NASA Technical Memorandum 110295

Investigation of Low-Reynolds-Number Rocket Nozzle Design Using PNS-Based Optimization Procedure

M. Moin Hussaini and John J. Korte
Langley Research Center, Hampton, Virginia

November 1996

National Aeronautics and
Space Administration
Langley Research Center
Hampton, Virginia 23681-0001

Abstract

An optimization approach to rocket nozzle design, based on computational fluid dynamics (CFD) methodology, is investigated for low-Reynolds-number cases. This study is undertaken to determine the benefits of this approach over those of classical design processes such as Rao's method. A CFD-based optimization procedure, using the parabolized Navier-Stokes (PNS) equations, is used to design conical and contoured axisymmetric nozzles. The advantage of this procedure is that it accounts for viscosity during the design process; other processes make an approximated boundary-layer correction after an inviscid design is created. Results showed significant improvement in the nozzle thrust coefficient over that of the baseline case; however, the unusual nozzle design necessitates further investigation of the accuracy of the PNS equations for modeling expanding flows with thick laminar boundary layers.

Introduction

Two low-Reynolds-number rocket nozzles have been designed by using a CFD based optimization procedure. The investigation entailed the following five steps:

(1) Link a well-tested computational dynamics (CFD) code to an optimization computer code.
(2) Create an objective function for the thrust.
(3) Obtain previously studied nozzle geometry that were classically designed.
(4) Optimize those cases with CFD.
(5) And compare the thrust coefficients of the CFD based designs with those of the classically designed cases.

The study was conducted to determine whether CFD contributed a significant performance advantage over classical design methods in the optimization of rocket nozzles with large boundary layers. CFD-based optimization has proved highly beneficial in the design of hypersonic wind-tunnel nozzles, in which the viscous effects are very high.[1] If CFD-based optimization proves beneficial for low-Reynolds-number rocket nozzles, then significant advancement will be realized in the design process of resistojets, arcjets, and other nozzles that are designed for low chamber pressures.

Relevance and Background

With an increasing demand for the reduced cost of space endeavors, researchers are forced to investigate new technologies, such as improving fuel economy. Greater fuel efficiency can be achieved in satellites and spacecraft with the design of better propulsion and positioning systems. A push for improved technology in this area has led to the development of rocket nozzles that can operate effectively and reliably in space. Resistojets, for example, meet the high efficiency and high reliability requirements of space travel.[2] These rockets are not intended for the initial launch function, nor are they able to create thrust forces comparable to those of conventional rockets; rather, their purpose is to produce small amounts of thrust over long periods of time, to gradually accelerate a spacecraft up to a certain speed. To meet these requirements, a resistojet nozzle must be able to operate at low chamber pressures and must be as small as possible in size. Low-Reynolds-number nozzles differ from conventional high-Reynolds-number rocket nozzles because of the large boundary layer that is present at the nozzle exit. Despite their unconventionality, these low-Reynolds-number nozzles are still primarily designed using conventional means. Furthermore, despite the need for efficiency, relatively little effort has been devoted to developing methods for designing low-Reynolds-number thrust nozzles.

Existing Methods for Rocket Nozzle Design

In the early years of rocket nozzle design, conical nozzles were designed because methodology did not exist for the design of efficient contoured nozzles. Although they were necessary at the time, conical nozzles were an inadequate solution. Such nozzles resulted in performance losses because they did not produce axially directed exhaust; in some cases, these nozzles were excessively heavy because of their length. Short and efficient contoured nozzles were finally developed when the truncated perfect nozzle approach was pioneered; in this approach, a wind-tunnel nozzle was designed for uniform flow and then truncated at a much shorter length. Ahlberg et al.[3] present some results that were obtained by using this method; Hoffman[4] discusses some variations of the original method.

Within the same time period, an alternate approach to this problem, based on the calculus of variations, was formulated. This method was first investigated by Guderley and Hantsch[5] and was improved upon by Rao.[6] According to the procedure outlined by Rao, the length, ambient pressure, and flow properties in the immediate vicinity of the throat are the governing conditions under which thrust is maximized. This classical process assumes that an isentropic flow exists in the nozzle; thus, a variational integral is formulated by taking into account an appropriately selected control surface. The solution of the variational integral yields certain flow properties on the control surface, and the nozzle contour is constructed by the method of characteristics to give the desired flow. Historically, Rao's method has been considered the best, as other classical approaches fail to surpass its performance. However, the one drawback to Rao's method, which is its dependence on an inviscid design, leaves room for improvement.

