24th Applied Aerodynamics Conference
5 - 8 June 2006, San Francisco, California

AIAA 2006-3665

Contour Design Techniques for Super/Hypersonic Wind Tunnel Nozzles

Frederick L. Shope*
Aerospace Testing Alliance
Arnold Engineering Development Center, Tennessee 37389

A brief tutorial is offered on several techniques for designing nozzle contours for supersonic and hypersonic wind tunnels. A set of conditions sufficient for the existence of a contour that will deliver a uniform test section flow is proposed. A specific iterative second-order-accurate algorithm for solution by the method of characteristics (MOC) is described along with its use in contour design. A widely distributed design code, the CONTUR code by J. C. Sivells, is discussed in some detail, and recommendations for application procedures based on CONTUR are given. Design examples from CONTUR are shown. With MOC as a starting point, a brief description of design-by-analysis methods is presented along with a sample design.

I. Introduction

Hypersonic cruise vehicles must push the state of the art in many areas of technology to achieve practicality and simply to survive the harsh aerothermal flight environment. Vehicles must necessarily operate close to, but within, safe limits without large safety factors. As hypersonic flight experience slowly accumulates, it is becoming increasingly clear just how difficult it will be to achieve practical, safe, and routine hypersonic flight. Accordingly, the need to achieve successful vehicle designs mandates highly accurate design information on aerodynamics, aeroheating, and propulsion system performance. The source of much of this design information is the world's complement of hypersonic wind tunnels, which must satisfy increasingly stringent flow quality requirements in order for designers to know accurately the limits of hypersonic flight. Of the many factors that affect wind tunnel data quality, flow quality, with its dependence on the nozzle contour design, is the focus of the present paper. The intent here is to briefly survey techniques for design of nozzle contours for supersonic and hypersonic wind tunnels.

The role of the nozzle in a hypersonic facility is to expand a low-subsonic flow to a supersonic flow in the test section. Accordingly, the nozzle is almost always in the form of a converging duct followed by a diverging duct, the two being separated by a minimum area throat. The ideal nozzle will deliver a flow that is uniform in all flow properties over a region of the test section. The delivered flow must be at specified flow properties, usually a specified flow speed, temperature, and pressure. The ultimate need, of course, is to duplicate flight in the real atmosphere. However, as any airline passenger experiencing turbulence knows, the atmosphere is not necessarily a uniform medium to fly through. The purpose of uniform wind tunnel flow is to provide a controlled environment for making measurements.

A characteristic of all hypersonic facilities is that the flow must be heated, either to avoid liquefaction of some species of the test medium, or to duplicate flight temperature. The heating rate is sufficiently large for continuous flow beyond about Mach number 6 that the nozzle must usually be cooled; this is a primary consideration in the structural design and, to a lesser extent, the contour design of a nozzle. The heat flux reaches a maximum near the throat, and for facilities that attempt to duplicate flight temperature, the heating rates are among the highest sustained rates in engineering problems. The facilities that use these types of nozzles include continuous-flow facilities; long run-time, blowdown facilities; Ludwieg tubes; piston-driven impulse facilities, and shock tunnels.

---

* The research reported herein was performed by the Arnold Engineering Development Center (AEDC), Air Force Materiel Command. Work and analysis for this research were performed by personnel of Aerospace Testing Alliance, the operations, maintenance, information management, and support contractor for AEDC. Further reproduction is authorized to satisfy needs of the U. S. Government.

Ú Engineer, Integrated Test and Evaluation Dept., 676 Second St., Arnold AFB, TN 37389-4001, AIAA Senior Member.

There are several types of supersonic/hypersonic wind tunnel nozzles. The most common type, and the primary focus here, is axisymmetric with a circular cross section at every station. A few facilities use two-dimensional (2-D) nozzles, where two opposite walls are contoured in a converging-diverging shape but are bounded by parallel walls, giving a rectangular cross section at every station. The 2-D configuration admits the use of flexible plates driven by jacks to alter the contour as needed to vary the flow speed in the test section. A single flexible plate-nozzle can produce many flow speeds in the test section. However, the 2-D nozzle is generally not used above Mach number 5 or 6 for two reasons: the complexity of water cooling a flexible structure, and the very small slit-throat height needed to achieve the large nozzle exit area ratio. With thermal deflections and pressure loading, variations in the slit height lead to unacceptable flow nonuniformity in the test section and might result in missing the target flow speed. A third type of hypersonic nozzle has an exit cross section with a shape tailored to a specific application, such as direct-connect scramjet combustor testing. For such an application, four sides of a near-rectangular cross-section nozzle may be contoured to deliver uniform flow. A fourth type of nozzle, used at least once in an arc-heated facility, has a circular throat and a semicircular exit, its purpose being to maximize exposure of wedge surface area to the hot jet for material testing. Finally, there is the concept of a minimum-length nozzle in which the throat is sharp, and thus causes the flow to turn suddenly to the nozzle inflection angle (the maximum angle of the contour).

