1up

NASA TECHNICAL MEMORANDUM

NASA TM X- 71918

COPY NO. 12.

NASA TM X-71918

THE COMPUTATION OF STEADY NOZZLE FLOW BY A TIME-DEPENDENT METHOD

By Michael C. Cline

April 1974

(NASA-TM-X-71918) THE COMPUTATION OF STEADY NOZZLE FLOW BY A TIME-DEPENDENT METHOD (NASA) ~~20~~ 21 p HC $4.00 CSCL 20D

N74-17013

Unclas

G3/12 29638

This informal documentation medium is used to provide accelerated or special release of technical information to selected users. The contents may not meet NASA formal editing and publication standards, may be revised, or may be incorporated in another publication.

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION
LANGLEY RESEARCH CENTER, HAMPTON, VIRGINIA 23665

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM X-71918		

4. Title and Subtitle	5. Report Date
The Computation of Steady Nozzle Flow By A Time-Dependent Method	April 1974
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
Michael C. Cline	
9. Performing Organization Name and Address	10. Work Unit No.
NASA Langley Research Center Hampton, VA 23665	501-24-06-01
	11. Contract or Grant No.
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Technical Memorandum
National Aeronautics and Space Administration Washington, D.C., 20546 &/ U.S. Atomic Energy Commission, Los Alamos, NM 87544	14. Sponsoring Agency Code

15. Supplementary Notes

Special technical information release, not planned for formal NASA publication

16. Abstract

The steady flow in two-dimensional and axisymmetric nozzles was computed using a time-dependent method. In this method the interior mesh points were computed using the MacCormack finite-difference scheme, while a characteristic scheme was used to calculate the boundary mesh points. No explicit artificial viscosity term was included. The fluid was assumed to be a perfect gas. This method was used to compute the flow in a 45° - 15° conical, converging-diverging nozzle, a 15° conical, converging nozzle, and a 10° conical, plug nozzle. Good agreement between the numerical solution and experimental data was found. In contrast to previous time-dependent methods, the computational times were less than one minute on a CDC 6600 computer.

17. Key Words (Suggested by Author(s)) (STAR category underlined)	18. Distribution Statement
<u>Aerodynamics</u> Subsonic Flow — Nozzle Flow Transonic Flow — Channel Flow Supersonic Flow Hypersonic Flow	Unclassified - Unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	~~19~~ 21	NTIS 4.00

*Available from { The National Technical Information Service, Springfield, Virginia 22151
STIF/NASA Scientific and Technical Information Facility, P.O. Box 33, College Park, MD 20740

THE COMPUTATION OF STEADY NOZZLE FLOW

BY A TIME-DEPENDENT METHOD*

Michael C. Cline**
University of California
Los Alamos Scientific Laboratory
Los Alamos, New Mexico 87544

Abstract

The steady flow in two-dimensional and axisymmetric nozzles was computed using a time-dependent method. In this method the interior mesh points were computed using the MacCormack finite-difference scheme, while a characteristic scheme was used to calculate the boundary mesh points. No explicit artificial viscosity term was included. The fluid was assumed to be a perfect gas. This method was used to compute the flow in a 45° - 15° conical, converging-diverging nozzle, a 15° conical, converging nozzle, and a 10° conical, plug nozzle. Good agreement between the numerical solution and experimental data was found. In contrast to previous time-dependent methods, the computational times were less than one minute on a CDC 6600 computer.

---

*The initial phase of this research was accomplished while the author held a National Research Council Postdoctoral Resident Research Associateship supported by the NASA Langley Research Center, Hampton, Virginia. The final phase was performed under the auspices of the United States Atomic Energy Commission.

Index categories: Subsonic and Transonic Flow; Supersonic and Hypersonic Flow; Nozzle and Channel Flow.

**Group T-3, Member AIAA

Introduction

The equations of motion governing steady, inviscid flow are of a mixed type, that is, hyperbolic in the supersonic region and elliptic in the subsonic region. These mathematical difficulties may be removed by using the so called time-dependent method. In this technique the flow is assumed to be unsteady or time-dependent. The governing equations are, therefore, hyperbolic in both the subsonic and supersonic regions. The steady state solution may be obtained as the asymptotic solution for large time. This technique has been used to compute converging-diverging nozzle flows by Prozan (as reported by Saunders[1] and Cuffel, et al.[2]), Migdal, et al.[3], Wehofer and Moger[4], Laval[5], and Serra[6]. This technique has also been used to compute converging nozzle flows by Wehofer and Moger[4] and Brown and Ozcan[7]. While the results of the above calculations are for the most part good, the computational times are rather large. In addition, although the computer program of Ref. 6 included a centerbody and those of Refs. 4 and 7 included the exhaust jet, none of the above codes is able to calculate both, that is, plug nozzles. Therefore, the object of this research was to develop a production type computer program capable of solving converging, converging-diverging, and plug two-dimensional nozzle flows in computational times of one minute or less on a CDC 6600 computer.

Literature Review

The following is a discussion of the methods used in Refs. 1 through 7. The first paragraph deals with the computation of the interior mesh points while the next three paragraphs are concerned with the boundary mesh points.