Classical procedures are based on an approximation of viscous effects; these older methods rely on an inviscid design (such as the Rao's method of design). After an inviscid design has been completed, a boundary-layer correction is added to compensate for the viscous effects. Because, inviscid and viscous effects could not be calculated simultaneously, the effects of such frictional phenomenon were approximated with a boundary-layer correction. Although this type of procedure is reliable to a certain extent, an inherent amount of error exists in any approximation. For instance, in the design of hypersonic wind-tunnel nozzles, Candler and Perkins[7] showed that classical methods break down when the boundary layer reaches the same order of magnitude as the local radius. Similar results were also found by Kim[8] for low-Reynolds-number rocket nozzles. Thus, as the Reynolds number of a nozzle decreases so does the accuracy of the boundary-layer correction. In turn, Rao's design method becomes less effective because it relies on the boundary-layer approximation to correct for viscous effects. An alternative to calculating the inviscid and viscous flows separately is to numerically solve the Navier-Stokes (NS) equations; with these equations, viscous effects can be accurately determined. In the past, computational capability was such that the NS equations could not be used in the design of contours. More recent advances in computational technology have allowed scientists to calculate the NS equations, which previously had to be simplified for computation. Such advances have given rise to efficient CFD codes that have eliminated the need to approximate viscous effects in aerodynamic design.

In this study, CFD-based design was predicted to show significant improvement over classical design procedures. To test this hypothesis, thrust optimization in resisto-jet nozzles was chosen because it is an area likely to benefit significantly from CFD-based design. Kim[5] states, "ongoing research in low thrust space propulsion has resulted in many high-performance space propulsion rockets such as arcjets, resistojets and magneto plasma dynamic thrusters." Resistojet nozzles have very low Reynolds numbers and are not well suited to classical design procedures because such thrusters operate at low chamber pressures. The dimensions of such nozzles are small, and the Reynolds number is low; consequently, the viscous effects are high. Nevertheless, the classical method of design is still widely used in determining nozzle wall contour. A design improvement in

such nozzles would be a significant development, and CFD is an appropriate tool for realizing this goal.

CFD-Based Optimization

The use of CFD codes in design has usually been to provide a direct analysis of the performance of existing or proposed designs; CFD codes have been coupled with design procedures for a limited number of applications. The most powerful and general CFD design procedures are based on solving an optimization problem. A typical design or optimization includes the following steps:

(1) The design requirements are specified.
(2) An objective function is constructed, the minimum or maximum of which yields the design requirements.
(3) The set of design parameters or variables is specified.
(4) An initial value for each of the design parameters is estimated.
(5) An initial CFD solution is computed by using the estimated design parameters.
(6) The objective function is computed from the difference between the design requirements and the computed solution.
(7) The sensitivity of the objective function to the design parameters is calculated (sensitivity coefficients).
(8) An optimization problem is solved to generate a new set of design variables.
(9) A new CFD solution is computed and compared with the design requirements.
(10) If the design requirement is met or a minimum or maximum is reached, then the procedure stops, otherwise the process is repeated from step 6 onward.

In this investigation, the above-mentioned optimization procedure has been applied to the design of low-Reynolds-number rocket nozzles. For computational efficiency, the nozzle flow field was computed in two steps. The first step was to compute the subsonic and transonic flow regions with the NS equations. This step is time iterative, which makes it a time consuming but necessary step. Upon convergence of the NS equations, a subset of the same equations can be used to more quickly determine the supersonic portion of the flow field. The subset of formulas is known as the parabolized Navier-Stokes (PNS) equations. These formulas are simplified or parabolized by neglecting both streamwise diffusion effects and a portion of the subsonic streamwise pressure gradient. As a result, the PNS equations can be integrated by using efficient space-marching techniques. For many practical flow fields, the approximations made by the PNS code are valid as long as streamwise separation does not occur or as long as the expansion is not too large. For a more detailed description of this procedure, see Korte et al.[9]

With the increased use of supercomputers, both CFD and CFD-based design optimization have become significantly faster and more efficient. In the current study these new techniques were successfully used to improve the existing design process for low-Reynolds-number rocket nozzles. Gains in thrust of greater than 3 percent have been obtained for a very simple nozzle wall geometry. This paper discusses the results and the methodology used. In this work, the CFD-based thrust optimization process is demonstrated by optimizing a low-Reynolds-number thrust nozzle for conditions that have been previously investigated by other researchers.

Numerical Methods

The CFD solutions for the nozzles are calculated with the code described in Ref. 9. The nozzle is divided into two regions for improving computational efficiency: the subsonic-transonic section and the supersonic-hypersonic section. In the subsonic-transonic section,

the full NS equations are solved. In the supersonic-hypersonic section, the PNS equations are solved.

