approaches zero and curvature becomes very large. In these situations physical turbulence can exhibit unusual behavior, but that behavior is always bounded in some sense. Most curvature models, including the present one, may give unbounded behavior. The arbitrary limits used to circumvent this problem in the present work were not based on physics, and an improved means of treating extreme flow curvature is needed.

The present research has also indicated opportunities for future advances. The use of a turbulence model with Mach number and curvature modifications may be appropriate in a range of other flows of interest to the Army, for example, high speed missiles at moderate to high incidence or helicopter rotor wakes. In both of these areas the importance of good turbulence modeling and the inadequacy of some commonly used models have been established. The present work also suggests that a good solution-adaptive grid generation method is needed. Truncation error analysis and alignment between grid lines and streamlines are features not currently employed in existing solution-adaptive grid generation methods. With such a method it should be possible to obtain more accurate solutions at less expense or to address more difficult problems than is currently possible.

PUBLICATIONS

The following conference presentations have resulted from this contract:

1. Childs, R. E. and Caruso, S. C., "On the Accuracy of Turbulent Base Flow Predictions," AIAA-87-1439.
2. Caruso, S. C., and Childs, R. E., "Aspects of Grid Topology for Reynolds-Averaged Navier-Stokes Base Flow Calculations," AIAA-88-0523.
3. Childs, R. E. and Caruso, S. C., "Assessment of Modeling and Discretization Accuracy for High Speed Afterbody Flows," AIAA-89-0531, to be presented at the AIAA 27th Aerospace Sciences Meeting, Reno, January 1989.

The most recent presentation, AIAA-89-0531 will be submitted for publication in a refereed journal, probably the Journal of Propulsion and Power.

TECHNICAL PERSONNEL

Drs. Robert E. Childs, Steven C. Caruso, and David Nixon have worked on this project. No degrees were awarded.

REFERENCES

1. Petrie, H. L. and Walker, B. J., "Comparison of Experiment and Computation for a Missile Base Region Flowfield with a Centered Propulsive Jet," AIAA-85-1618.
2. Putnam, L. E. and Bissinger, N. C., "Results of AGARD Assessment of Prediction Capabilities for Nozzle Afterbody Flows," AIAA-85-1464.
3. Launder, B. E. and Spaulding, D. B., "The Numerical Computation of Turbulent Flows," *Computer Meth. in App. Mech. and Engrg.* 3, 1974, pp. 169-189.
4. Childs, R. E. and Caruso, S. C., "On the Accuracy of Turbulent Base Flow Predictions," AIAA-87-1439.
5. Caruso, S. C., and Childs, R. E., "Aspects of Grid Topology for Reynolds-Averaged Navier-Stokes Base Flow Calculations," AIAA-88-0523.
6. Childs, R. E. and Caruso, S. C., "Assessment of Modeling and Discretization Accuracy for High Speed Afterbody Flows," AIAA-89-0531.
7. Coakley, T. J., "Implicit Upwind Methods for the Compressible Navier-Stokes Equations," *AIAA J.*, Vol. 23, No. 3, 1985, pp. 374-380.
8. Deiwert, G. S., Andrews, A. E., and Nakahashi, K., "Theoretical Analysis of Aircraft Afterbody Flow," *J. Spacecraft*, Vol. 24, No. 6, 1987, pp. 496-503.
9. "Commercial Winged Booster to Launch Satellites from B-52," *Aviation Week & Space Technology*, June 6, 1988, pp. 14-16.
10. Baldwin, B. S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA-73-0257.
11. Kline, S. J. et al., eds., The 1980-81 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows, Stanford University, 1981.
12. Morkovin, M. V., "Effects of Compressibility on Turbulent Flow," presented at Int. Symp. "Mechanique de la Turbulence," Centre Nat. de la Rech. Sci. Paris, 1962, pp. 367-380.
13. Bradshaw, P., "Effects of Streamline Curvature on Turbulent Flow," AGARDograph No. 169, 1973.
14. Rayleigh, Lord, "On the Dynamics of Revolving Fluids," *Proc. Royal Society*, Vol. 93, 1916, p. 193.
15. Gillis, J. C. and Johnston, J. P., "Turbulent Boundary Layer Flow and Structure on a Convex Wall and its Redevelopment on a Flat Wall," *J. Fluid Mech.*, Vol. 135, 1983, pp. 123-153.
16. Childs, R. E. and Nixon, D., "Unsteady Three-Dimensional Simulations of a VTOL Upwash Fountain," AIAA-86-0212.
17. Launder, B. E., Priddin, C. H., and Sharma, B. I., "The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces," *J. Fluids Engrg.*, Mar. 1977, pp. 231-239.
18. Leschziner, M. A. and Rodi, W., "Computation of Strongly Swirling Axisymmetric Free Jets," *AIAA J.*, Vol. 22, No. 12, 1984, pp. 1742-1747.
19. Rodi, W. and Scheuerer, G., "Calculation of Curved Shear Layers with Two-Equation Turbulence models," *Phys. Fluids*, Vol. 26, No. 6, 1983, pp. 1422-1436.