Prozan[1], Wehofer and Moger, and Laval used variations of the two-step Lax-Wendroff scheme to compute the interior mesh points. Migdal, et al., and Brown and Ozcan employed the original one-step Lax-Wendroff scheme, but with the equations of motion in non-conservation form. Serra applied the original Lax-Wendroff scheme with the equations of motion in conservation form. In order to stabilize their schemes, Laval and Serra used artificial viscosity terms in their difference equations. Wehofer and Moger reset the stagnation conditions along each streamline, reset the mass flow at each axial location, and smoothed the subsonic portion of the flow after each time step.

In order to compute the nozzle inlet mesh points, Prozan[1] assumed the inlet flow to be uniform. Wehofer and Moger assumed only that the pressure was radially uniform at the inlet. Migdal, et al., and Brown and Ozcan mapped the inlet to minus infinity after Moretti[8], thus allowing the static conditions to be set equal to the stagnation conditions. Laval used extrapolation of the interior mesh points to determine the inlet mesh points, while Serra employed a characteristic scheme.

Prozan,[1] Wehofer and Moger, Laval, and Brown and Ozcan used an extrapolation technique to compute the wall mesh points. Migdal, et al., employed a characteristic scheme after Moretti[8] to compute the wall mesh points, while Serra applied a reflection technique. In order for the converging nozzle problem to be properly posed, an exhaust jet calculation must be included. Wehofer and Moger used an extrapolation procedure to compute the exhaust jet boundary mesh points, while Brown and Ozcan employed a characteristic scheme after Moretti[8].

All of the above authors used extrapolation to compute the exit mesh points when the flow was supersonic, since any errors incurred would be swept out of the mesh. Serra employed a characteristic scheme when the exit flow was subsonic.

Choice of Method

The lengthy computational times associated with time-dependent calculations are usually due to either inefficient numerical schemes or poor treatment of the boundaries resulting in the requirement for excessively fine computational meshes. In the following paragraphs, a technique for the much more efficient calculation of the interior and boundary mesh points will be discussed.

The computation of steady flows by a time-dependent method differs from ordinary initial-value problems in that the initial data and much of the transient solution have a negligible effect on the final or steady solution. Therefore, accuracy is important only for the asymptotic state, and special attention can be given to intermediate efficiency in order that computational times are made reasonable. For this reason, the interior mesh points can be computed using a very efficient finite-difference scheme, as opposed to those less efficient finite-difference or characteristic schemes that achieve high accuracy at every step.

In the class of finite-difference schemes, the two-step methods such as the MacCormack[10] and the two-step Lax-Wendroff schemes[11] are more efficient than the original Lax-Wendroff scheme[11], especially if the governing equations are in conservation form. Moretti[12] showed that using the equations of motion in conservation form deceased efficiency and ease of programming while only slightly increasing the accuracy of shock calculations.

The use of an explicit artificial viscosity term also decreases efficiency and was shown to be physically unjustified by Moretti[12]. In addition, such increases in the numerical dissipation can often destroy the weak shock structure of transonic flows. Therefore, the MacCormack scheme with the equations of motion in non-conservation form is used to calculate the interior mesh points. No explicit artificial viscosity term was included, although an implicit dissipation is present as an effect of truncation terms, which is sufficient to assure numerical stability for the present flows.

The boundary mesh points, while making up only a small part of the total mesh points, are the most important to be treated accurately[8], because of the flow-field sensitivity to precise boundary geometry. Moretti[8] and Abbett[9] showed that reflection, extrapolation, and one-sided difference techniques for computing solid wall boundaries give poor results and should be avoided. Therefore, the wall and centerbody mesh points are computed using a characteristic scheme. Likewise, the exhaust jet boundary mesh points are also calculated employing a characteristic scheme.

In the case of the nozzle inlet mesh points, the use of extrapolation techniques and the assumption of one-dimensional flow presume the form of the solution and in many cases are physically unjustified. On the other hand, a characteristic scheme could be used to calculate the inlet mesh points. While the stagnation pressure and temperature are assumed to remain constant at the inlet in a characteristic scheme, which is not necessarily the case for unsteady flow, this assumption would appear to be valid for the time-dependent calculation of steady flows. However, Moretti[8] recommends mapping the inlet to minus infinity, thus allowing the static conditions to be set equal to the stagnation conditions. In theory, this would appear to be the best approach. However, it should be kept in mind that the infinite physical plane must be replaced by a finite computational plane. Also, this technique requires additional mesh points upstream of the nozzle inlet. It is not presently resolved as to whether the characteristic scheme approach used by Serra or the mapping to minus infinity approach suggested by Moretti[8] and employed by Migdal, et al., and Brown and Ozcan is the best technique. To reduce the total number of mesh points to be computed, a charactertistic scheme is used to compute the inlet mesh points.

Extrapolation is used to compute the exit mesh points when the flow is supersonic and a characteristic scheme is employed when the flow is subsonic.