Although the goal of contour design is to deliver a perfectly uniform flow, there are limitations to what can be achieved through contouring alone. The following conditions are proposed as sufficient (in the mathematical sense) for a contour to exist that will deliver uniform flow:

- Continuum flow
- Steady flow
- Inviscid flow
- Isentropic flow
- 2-D or axisymmetric flow
- Uniform entrance flow at the nozzle inlet station
- Equation of state is perfect,[†] or in thermochemical equilibrium, or is otherwise representable with models that are approximations to equilibrium. Thermodynamically, the medium is a "simple compressible substance," meaning (1) that specification of two properties is sufficient to compute all other properties and (2) that the only reversible work mode is volume change (i.e., $P\,dv$ work).

The author does not intend to argue that the above conditions are necessary *and* sufficient. Clearly, the inviscid assumption is necessary, since zero velocity at the wall precludes the possibility of uniform flow across the test section. The isentropic assumption is not necessary because a uniform flow through a normal shock produces a uniform flow downstream of the shock. Thus, what is proposed is that the above list represents conditions sufficient for the existence of a rigorous contour that will yield uniform flow. If the actual flow violates the present sufficiency criteria, the designer should at least be aware that a rigorous contour design may not exist. If the present sufficiency criteria are satisfied, a rigorous contour design is possible using the method of characteristics; otherwise, approximate or possibly optimizable contours may be the only practical alternatives. In a real flow, of course, none of these conditions is ever satisfied without approximation. A primary observation is that the nozzle should not be expected to "clean up" an inlet flow that is highly nonuniform or unsteady or that may undergo a nonequilibrium expansion downstream of the throat. However, it may be possible to minimize these effects through formal design optimization.

Beyond theoretical considerations, there are also practical issues that will affect nozzle flow quality. These include machining accuracy, assembly accuracy (such as in joint alignment), surface finish and degeneration with age and usage (particularly in the throat region), boundary-layer transition (either from laminar to turbulent downstream of the throat or laminarization of a turbulent boundary layer entering the contraction), thermochemical nonequilibrium, condensation of water or other species, and contour distortion under pressure loading or thermal distortion.

An important fact to realize regarding rigorous contour design is that there is an infinite number of contours that will deliver uniform flow at specified conditions. Parameters that influence the shape of the contour include nozzle length and diameter, inflection angle, and specifications made for the various boundary conditions, particularly on the nozzle centerline for some design techniques. While any contour

[*]Herein, the term "perfect gas" means the gas is thermally and calorically perfect: it obeys the thermal equation of state $P=\rho RT$; the specific heats c_P and c_V are constant; $R= c_P-c_V$ is constant; and the specific-heat ratio $\gamma = c_P/c_V$ is defined.

among the infinity of choices that would yield uniform flow might be chosen, some choices are better than others. This fact is illustrated below.

Of the many factors that can influence flow quality, some effects can be significantly ameliorated or exacerbated by the choice of design options. In general, long nozzles with small inflection angles (as a rule, those of less than about 12 deg) yield the most uniform flow, which is a primary criterion for aerodynamic testing. On the other hand, for high-enthalpy facilities, such long nozzles produce large losses of the often elaborately achieved total enthalpy. For arc-heated and combustion-heated facilities, very short designs are usually chosen with the cognizance that some flow quality is being sacrificed. Short nozzles often have large inflection angles, which increase the concern about disastrous flow separation. Short nozzles also tend to have a small wall radius of curvature at the throat, perhaps as small as the throat radius, which is said to make accurate machining more difficult. Short nozzles with large flow expansion rates tend to exacerbate the effects of nonequilibrium on flow quality, while long nozzles give the flow more transit time to relax toward equilibrium. For many nonequilibrium nozzle flows, even absurdly long nozzles are not sufficient for relaxation to occur.