Transformation to Computational Coordinates

The physical domain is transformed into computational space by using the generalized transformation

$$\xi(x,y)=\xi, \quad \eta(x,y)=\eta \tag{1}$$

where ξ represents the streamwise direction and η is the crossflow direction.

The nozzle wall radius or height $y_w(x)$ is defined by using a set of points, the coefficients to a spline, or an analytical expression. For a given streamwise location, the stretching transformation of Roberts can be used either to cluster points at the wall only or at the wall and the centerline.[10] For the extremely thin boundary layers typically found in the throat region, neither of these stretching functions is satisfactory. If points are clustered only near the wall, then an insufficient number of points is at the centerline and the solution deteriorates in that region. Likewise, if points are stretched at both the wall and the centerline, then an insufficient number of points is located in the middle of the domain to accurately compute the flow field. To overcome this problem, two different stretching functions can be blended together by using a fourth-order polynomial in $\bar{\eta}$.[11]

This transformation is given by

$$y(j)/y_w(x) = \begin{aligned} &(1-\bar{\eta}^4)\left\langle 1-\beta_c+2\beta_c/\left\{1+\left[(\beta_c+1)/(\beta_c-1)\right]^{(1-\bar{\eta})}\right\}\right\rangle \\ &+\bar{\eta}^4\left\langle \beta_w-2\beta_w/\left\{1+\left[(\beta_w+1)/(\beta_w-1)\right]^{\bar{\eta}}\right\}\right\rangle \end{aligned} \tag{2}$$

$$\bar{\eta} = (j-1)/(j_{\max}-1) \tag{3}$$

The coefficient β is a positive number greater than 1. The closer that β is to 1, the more clustered the grid becomes. The first line of equation (2) clusters the points at the centerline based on the value of β_c, and the second line clusters the points at the wall based on β_w. The clustering at the centerline is held fixed at β_c =1.02, the clustering at the wall is selected for each particular case.

Optimization Setup

In this section, the assumption is made that the shape of the subsonic-transonic nozzle contours of the nozzle has been specified and that an NS solution has been computed. The PNS solver used in reference. (9), (with some minor modifications) is coupled to the optimization program described below.

Objective Function

A nonlinear optimization problem is solved to determine the design parameters by the minimization or maximization of an objective function. An objective function *Obj* is dependent on a set of design parameters ***X:***

$$Obj(X) = \int_0^{Aexit} (\rho u^2 + p) dA \tag{4}$$

where ρ, u, p, and A are the local density, axial velocity, pressure, and cross-sectional area, respectively. For a rocket nozzle, *Obj(X)* is defined as the exit thrust (into a vacuum) which must be maximized to obtain an optimum design.

Design Parameters

Like the objective function, the selection of the design parameters depends on the problem to be solved. The ideal set would contain the minimum number of elements and would be strongly coupled to the objective function. The design parameters are usually coefficients used to define a wall boundary or quantities that specify the flow-field conditions. For a nozzle design, the flow conditions are usually given, and the wall contour must be determined.

Wall Contour

Three different functions (i.e. linear, cubic, and sixth-order polynomial equations) are used to describe different nozzle wall shapes downstream of the throat. The linear function describes a conical nozzle, for which the final slope (or angle) is the design parameter. The length is then determined by using the area ratio and the specified angle. For a contoured nozzle, defining the wall becomes more complicated. The simplest method in this case would be to fit a third-order polynomial to the beginning radius, initial slope, exit slope, and length.

Subsonic-Transonic Section Geometry

The subsonic-transonic section is specified with a circle:

$$x^2 + (y_w - R_c - r^*)^2 = R_c^2 \tag{5}$$

where R_c and r^* are the throat radius of curvature and the throat radius, respectively.

Linear (Conical Nozzle) Geometry

The conical nozzle begins at the axial location at which the slope of the subsonic-transonic section equals the tangent of the cone angle:

$$y_w = y_o + \tan\theta_w(\Delta x) \tag{6}$$

where θ_c is the cone angle, $\Delta x = x - x_o$ and x_o and y_o are the initial points for the conic section.

Contoured Nozzle Geometry-Cubic Polynomial

The cubic polynomial begins at the first section used in the PNS calculations and the radius

$$
\begin{aligned}
y_w &= y_o + y_o'(\Delta x) + c\Delta x^2 + d\Delta x^3 \\
c &= 3(y_L - y_o - y_o' L)/(L^2) - (y_L' - y_o')/L \\
d &= [(y_L' - y_o')/L^2] + 2(-y_L + y_o + y_o' L)/(L^3)
\end{aligned}
\tag{7}
$$

where $\Delta x = x - x_o$, and (x_o, y_o, y_o') is the initial point and slope for the supersonic section, (y_L, y_L') is the radius and slope at the exit, and L is the length of the contoured section.