Equations of Motion

The appropriate non-conservation form of equations for two-dimensional, inviscid, isentropic, rotational flow are

$$\rho_t + u\rho_x + v\rho_y + \rho u_x + \rho v_y + \varepsilon \rho v/y = 0 \tag{1}$$

$$u_t + uu_x + vu_y + p_x/\rho = 0 \tag{2}$$

$$v_t + uv_x + vv_y + p_y/\rho = 0 \tag{3}$$

$$p_t + up_x + vp_y - a^2(\rho_t + u\rho_x + v\rho_y) = 0 \tag{4}$$

where ρ is the density, u is the axial velocity, v is the radial velocity, p is the pressure, a is the local speed of sound, t is the time, x and y are the axial and radial coordinates, and the subscripts denote partial differentiation. The symbol ε is 0 for planar flow and 1 for axisymmetric flow.

The physical (x, y) plane is mapped into a rectangular computational plane (ζ, η) by the following coordinate transformation;

$$\zeta = x \; ; \qquad \eta = \frac{y - y_c(x)}{y_w(x,t) - y_c(x)} \; ; \qquad \tau = t \tag{5}$$

where $y_w(x, t)$ denotes the nozzle wall and exhaust jet boundary radius as a function of x and t and $y_c(x)$ denotes the nozzle centerbody radius as a function of x. In the (ζ, η, τ) coordinate system Eqs. (1) through (4) become

$$\rho_\tau + u\rho_\zeta + \bar{v}\rho_\eta + \rho u_\zeta + \rho\alpha u_\eta + \rho\beta v_\eta + \varepsilon\rho v\beta/(y_c + \eta/\beta) = 0 \tag{6}$$

$$u_\tau + uu_\zeta + \bar{v}u_\eta + p_\zeta/\rho + \alpha p_\eta/\rho = 0 \tag{7}$$

$$v_\tau + uv_\zeta + \bar{v}v_\eta + \beta p_\eta/\rho = 0 \tag{8}$$

$$p_\tau + up_\zeta + \bar{v}p_\eta - a^2(\rho_\tau + u\rho_\zeta + \bar{v}\rho_\eta) = 0 \tag{9}$$

where

$$\beta = \frac{1}{y_w - y_c} \; ; \; \alpha = -\beta\frac{\partial y_c}{\partial x} - \eta\beta\left(\frac{\partial y_w}{\partial x} - \frac{\partial y_c}{\partial x}\right) \; ; \; \delta = -\eta\beta\frac{\partial y_w}{\partial t} \tag{10}$$

$$\bar{v} = \alpha u + \beta v + \delta \tag{11}$$

The fluid is assumed to be thermally and calorically perfect; that is, a constant ratio of specific heats.

Numerical Method

The computational plane is divided up into five sets of mesh points. These five sets are the interior, inlet, exit, wall and centerbody, and exhaust jet boundary mesh points. The treatment of each of these sets is given below.

Interior Mesh Points

The interior mesh points are computed using the MacCormack scheme. This scheme is a second-order, noncentered, two-step, finite-difference scheme. Backward differences are used on the first step while forward differences are employed on the second step. The governing equations are left in non-conservation form. No explicit artificial viscosity term is used. Centerline mesh points are computed enforcing symmetry of the flow. A complete description of the method is given in Ref. 10.

Inlet Mesh Points

The inlet mesh points are computed using a second-order, reference-plane characteristic scheme. In this scheme the partial derivatives with respect to η are computed in the initial-value and solution surfaces using noncentered differencing as in the MacCormack scheme. These approximations to the derivatives with respect to η are then treated as forcing terms and the resulting system of equations is solved in the η = constant reference planes using a two-independent variable, characteristic scheme. The scheme consists of a first step using backward differences in the initial-value plane and a second and final step using an average of the backward η differences in the initial-value plane and forward η differences in the solution plane. The boundary condition is the specification of the stagnation temperature and stagnation pressure. The use of a reference-plane characteristic scheme requires an additional boundary condition. This additional boundary condition is the specification of the inlet flow angle, since in general the inlet flow angle can be approximately determined from the nozzle geometry. These three quantities, along with the one Mach-line compatibility equation, are sufficient to determine the four dependent variables, u, v, p, and ρ.

Exit Mesh Points

Subsonic Flow. For subsonic flow a reference-plane characteristic scheme similar to the inlet scheme is used. The exit pressure is specified. This pressure, one Mach-line compatibility equation, and two streamline compatibility equations are sufficient to determine the four dependent variables.

Supersonic Flow. For supersonic flow the exit mesh points are computed using linear extrapolation. Since any errors incurred here will be swept out of the mesh, such a procedure should be sufficient.

Wall and Centerbody Mesh Points

The wall and centerbody mesh points are also computed using a reference-plane characteristic scheme. In this scheme the derivatives with respect to ζ are approximated and the resulting system of equations is solved in the ζ = constant reference planes. The wall and centerbody contours, and therefore, their slopes are specified. The tangency condition given by

$$v = u \tan \theta + \partial y_w / \partial t \quad , \qquad (12)$$

where θ is the local wall or centerbody angle, one Mach-line compatibility equation, and two streamline compatibility equations are sufficient to determine the dependent variables.

Exhaust Jet Boundary Mesh Points

The exhaust jet boundary mesh points are computed by the wall routine such that the pressure boundary condition

$$p = p_{ambient} \tag{13}$$

is satisfied. This is accomplished by first assuming the shape of the jet boundary and then using the wall routine to calculate the pressure. Next, the jet boundary location is slightly changed and a second pressure is computed. By use of an interpolation-extrapolation procedure, a new jet boundary location is determined. This interpolation-extrapolation procedure is then repeated at each point until the jet boundary pressure and the ambient pressure agree to within some specified tolerance.