FIGURES

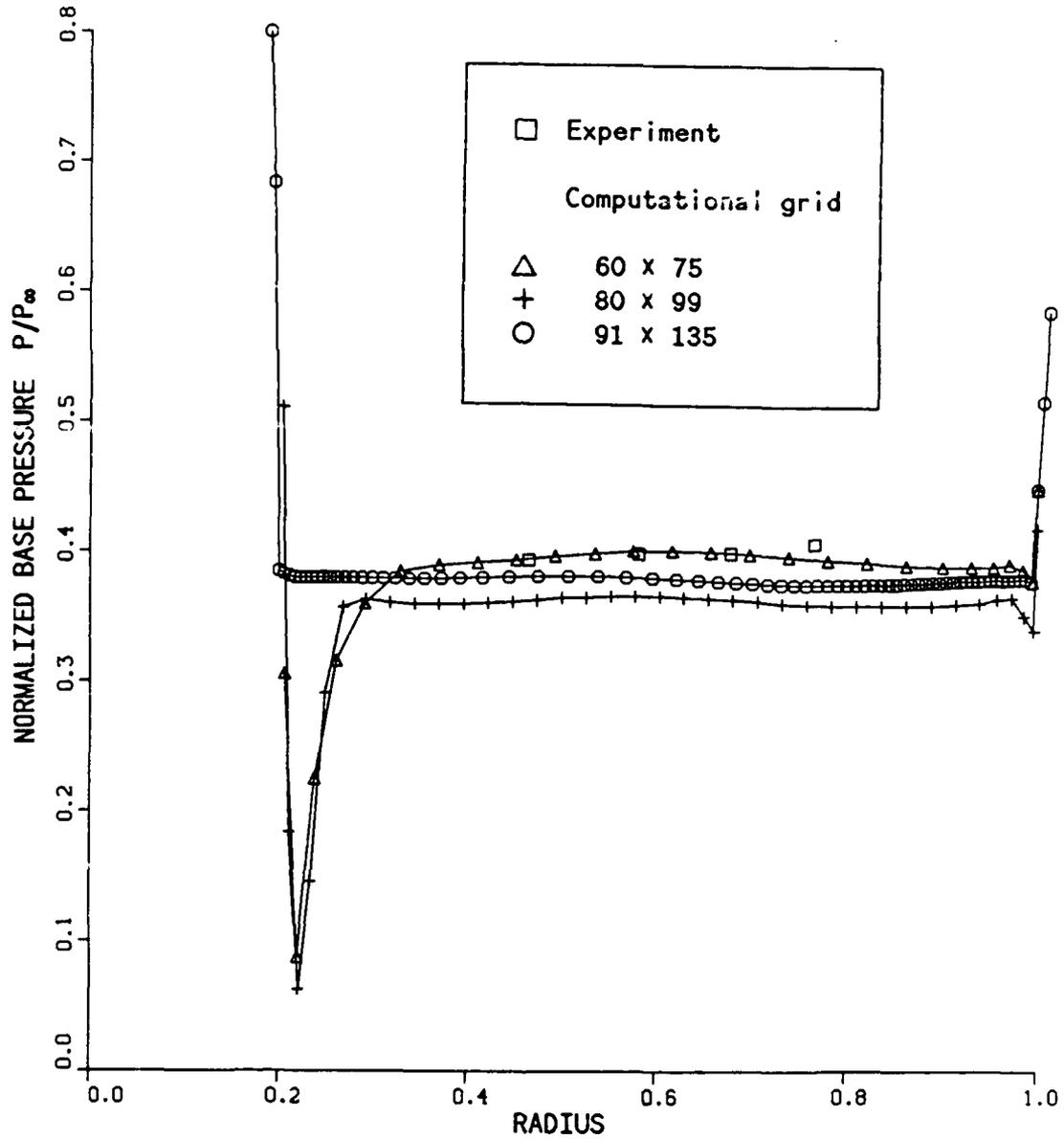


Figure 1. Grid-dependence in the base pressure as the grid is refined and improved by aligning grid lines and streamlines in shear layer.