The conical and contour nozzle geometry used in this study are shown in Fig. 1.

Low-Reynolds-Number Thrust Nozzle Design and Optimization

The CFD-based thrust optimization process was conducted by:

(1) choosing an existing nozzle design that was previously calculated by other researchers;
(2) varying the grid density of the solution to determine if the specified resolution produces accurate results;
(3) optimizing those cases by using a CFD code coupled to an optimizer,
(4) and looking for improvements in the thrust coefficient for each case.

This four-step process was followed to determine the advantages of CFD-based optimization over existing methods of design for low-Reynolds-number rocket nozzles. Kim[8] investigated 20° and 30^0 degree conical resistojet nozzles and found that they have higher thrust coefficients than contoured resistojet nozzles designed with Rao's method for the same area ratio. For this reason, the optimum angle for a conical resistojet nozzle with an area ratio of 82 was determined to provide a target for the CFD-based optimization process; thus, if the CFD-based optimization procedure produces a contoured design that has a higher thrust coefficient than is obtained from the conical nozzle with the optimum angle, the hypothesis is supported and the CFD-based design process is a better design process for the studied cases.

Solution Accuracy and Grid Dependence

Initial calculations were made with the flow and geometry conditions given by Kim[8] for 20° and 30° conical nozzles. Viscous flow solutions were computed for hydrogen-gas flows through the nozzles. The following assumptions were made: the flow was laminar; the back pressure was 0; the wall was adiabatic; the stagnation pressure was 150000 Pa, and the stagnation temperature was 1500K. The computed nozzles had a throat radius of 4.2×10^{-4} m and an exit area ratio of 82.

Obviously, grid density was an important factor in determining both the precision of a solution and the computational time. To minimize the computational time in demonstrating the accuracy of the solution, the 20° conical nozzle was run with different normal grid densities of 63 and 125 points. The stability requirements of the explicit PNS space-marching algorithm force the streamwise grid to have a large number of stations (> 1000), which ensures accurate resolution in the streamwise direction. A constant grid-stretching factor of β_w =1.2 was used in all calculations. Figure 2 compares the streamwise momentum profiles at the nozzle exit for the different normal grid densities. Because the streamwise momentum is integrated to obtain the nozzle thrust, the profiles are believed to be a good measure of the solution accuracy. Very good agreement was obtained for the two grid densities; the 125 point solution predicted a slightly lower streamwise momentum in the inviscid core. The nozzle thrust coefficients for the 63 and the 125 normal point distributions were 1.482 and 1.480, respectively, which is a difference of only 0.14

percent. This agreement was sufficient to justify the use of a grid with 63 normal points for the nozzle design and analysis. A complete NS calculation was then computed for the nozzle to determine the effect of the PNS approximation on the thrust coefficient. A NS solution was computed with the same distribution of 63 normal points and 235 axial stations. The thrust coefficient for the NS calculation was 1.492, which compares favorably with Kim's[8] NS calculation of 1.498 (603 lb_f-sec/lb_m vacuum specific impulse) with 60 normal grid points. The difference in the thrust coefficients for the NS and PNS solutions is less than 0.7 percent. We expect the PNS solutions to follow the same trends as the NS solutions, which justifies the use of the PNS equations in the design procedure.

Low-Reynolds-Number Conical Nozzle Design

Next, the optimum angle was determined for a low-Reynolds-number conical thrust nozzle with a fixed area ratio of 82. The throat geometry was constructed as a circular arc (fig. 1) and the PNS calculations began at 1.747×10^{-4} m downstream of the throat, where the wall radius and slope are 4.447×10^{-4} m and 0.2887, respectively. The subsonic flow field of the nozzle was predicted by solving the NS equations to determine the initial conditions for the PNS calculations. The PNS code was then modified to construct conical geometry as a function of the cone angle (Eq. 6) and to compute the nozzle thrust (Eq. 4). The solution had a grid density of 63 normal points, approximately 40,000 streamwise stations, and a grid-stretching factor of 1.2. The remaining calculations presented were computed on similar grids.