Many of the above assertions, stated without proof or reference, are based either on experience by the author or on experiences related to the author by prior designers or authorities.

Most operational nozzle design techniques can be placed in one of two categories: direct design or design by analysis (DBA). In direct design, the nozzle contour is computed as the primary output of the computation with only a single sweep through the flow field using some numerical procedure. Nearly all direct-design methods are based on the classical method of characteristics (MOC). In design by analysis, a computational fluid dynamics (CFD)-based analysis flow solver, to which the nozzle contour is an input, is coupled with a numerical optimization technique. The optimization technique alters the contour to drive the exit flow toward better uniformity and may require a flow-field solution for each contour perturbation.

The remainder of this paper presents ideas of possible use to nozzle designers who wish to obtain low-investment designs based on MOC techniques. A short discussion of DBA methods is also given.

II. Direct-Design Techniques – MOC

Direct-design techniques based on the method of characteristics date back to Busemann in 1929.[1] The essential idea is that specification of a uniform nozzle exit flow at some Mach number along with a centerline Mach number distribution, varying monotonically from sonic flow at the throat to the exit Mach number, is sufficient to allow construction of a characteristic network, starting at the nozzle exit and marching upstream. Any streamline in this flow will provide a suitable definition of an inviscid nozzle wall. Variations of this basic idea are preserved in every MOC-based design code in use today.

A. Method of Characteristics

There are several ways to derive a method of characteristics. In one approach, the 2-D, or axisymmetric, Euler equations are transformed to directions along which the partial differential equations reduce to ordinary differential equations. The finding is that

$$d\nu \pm d\theta = \sigma \frac{\sin\theta \sin\mu}{y} ds \tag{1}$$

where ν is the Prandtl-Meyer angle (for a perfect gas) given by

$$\nu = \sqrt{\frac{\gamma+1}{\gamma-1}} \arctan\sqrt{\frac{\gamma+1}{\gamma-1}(M^2-1)} - \arctan\sqrt{M^2-1}\,, \tag{2}$$

μ is the Mach angle defined as arcsin (1/M), θ is the flow angle relative to the nozzle centerline, s is the arc length along the characteristic, y is the radial distance, and γ is the specific heat ratio. For 2-D flow $\sigma = 0$, and for axisymmetric flow $\sigma = 1$. All angles are in radians. In Eq. (1) the "+" corresponds to right-running characteristics and the "–" to left-running characteristics. Mach lines emanate from a point at an angle of $\theta - \mu$ for the right characteristic and $\theta + \mu$ for the left characteristic. Equation (1) applies along the Mach lines defined by

$$\frac{dy}{dx} = \tan(\theta \pm \mu) \tag{3}$$

where "–" corresponds to right characteristics and "+" to left characteristics. The notation is illustrated in Fig. 1. A rigorous derivation may be found in Ref. 2.

A simple numerical method can be formed as follows: Eq. (1) is integrated along the left and right characteristics for a short distance, Δs. A constant mean value for the right-hand side (RHS) is assumed. Two characteristics emanating from points at which a solution is known are assumed to cross as shown in Fig. 2. The solution (values for θ and M) is known at points U (for upper) and L (for lower) in Fig. 2. The solution and coordinates of the unknown point N (for new) are to be computed. The integration of Eq. (1) yields the following algebraic equations:

$$\nu_N - \nu_U + (\theta_N - \theta_U) = \sigma \frac{\sin\theta^* \sin\mu^*}{y^*} \Delta s_{UN} \quad \text{(valid along right characteristics)} \tag{4}$$

$$\nu_N - \nu_L - (\theta_N - \theta_L) = \sigma \frac{\sin\theta^+ \sin\mu^+}{y^+} \Delta s_{LN} \quad \text{(valid along left characteristics)} \tag{5}$$