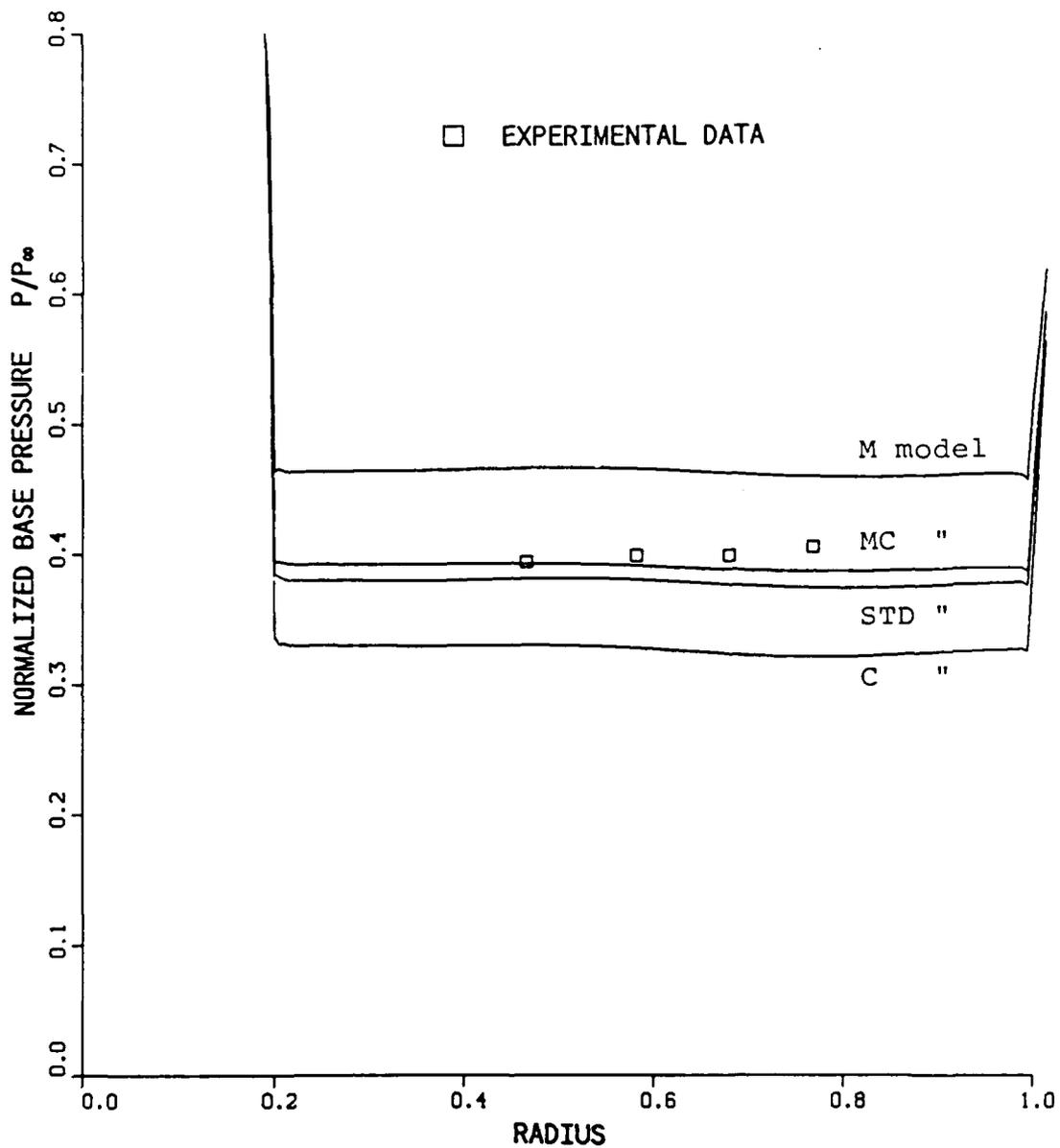


Figure 2. Base pressures for the $NPR = 2.15$ case of Petrie and Walker predicted with the standard $k-\epsilon$ model (STD), and that model with modifications for the effects of streamline curvature (C), Mach number (M), and both curvature and Mach number (MC). Symbols give experimentally measured pressure.