After the modification was completed, the CFD code was linked to the OptdesX[12] software system, which is an interactive computer program for computer-aided optimization and design. An optimization problem was defined to maximize thrust by determining the optimum expansion angle. For each case, the software's default optimization scheme was used (Generalized Reduced Gradient algorithm). The derivative of the nozzle thrust with respect to the cone angle was computed with the forward-difference selected by OptdesX as the best method with a step size of 0.001. Two cases were run using starting angles of 20° and 30°. Because of the simplicity of the objective function (i.e. one design parameter), the optimizer required few iterations to achieve the optimum expansion angle of 25.96° resulting in a thrust coefficient of 1.488. This result was achieved using the different initial conditions for the conical angle. The Mach contours for the optimum conical nozzle are shown in Fig. 3. A solution was reached on an Iris Indigo after 3 iterations and 25 analysis calls (approximately 3 hr).

Low-Reynolds-Number Contoured Nozzle Design

For this problem, the wall contour was defined with a third-order polynomial. Only two variables, the length and the exit slope, were varied to determine the optimum geometry. A cubic was fitted to the previously determined optimum conical nozzle and was used as the starting point for the contour optimization. Then, the initial case was calculated and the optimizer varied the parameters in each iteration to generate a new wall contour. The CFD code produced a thrust coefficient value for each contour, and the process continued until an optimum was reached. The derivatives were computed with a forward difference-method with a step size of 0.001. The initial design parameters were specified from the optimum conical nozzle. The solution required 23 analysis calls (approximately 3 hr on an Iris Indigo.) The exit slope converged to the side constraint value of 0.0, and the nozzle length (from the throat to the exit) converged to 0.00767 cm. The final thrust coefficient was 1.496. The Mach contours for the optimum third-order wall contour are given in Fig. 4.

To determine the effect of the exit angle side constraint on the solution, it was changed to -0.364 (-20 degrees) and the problem was restarted. The exit slope converged to the side constraint, the length to 0.00779 cm. and the thrust coefficient increase to 1.538. This

resulted in an unusual shaped nozzle (Fig. 5). To determine if this was an artifact of maintaining the area ratio at a fix value of 82, it was added as a design variable in the problem. This solution did not change when the area ratio was included as a design variable. The converging nozzle exit (-20 degrees) design is an unexpected result and does not follow normal convention. Previously, it had been assumed that the use of the PNS solutions in the design procedure would follow the same trends as NS solutions. However, the effect of ignoring part of the subsonic streamwise pressure gradient in the PNS equations may be more significant in large laminar boundary-layers present in this nozzle flow field. Additional investigations comparing the PNS and NS solution for this nozzle design need to be done to determine the validity of this approach and to quantify the effect of the approximation made using the PNS solutions for low Reynolds number expanding flows.

Concluding Remarks

A computational fluid dynamics approach to low-Reynolds-number rocket nozzle design has been investigated and compared with previously developed methods. Earlier studies have found that conical nozzles produce higher thrust than contoured nozzles designed with classical methods. The current study demonstrates a method that is an improvement to both the classical method and the conical design based on solutions to the parabolized Navier-Stokes (PNS) equations. A CFD code was modified to provide a thrust coefficient value for individual cases. The modified analysis software was linked to the OptdesX optimization software, and a contoured nozzle design was generated by solving an optimization problem. The CFD-based optimization procedure achieved better results than those obtained from Rao's method or from conical nozzles, however, the contoured nozzle designs were not as expected. The best results were obtained when the nozzle exit angle was allowed to be negative. Further study was suggested to compare the PNS results with Navier-Stokes solutions to see if the design parameters for maximum thrust coefficient agree and to quantify the accuracy of the PNS solutions for low Reynolds number expanding flow fields.

References

[1] Korte, John J.: Aerodynamic Design of Axisymmetric Hypersonic Wind-Tunnel Nozzles Using Least-Squares/Parabolized Navier-Stokes Procedure. *AIAA Journal of Spacecraft and Rockets*, vol. 29, Sept.-Oct. 1992, pp. 685-691. *Errata*, vol. 29, Nov.-Dec. 1992, pp. 870-871.

[2] Loh, W. H. T., ed.: *Jet, Rocket, Nuclear, Ion, and Electric Propulsion: Theory and Design*. Springer-Verlag, 1968.

[3] Ahlberg J. H.; Hamilton, S.; Migdal, D.; and Nilson, E. N.: Truncated Perfect Nozzles in Optimum Nozzle Design. ARS Journal, May 1961, pp. 614-620.

[4] Hoffman, Joe D.: *Design of Compressed Truncated Perfect Nozzles.* AIAA Paper 85-1172, Nov. 1995.

[5] Guderley, G.; and Hantsch, E.: Beste Formen Für Achsensymmetrische Uberschallschubdusen. *Zeitschrift für Flugwissenschaften*, Brauschweig, Sept. 1955.