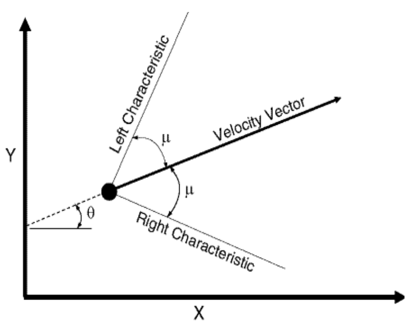

Figure 1. Characteristics Notation

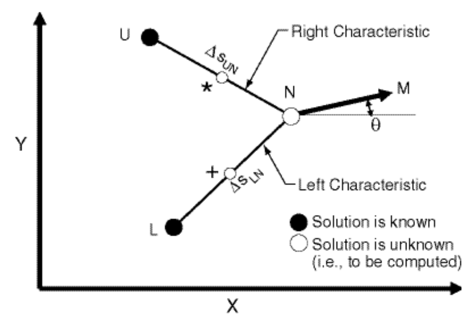

Figure 2. Notation for Method of Characterics

Equations (4) and (5) may be solved simultaneously for ν_N and θ_N. The coordinates of the point N are obtained by simultaneously solving the appropriately signed version of Eq. (3) along characteristics LN and UN. If the flow is axisymmetric (σ=1), the RHS of Eqs. (4) and (5) must be computed. These terms could be computed at the known points L and U. However, the resulting difference scheme would then only be first-order accurate, meaning the numerical error at each marching step would be proportional to the size of the step (Δs^1). For accuracy in the marching direction, second-order accuracy (error $\propto \Delta s^2$) is necessary. This necessity is asserted because, for first-order accuracy, the total accumulated error over all marching steps does not decrease with step size Δs, whereas for second-order accuracy, the *total* error is proportional to Δs and does decrease with step size. Thus, for a second-order scheme, the accuracy can always be improved by reducing the step size, but for a first-order scheme, perhaps not. Second-order accuracy in the solution of Eqs. (4) and (5) can be maintained by evaluating the RHS at the midpoints of the intervals ("*" and "+" in Fig. 2). One way to accomplish this is by iteration, evaluating the RHS first at the known points U and L, solving for point N, and then interpolating the solution at the midpoints. The iteration would continue until some measure of the error, say $\delta\nu_N$ and $\delta\theta_N$ between successive iterations, was small enough.

This MOC scheme can be used to calculate the flow in a given nozzle contour if a valid solution is specified along some curve laterally spanning the nozzle where the flow is all supersonic, such as just downstream of the throat. However, there are caveats limiting the choice of the starting line: a sonic line is unsuitable because the slope of the characteristics is infinite at M = 1, and the line cannot be a characteristic [one of Eqs. (4) or (5) would be lost]. Another consideration is that the numerical scheme must be altered to accommodate the boundary conditions. On the centerline, Eq. (1) appears to be singular (y = 0) but, in reality, never needs to be evaluated on the centerline: imposition of symmetry conditions ($d\theta = 0$) and midpoint evaluation preclude the need for computation of the RHS on the centerline. At a wall point, imposition of the wall slope effectively replaces the right-characteristic equation. A characteristic network computed with this algorithm for a Mach number 8 nozzle is shown in Fig. 3.

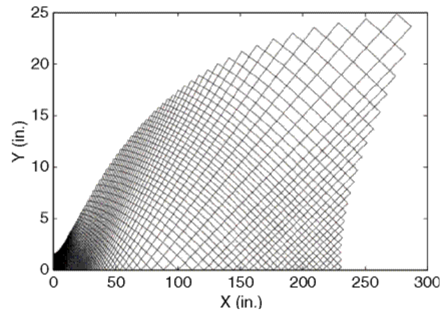

Figure 3. Characteristic Network for a Mach Number 8 Nozzle

B. MOC Applied to Contour Design

The MOC technique meshes very cleanly with the needs of a contour design procedure. Further, MOC is easier to apply to the direct-design problem than to analysis of a known contour because the boundary conditions are easier to apply in the design problem. With the help of Fig. 4, the direct-design procedure may be summarized as follows. First, the nozzle exit conditions are specified in terms of the exit Mach number and the nozzle exit radius. With the assumption of uniform and parallel exit flow along characteristic CD in Fig. 4, the entire solution (ν, θ, μ, and M) along CD is known. Grid points may be distributed along CD. Two additional boundary conditions are needed: (1) a centerline Mach number distribution, and (2) an upstream boundary condition near the throat. A typical throat boundary condition is the right-running throat characteristic TI in Fig. 4, which may be constructed if a flow-field solution is available for a specified throat contour up to point T. Note that for the design application, a characteristic *can* serve as a boundary condition. Other starting line alternatives with varying degrees of suitability include:

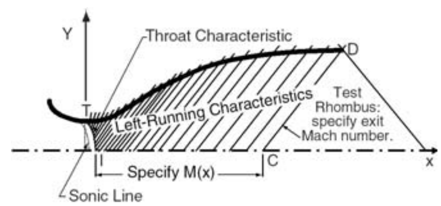

Figure 4. MOC Applied to Direct Design Problem

- a straight, vertical, slightly supersonic (uniform flow) line at geometric throat
- a straight, vertical, slightly supersonic line at proper quasi-1-D area ratio and x-station with flow angle linearly varying from zero on centerline to wall angle
- a spherical source-flow start line at a specified station
- a transonic solution for circular-arc throat along a vertical line at specified station
- a transonic solution for circular-arc throat along a straight line with specified slope angle

With the inlet and outlet boundary conditions defined, the centerline Mach number distribution may be chosen, preferably for compatibility with the inlet and outlet. The centerline Mach number distribution should be monotonic and should match some derivatives of the inlet and exit boundary conditions (typically derivatives of Mach number with respect to x up to third order, but frequently less).

With the boundary values computed at selected grid points along TICD in Fig. 4, the MOC design procedure can begin (Fig. 5). Starting at point C on the centerline, interior characteristic intersection points can be constructed, first marching up characteristic CD to construct the next upstream characteristic. A numerical procedure similar to that for the analysis problem can be applied with modifications to solve Eqs. (4) and (5), but this procedure will not be detailed here. As each characteristic is constructed, mass flux can be integrated along the characteristic, the left characteristic being treated as a surface through which mass is flowing, until a mass flow rate equal to that crossing CD is reached. This defines a point on the nozzle wall. When the entire characteristic network has been so erected, the complete nozzle contour is fully defined. Note that the MOC-based contour design does not proceed upstream of the throat. Consider also that the inviscid and perfect-gas contour so designed does not depend on tunnel stagnation conditions.

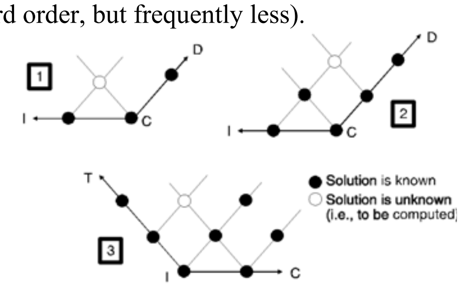

Figure 5. Construction of Characteristic Solution

C. MOC-Based Design Codes

Since large-scale digital computers and simple high-level programming languages (such as Fortran) became available, many MOC-based nozzle design codes have been developed. Two of these codes of particular interest to this author are described in Refs. 3 through 5. This paper concentrates on applying the code CONTUR developed by Sivells.[3,4].

CONTUR uses a method of characteristics much like that described above. The code designs axisymmetric, or 2-D, inviscid contours. The code assumes that the gas is perfect. A unique feature of CONTUR is its use of an analytical series solution for inviscid irrotational flow in a circular arc throat.[6,7]. The throat solution is used to construct a right-running throat characteristic (Fig. 6), which serves as the upstream boundary condition for the MOC computation. The throat characteristic, which originates at the geometric throat at the wall, permits the characteristic solution to fill almost the entire supersonic domain and permits the contour design to cover the entire length from throat to exit, without gaps. (Early design procedures filled a contour gap near the throat with nonrigorous polynomials.) Another feature of CONTUR is the optional use of a radial flow region that may be inserted between two separate characteristics networks, one network between the throat and the radial flow region, and a second between the radial flow region and the test rhombus. Within the radial flow region of an inviscid perfect gas, the flow is precisely described by the source flow equation,

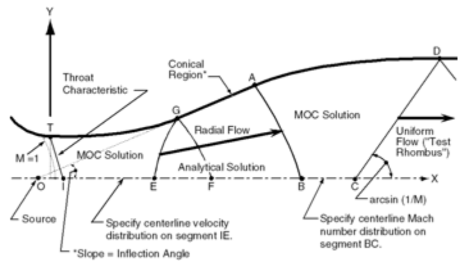

Figure 6. Notation and Design Procedure Used by Sivells

$$\left(\frac{r}{r^*}\right)^{\sigma+1} = \frac{1}{M}\left[\frac{2}{\gamma+1}\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}} \tag{6}$$