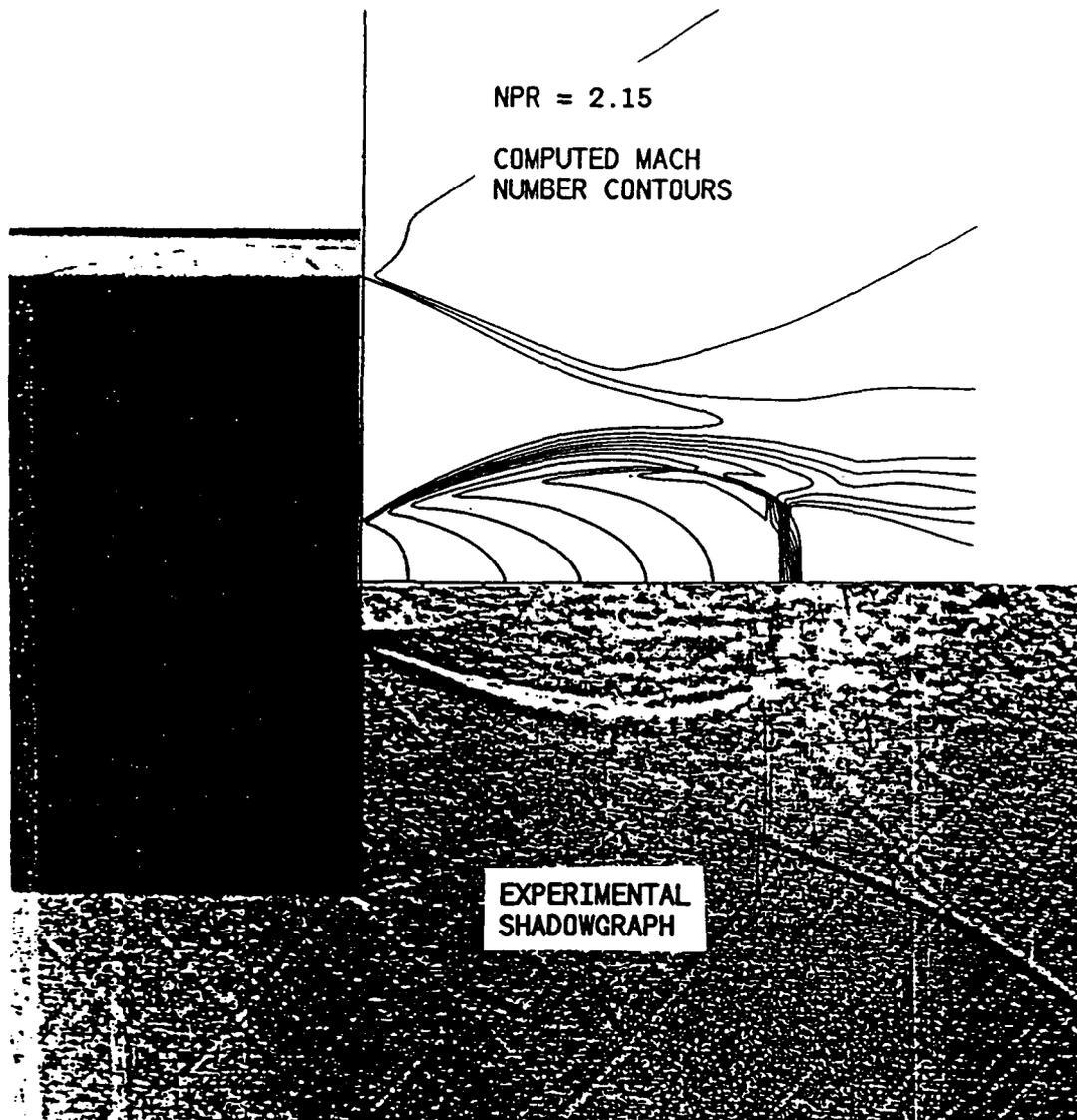


Figure 3. Comparison of computed Mach number contours and experimental shadowgraph for the $NPR = 2.15$ case of Petrie and Walker.