[6] Rao, G.V.R.: Exhaust Nozzle Contour for Maximum Thrust. *Jet Propulsion*, vol. 28, June 1958, pp. 377-382.

[7] Candler, G.; and Perkins, J.: *Effects of Vibrational Nonequilibrium on Axisymmetric Hypersonic Nozzle Design*. AIAA Paper 91-0297, Jan. 1991.

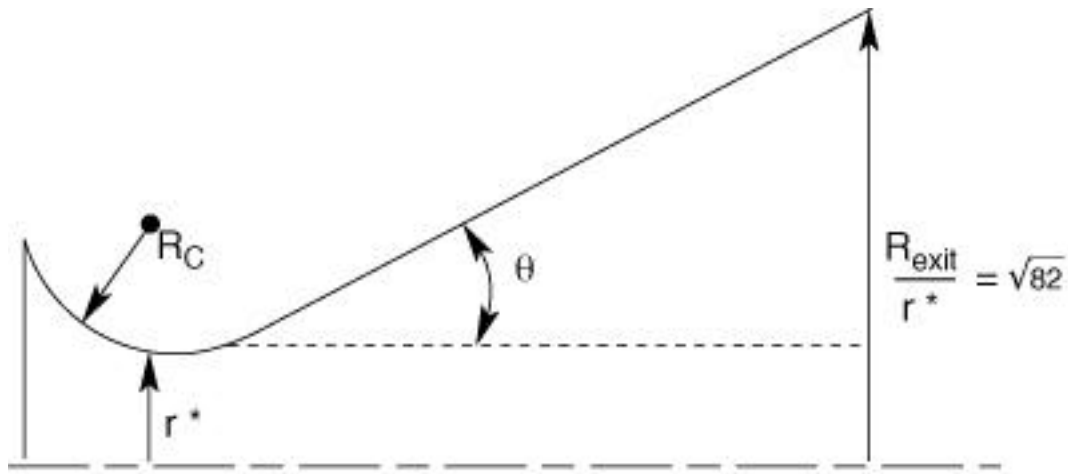
[8] Kim, S.: *Calculations of Low Reynolds Number Rocket Nozzles.* AIAA Paper 93-0888, Jan. 1993.

[9] Korte, J. J.; Kumar, A.;Singh, D. J.; and White, J. A.: CAN-DO - CFD-Based Aerodynamic Nozzle Design & Optimization Program for Supersonic/Hypersonic Wind Tunnels. AIAA Paper 92-4009, July 1992.

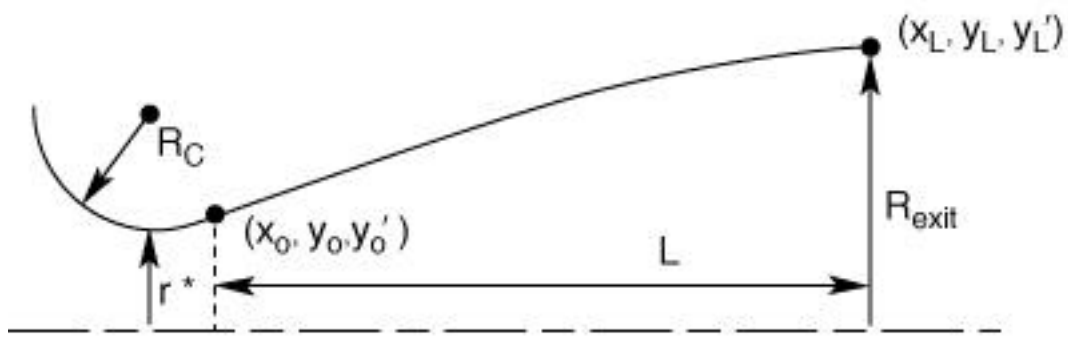
[10]Anderson, D. A.; Tannehill, J. C.; Pletcher, R. H.: *Computational Fluid Mechanics and Heat Transfer.* McGraw-Hill, New York, 1984, pp. 247-251.

[11] Korte, J. J.; and Hodge, J. S.: *Flow Quality of Hypersonic Wind-Tunnel Nozzles Designed Using Computational Fluid Dynamics.* Journal of Spacecraft and Rockets, vol. 32, July-Aug. 1995, pp. 569-580.

[12]OptdesX, A Software System for Optimal Engineering Design, Release 2.0. Design Synthesis, Inc. 1995.



(a) Conical nozzle.



(b) Contoured nozzle geometry.