When an exhaust jet calculation is made, the nozzle wall exit lip mesh point becomes a singularity and, therefore, is treated by a special procedure. This procedure consists of first computing an upstream solution at the exit mesh point using the flow tangency condition as the boundary condition and backward ζ differences in both the initial-value and solution planes. Next, a downstream solution is calculated using Eq. (13) as the boundary condition and the total conditions calculated from the upstream mesh point. The upstream solution is used when computing wall mesh points upstream of the exit mesh point, while the downstream solution is used when computing downstream wall mesh points. A third exit mesh point solution to be used for interior mesh point calculation is determined as follows. When the upstream solution is subsonic, the two solution Mach numbers are averaged such that the averaged Mach number is less than or equal to one. This Mach number is then used to calculate the exit mesh point solution to be used to compute the interior mesh points. When the upstream solution is supersonic, the upstream solution is used to calculate the interior mesh points.

Stability

The step size, Δt, is controlled by the well known Courant condition which can be expressed as

$$\Delta t \leq 1/[(V+a)(1/\Delta x^2 + 1/\Delta y^2)^{1/2}] \tag{14}$$

where V is the velocity magnitude. Using Eqs. (5) and (10), Eq. (14) can be written as

$$\Delta \tau \leq A/[(V+a)(1/\Delta \zeta^2 + \beta^2/\Delta \eta^2)^{1/2}] \tag{15}$$

where the coefficient A was determined from actual calculations and varied between 1.0 and 1.6 depending on the geometry of the flow in question.

Overall Program

The nozzle inlet flow as well as the flow leaving the nozzle may be either subsonic or supersonic. The flow may contain stream-wise variations in stagnation temperature and stagnation pressure. The nozzle wall and centerbody geometries may be either one of three analytical contours or a completely general tabular contour. The program is capable of calculating the exhaust jet boundary for either subsonic or supersonic flow. The initial data may be either read in or calculated internally by the program. The internally computed data are calculated assuming one-dimensional, steady, isentropic flow with area change. The program output includes the coordinates, velocities, pressure, density, Mach number, temperature, mass flow, and axial thrust in both English and metric units.

Results and Discussion

The results in the present study were obtained using a CDC 6600 computer. The computational times given are the central processor time not including compilation. In order to compare these results with those of other investigators, the following table of relative machine speeds is given.

Computer	Relative Machine Speed
IBM 7094	0.1
IBM 360/50	0.1
IBM 360/65	0.3
IBM 360/75	0.5
Univac 1108	0.5
CDC 6600	1.0

These relative speeds were obtained from Refs. 13 and 14 and are only rough estimates. These values may vary considerably depending on the compiler, machine configuration, and numerical technique. The initial data in each case were the one-dimensional values computed internally by the program. When the relative change in axial velocity in the throat and downstream regions was less than a prescribed convergence tolerance, the flow was assumed to have reached steady state. The convergence tolerance was found to be a function of the mesh spacing, flow speed, and nozzle geometry. For the results presented here a convergence tolerance of 0.003 per cent for flows without exhaust jet calculations and 0.005 per cent for flows with exhaust jet calculations was employed.

The present method was used to compute the steady state solution for flow in the 45° - 15° conical, converging-diverging nozzle shown in Fig. 1a. The Mach number contours and wall pressure ratio are shown in Fig. 2.

The experimental data are those of Cuffel, et al. The computed discharge coefficient is 0.983 as compared with the experimental value of 0.985. A 21 x 8 computational mesh was used, which required 301 time planes and a computational time of 35 seconds. In general, there is good agreement with the experimental data. This case was also solved by Prozan[2], Migdal, et al., Laval, and Serra. While the details of Prozan's computation were not reported by Cuffel, et al., Saunders reported a time of 45 minutes on a CDC 3200 (23 x 11 mesh) for computing the flow in a nozzle with a large radius of curvature. Migdal, et al., reported a computational time of less than 5 minutes on an IBM 360/75. Laval reported a computational time on the order of 2 hours on an IBM 360/50 (61 x 21 mesh). Serra reported a computational time of 80 minutes on a Univac 1108 (3000 mesh points). In addition, this case was also solved by Prozan and Kooker[15] using a relaxation scheme to solve the irrotational equations of motion. They reported a computational time of 5 to 10 minutes on an IBM 7094 (21 x 11 mesh).

The present method was also used to compute the steady state flow in a 15° conical, converging nozzle. The nozzle geometry is shown in Fig. 1b. The Mach number contours and wall pressure ratio for a nozzle pressure ratio of 2.0 are shown in Fig. 3. The experimental data are those of Thornock[16]. The computed discharge coefficient is 0.957 as compared with the experimental value of 0.960. A 23 x 7 computational mesh was used, which required 249 time planes and a computational time of 29 seconds. In general, there is good agreement with the experimental data. This case was also solved by Wehofer and Moger and Brown and Ozcan. Wehofer and Moger's solution for a pressure ratio of 2 required over 2 hours on an IBM 360/50 (47 x 11 mesh), while Brown and Ozcan's results required 17 minutes on an IBM 360/65 (20 x 6 mesh).