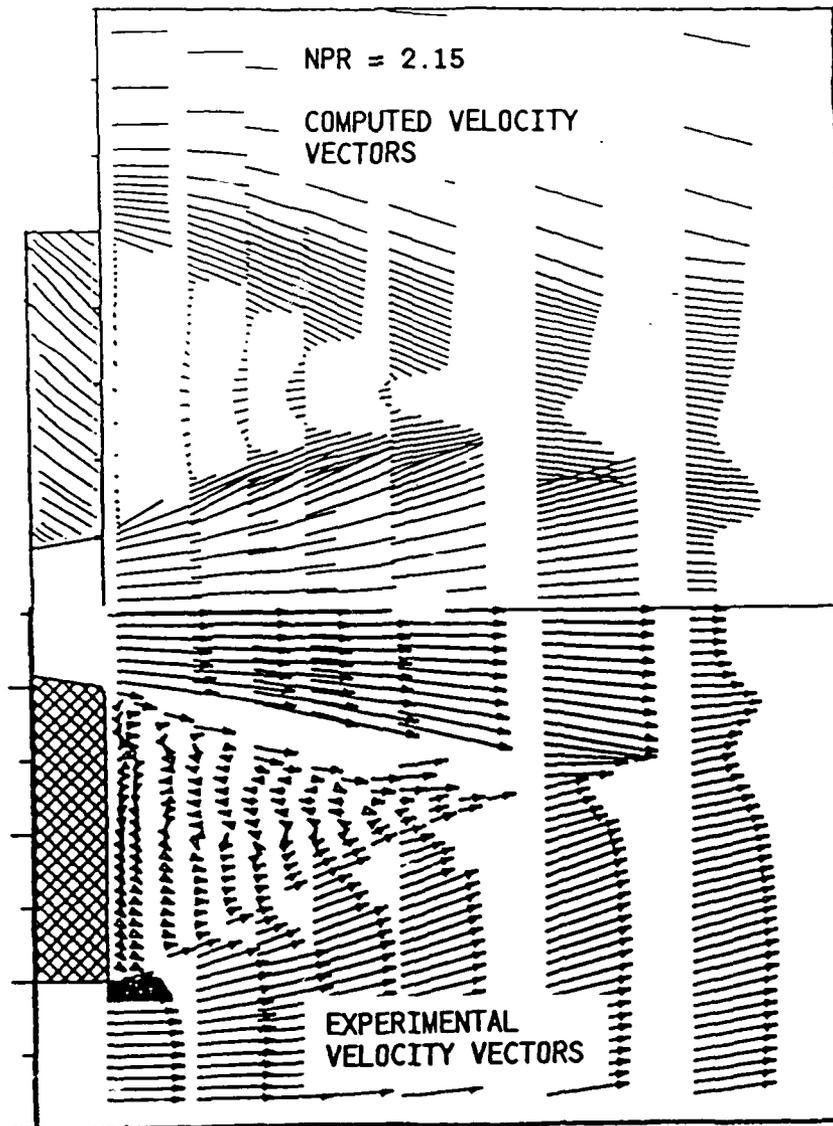


Figure 4. Comparison of computed and experimentally measured velocity vectors for the NPR = 2.15 case of Petrie and Walker.