Fig. 1. Conical and contoured nozzle geometry design parameters.

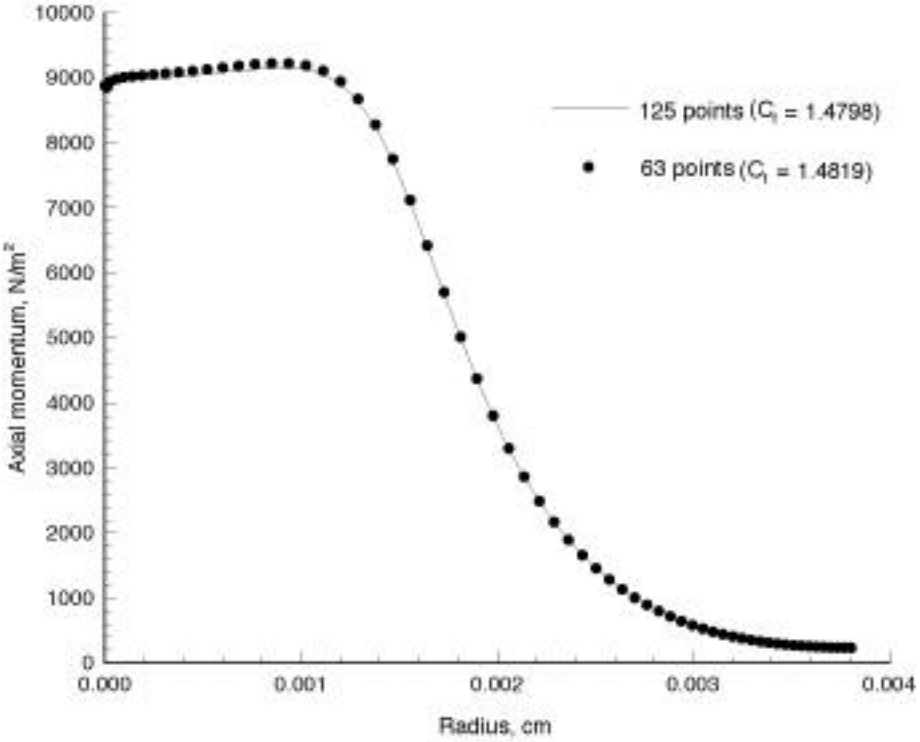


Fig. 2. Axial momentum profile at nozzle exit.

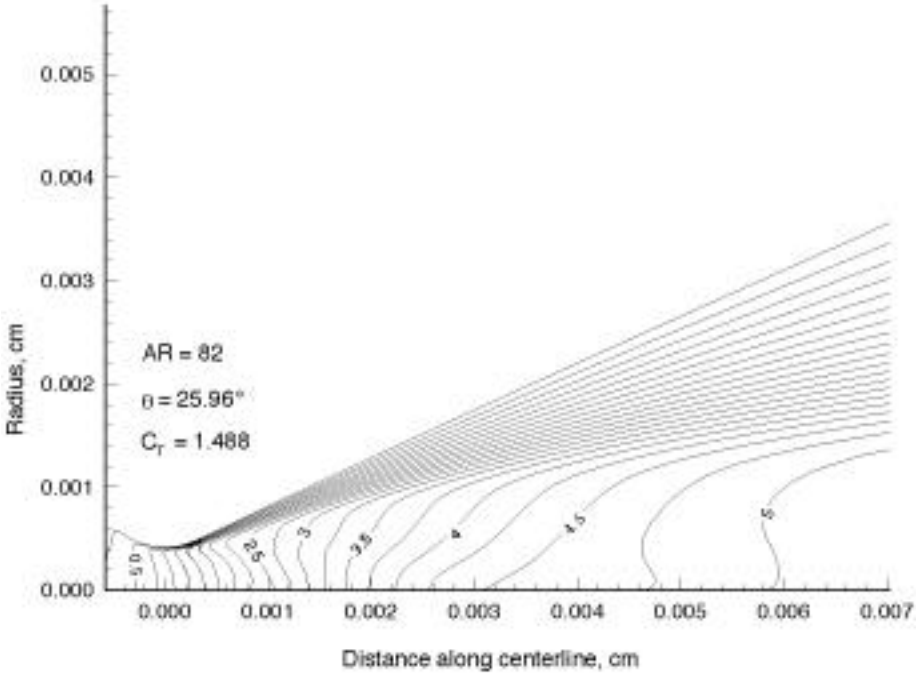


Fig. 3. Mach number contours for optimum conical nozzle design.