Finally, the present method was used to calculate the flow in a 10° conical, plug nozzle. The nozzle geometry is shown in Fig. 1c. The Mach number contours and plug pressure ratio for a nozzle pressure ratio of 3.29 are shown in Fig. 4. The experimental data are those of Bresnahan and Johns[17]. A 31 x 6 computational mesh was used, which required 327 time planes and a computational time of 52 seconds. In general, there is good agreement with the experimental data. The author is unaware of any other time-dependent analysis of plug nozzles.

Concluding Remarks

A method of computing nozzle flows has been presented. A production type computer program capable of solving a wide variety of nozzle flows has been developed. The accuracy of this program was demonstrated by computing the flow in a 45° - 15° conical, converging-diverging nozzle, a 15° conical, converging nozzle, and a 10° conical, plug nozzle. The computational times were less than one minute on a CDC 6600 computer, which is considerably faster than any of the previous time-dependent techniques.

References

1. Saunders, L. M., "Numerical Solution of the Flow Field in the Throat Region of a Nozzle," BSVD-P-66-TN-001 (NASA CR 82601), Brown Engineering Co., August 1966.

2. Cuffel, R. F., Back, L. H., and Massier, P. F., "Transonic Flow-Field in a Supersonic Nozzle with Small Throat Radius of Curvature," AIAA Journal, Vol. 7, No. 7, July 1969, pp. 1364-1366.

3. Migdal, D., Klein, K., and Moretti, G., "Time-Dependent Calculations for Transonic Nozzle Flow," AIAA Journal, Vol. 7, No. 2, Feb. 1969, pp. 372-374.

4. Wehofer, S. and Moger, W. C., "Transonic Flow in Conical Convergent and Convergent-Divergent Nozzles with Nonuniform Inlet Conditions," AIAA Paper No. 70-635, AIAA 6th Propulsion Joint Specialist Conference, June 1970.

5. Laval, P., "Time-Dependent Calculation Method for Transonic Nozzle Flows," Lecture Notes in Physics, Vol. 8, Jan. 1971, pp. 187-192.

6. Serra, R. A., "The Determination of Internal Gas Flows by a Transient Numerical Technique," AIAA Paper No. 71-45, AIAA 9th Aerospace Sciences Meeting, Jan. 1971.

7. Brown, E. F. and Ozcan, H. M., "A Time-Dependent Solution of Mixed Flow Through Convergent Nozzles," AIAA Paper No. 72-680, AIAA 5th Fluid and Plasma Dynamics Conference, June 1972.

8. Moretti, G., "Importance of Boundary Conditions in the Numerical Treatment of Hyperbolic Equations," Physics of Fluids, Supplement II, 1969, pp. 13-20.

9. Abbett, M. J., "Boundary Condition Calculation Procedures for Inviscid Supersonic Flow Fields," AIAA Computational Fluid Dynamics Conference, July 1973.

10. MacCormack, R. W., "The Effect of Viscosity in Hypervelocity Impact Cratering," AIAA Paper No. 69-354, AIAA Hypervelocity Impact Conference, April 1969.

11. Richtmyer, R. D. and Morton, K. W., "Difference Methods for Initial-Value Problems," Interscience Publishers, 1967.

12. Moretti, Gino, "A Critical Analysis of Numerical Techniques: The Piston-Driven Inviscid Flow," PIBAL Report No. 69-25, Polytechnic Institute of Brooklyn, July 1969.