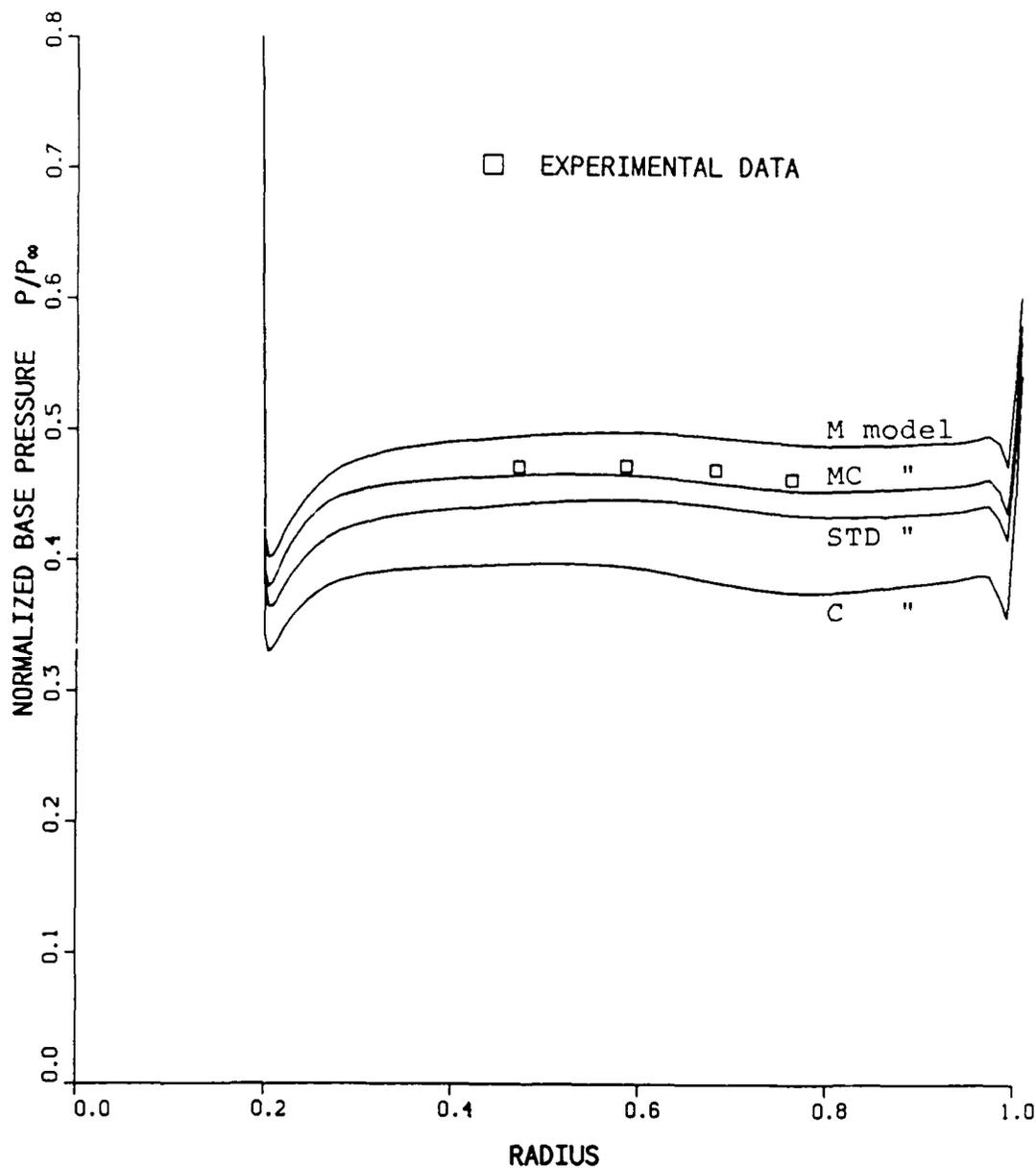


Figure 5. Base pressures for the $NPR = 2.15$ case of Petrie and Walker predicted with the standard $k-\epsilon$ model (STD), and that model with modifications for the effects of streamline curvature (C), Mach number (M), and both curvature and Mach number (MC). Symbols give experimentally measured pressure.