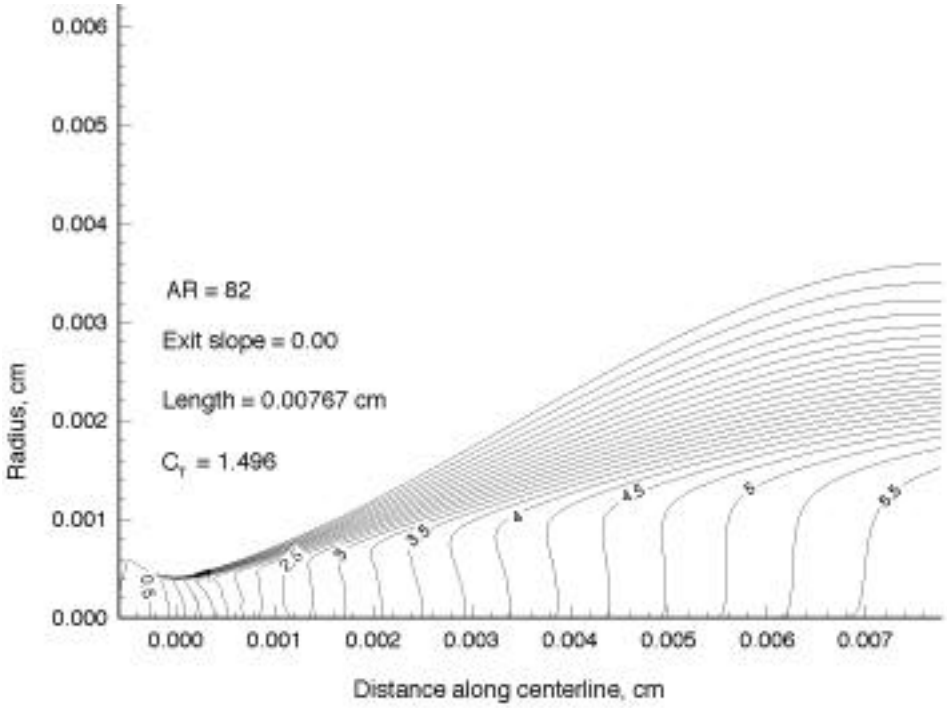


Fig. 4. Mach number contours for contoured nozzle design with minimum wall exit angle of 0°.

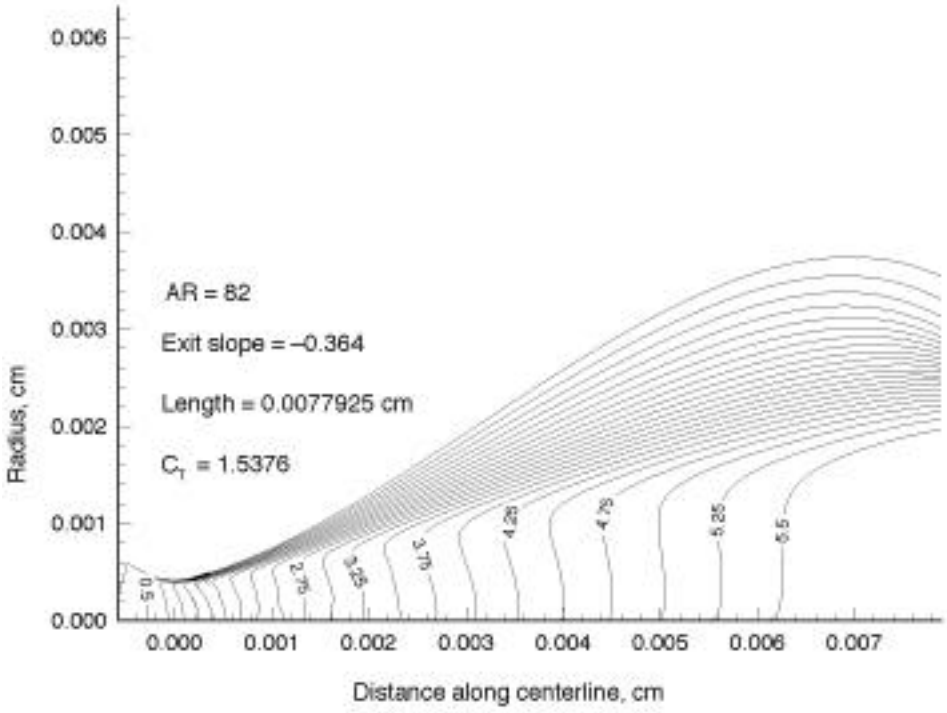


Fig. 5. Mach number contours for contoured nozzle design with minimum wall exit slope of –0.364.