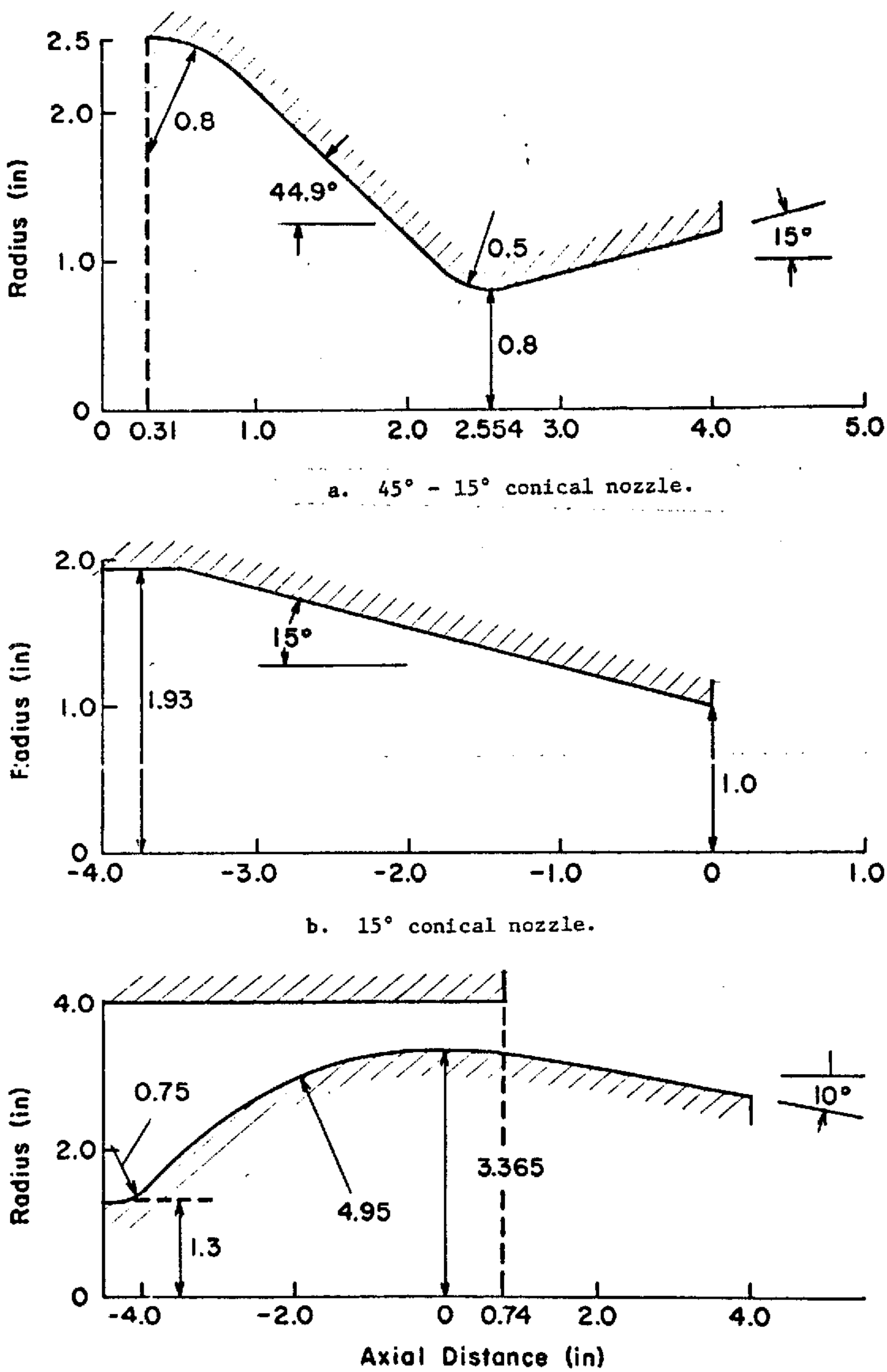
13. Worlton, J., Office Memorandum from the AEC Computer Information Meeting held at the Los Alamos Scientific Laboratory, Los Alamos, New Mexico, May 20-21, 1968.

14. Fernbach, S., private communication to Jeremiah Kratz, Lawrence Radiation Laboratory, Livermore, California, April 30, 1971.

15. Prozan, R. J. and Kooker, D. E., "The Error Minimization Technique with Application to a Transonic Nozzle Solution," Journal of Fluid Mechanics, Vol. 43, Part 2, August 1970, pp. 269-277.

16. Thornock, R. L., "Experimental Investigation of the Flow Through Convergent-Conical Nozzles," Document No. D6-20375, The Boeing Co., September 1968.

17. Bresnahan, D. L. and Johns, A. L., "Cold Flow Investigation of a Low Angle Turbojet Plug Nozzle with Fixed Throat and Translating Shroud at Mach numbers from 0 to 2.0," NASA TM X-1619, August 1968.



a. 45° - 15° conical nozzle.

b. 15° conical nozzle.

c. 10° conical, plug nozzle.