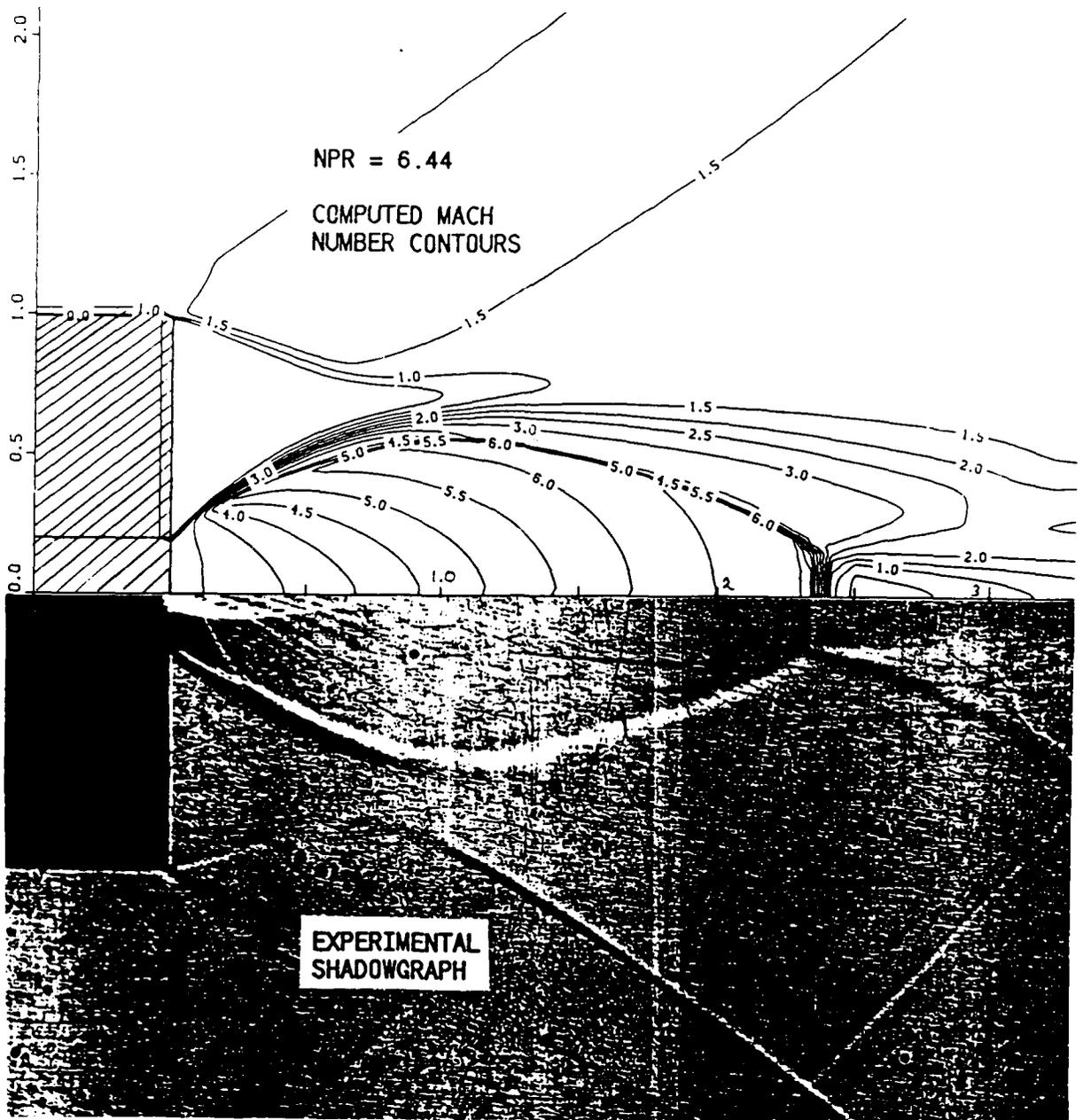


Figure 6. Comparison of computed Mach number contours and experimental shadowgraph for the NPR = 6.44 case of Petrie and Walker.

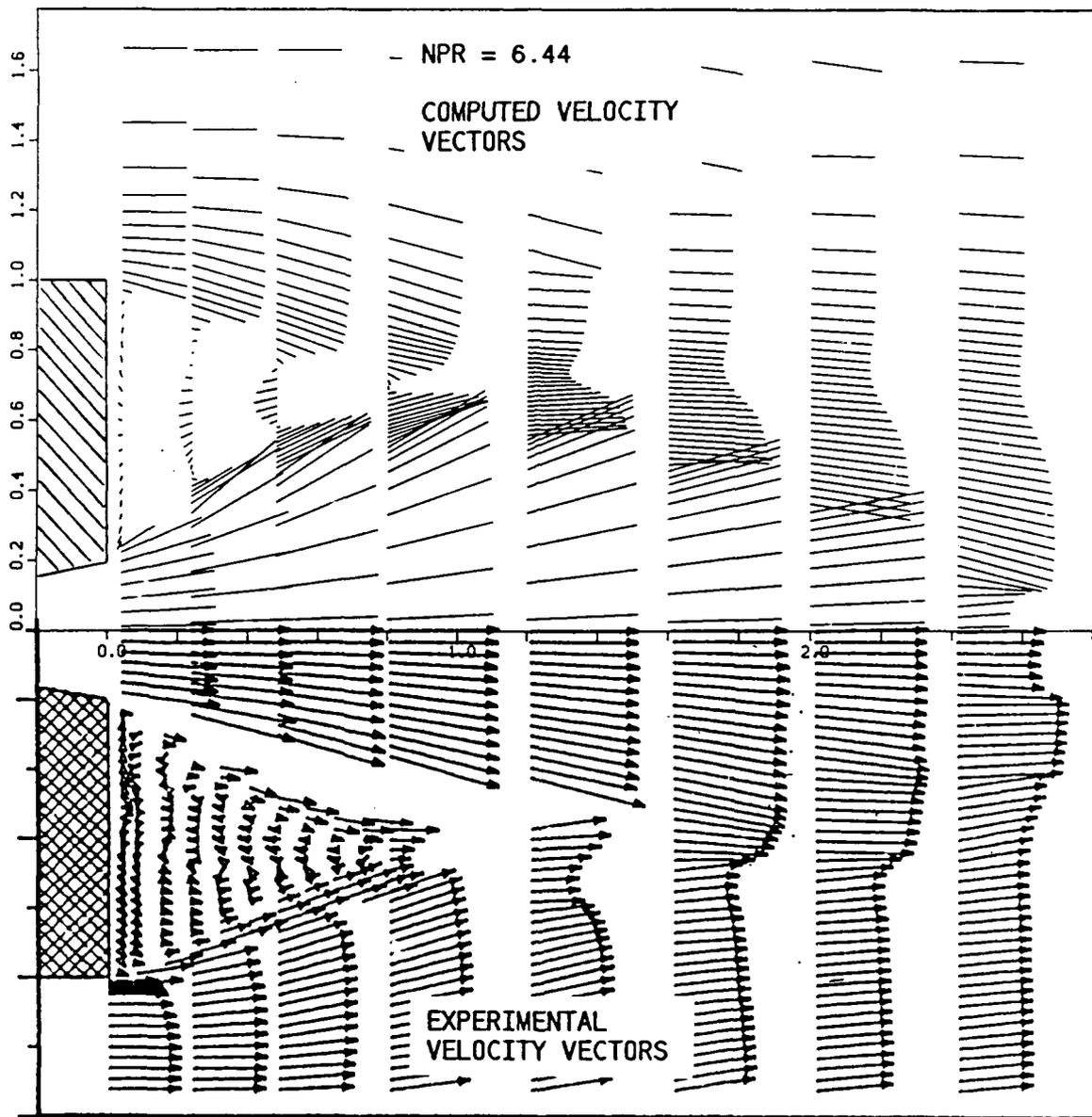


Figure 7. Comparison of computed and experimentally measured velocity vectors for the $NPR = 6.44$ case of Petrie and Walker.