Figure 1. Nozzle geometries

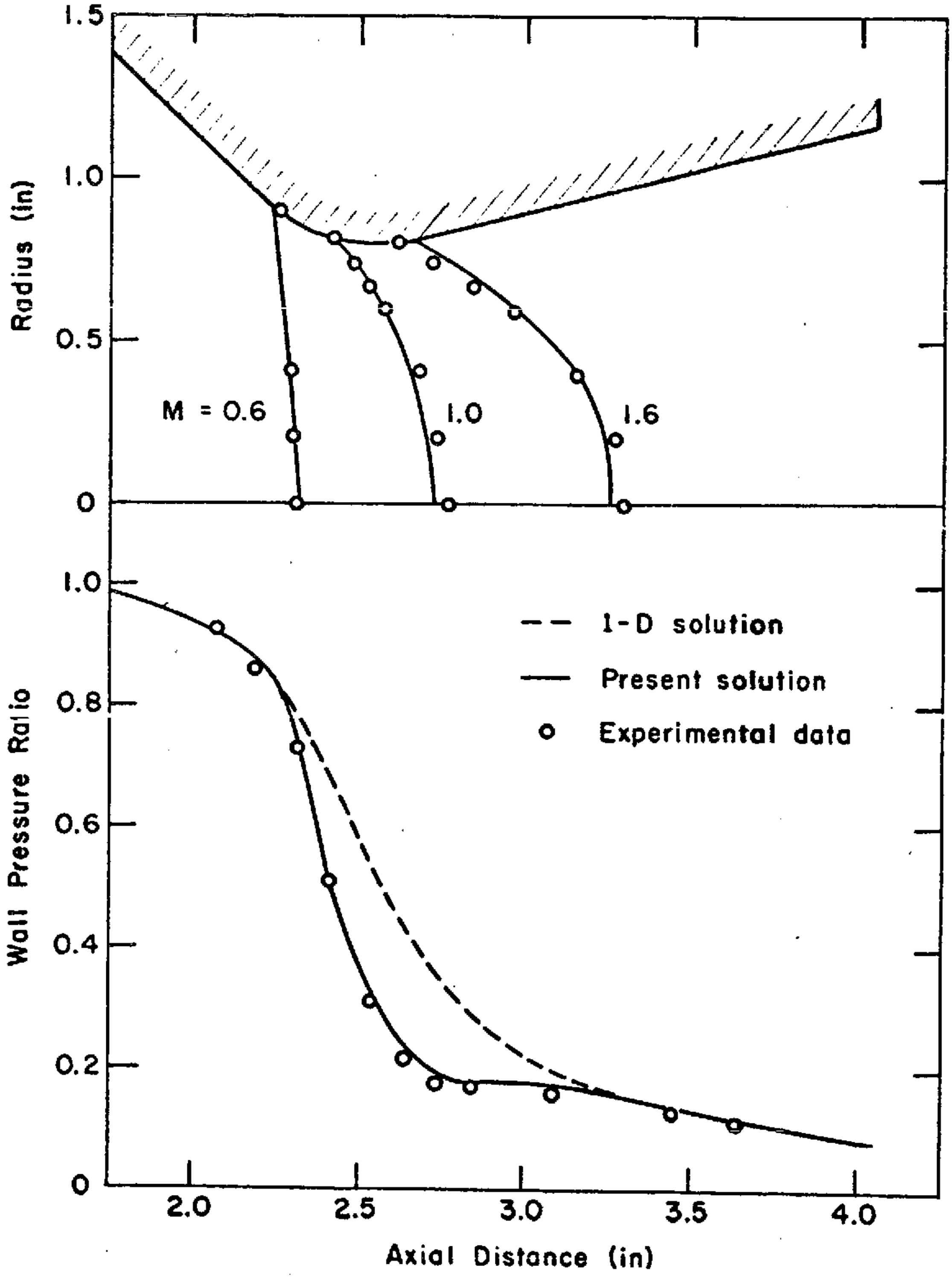


Figure 2. Mach number contours (above) and wall pressure ratio for 45°- 15° conical nozzle.

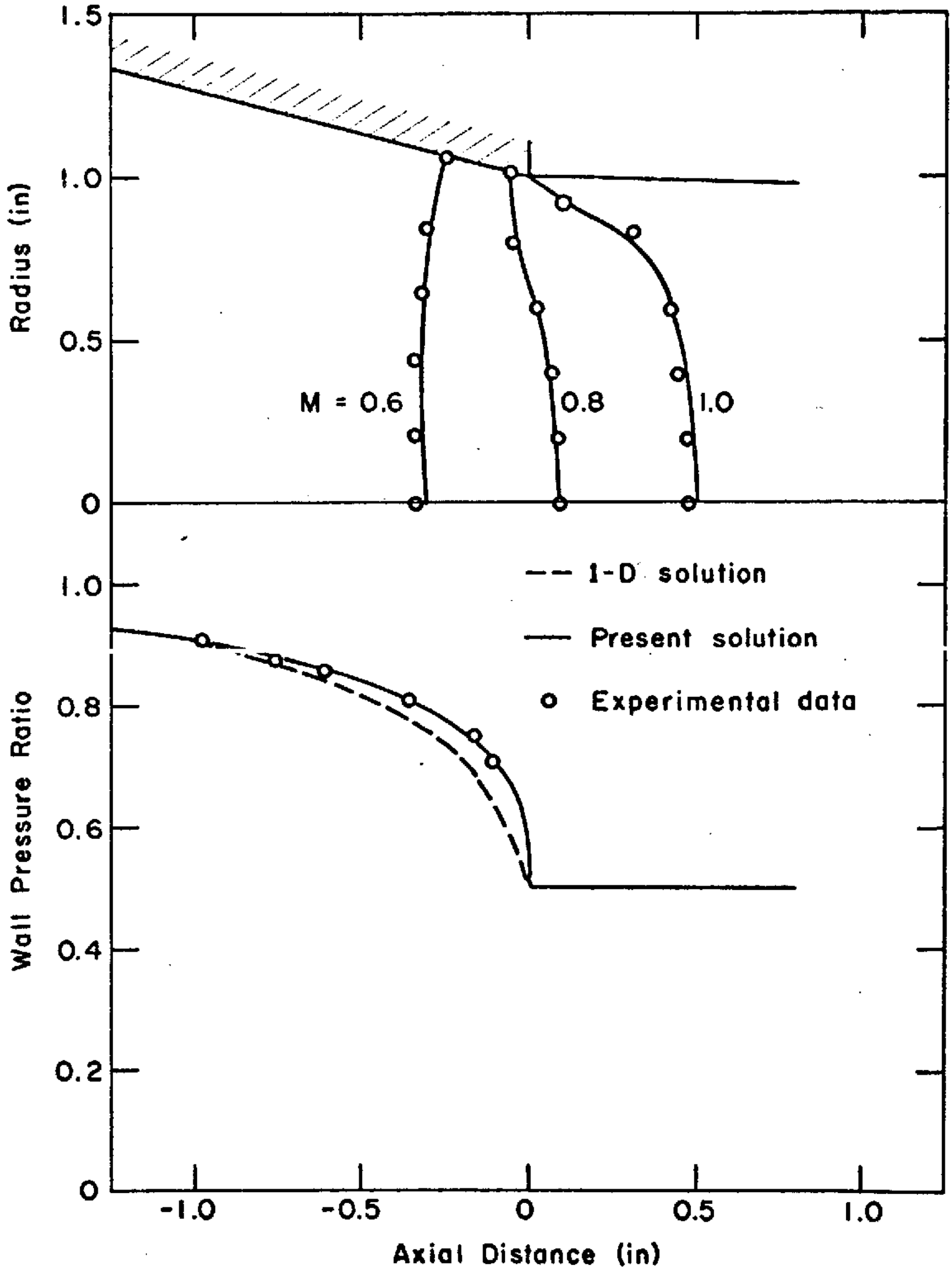


Figure 3. Mach number contours (above) and wall pressure ratio for 15° conical nozzle.

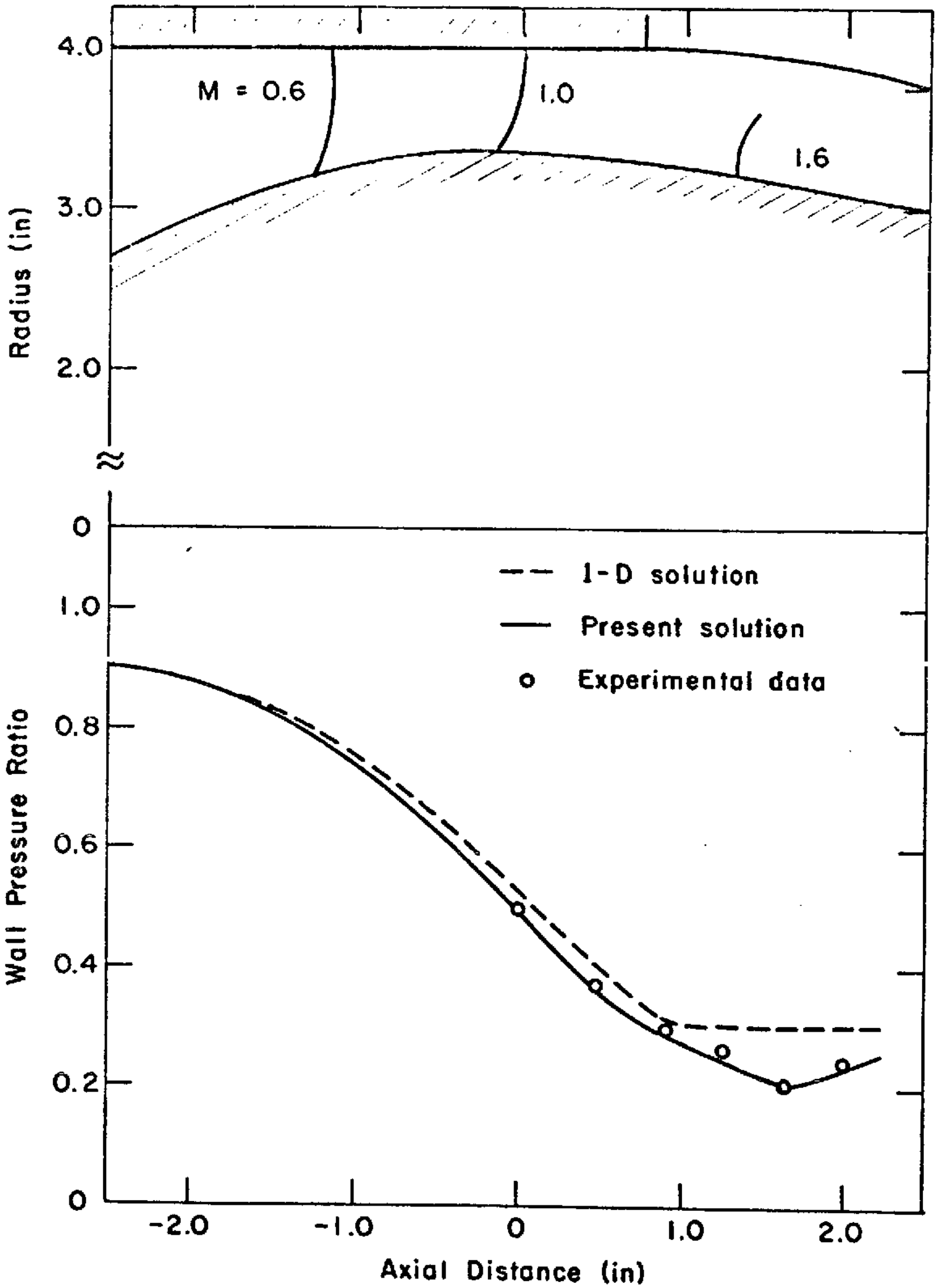


Figure 4. Mach number contours (above) and plug pressure ratio for 10° conical, plug nozzle.