where r is the distance from the source to a point in the flow, r^* is the distance from the source to the sonic line, M is the Mach number at point r, and γ is the ratio of specific heats. Equation (6) is nothing more than the familiar Mach-number/area-ratio equation, with the flow areas assumed to be spherical caps (conical/axisymmetric, $\sigma = 1$) or circular arcs (2-D, $\sigma = 0$). For inviscid flow, the radial flow region corresponds to flow in a conical section of half-angle equal to the inflection angle. The radial flow region was originally included in the design procedure to reduce computational work, since the MOC computation is not needed for the radial flow region. However, this also affords the designer more options in choosing a contour. Outside of the radial flow region, or for nozzles that do not use a radial flow region, CONTUR offers the designer precoded choices for centerline distributions, including polynomials of degree 3 to 5. Independent distributions may be used upstream and downstream of the radial flow region, and the designer may choose to specify either Mach number or velocity distributions. The notation used by CONTUR is illustrated in Fig. 6.

To correct the inviscid contour for viscous effects, CONTUR offers the option of adding a boundary-layer displacement thickness to the inviscid coordinates. Reference 4 gives considerable detail on the boundary-layer calculation. CONTUR numerically integrates the von Karman integral momentum equation, assuming a power-law velocity profile within the boundary layer. The boundary layer is assumed to be turbulent, and there is no laminar option. The boundary-layer correction is added after the MOC solution is completed, and there is no need for iteration between the MOC solution and the boundary-layer solution. For 2-D nozzles with parallel side walls, the boundary layer is growing on all four walls, but only the contoured walls can be altered for the boundary-layer correction. For this case, the contoured wall coordinates are increased to account for the mass defect incurred on all four walls, assuming the displacement thickness is constant around the perimeter at each axial station. Thus, the boundary-layer correction is more approximate for 2-D nozzles. CONTUR also allows specification of a wall temperature distribution or an adiabatic wall. The boundary-layer correction can be applied to axisymmetric or 2-D nozzles.

The main parameters available in CONTUR to control the design include the following:

SFOA	Distance from throat to radial flow region, point A or G (Fig. 6)
ETAD	Inflection angle if a radial flow region is to be used

RC	Ratio of the wall radius of curvature at the throat to the throat radius
FMACH	Mach number at point F (Fig. 6) if a radial flow region is to be used
BMACH	Mach number at point B (end of the radial flow region on the centerline)
CMC	Design Mach number along characteristic CD at the test section entrance
SF	Inviscid nozzle exit radius or half-height for 2-D nozzles
XC	Location of point C (Fig. 6), used to control order of polynomial for centerline distributions

Reference 3 gives a detailed description of these and other input variables.

A necessary contour component that CONTUR does not design is the contraction. To this author's knowledge, a rigorous contraction design procedure based on aerodynamics has not been published. Sivells' recommendation for axisymmetric, or 2-D, nozzles was to form a contraction contour from two quartic polynomials separated by a conical section. His model is based on maintaining continuous curvature along the contraction and matching of curvature to that from the MOC viscous contour at the throat. The contraction model equations are as follows (Fig. 7):

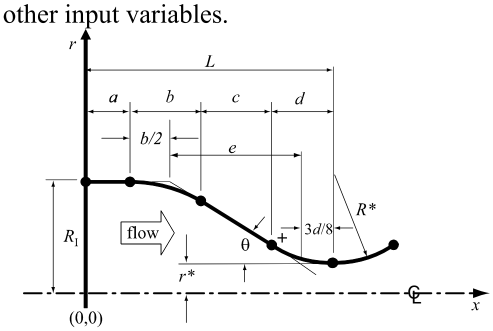

Figure 7. Construction of Characteristic Solution

$$x^+ = L - d \qquad r^+ = \frac{5d^2}{12R^*} + r^* \tag{7, 8}$$

$$b = 2\left(x^+ + \frac{r^+ - R_I}{\tan\theta} - a\right) \qquad e = \frac{R_I - r^*}{\tan\theta} \tag{9, 10}$$

$$d = \frac{3}{2}R^*\tan\theta \qquad c = e - \frac{b}{2} - \frac{5}{8}d \tag{11, 12}$$

$0 \le x \le a$ (cylindrical):

$$r(x) = R_I \tag{13}$$

$a \le x \le a + b$ (upstream quartic):

$$r(x) = R_i - \frac{b\tan\theta}{2}\left(\frac{x-a}{b}\right)^3\left(2 - \frac{x-a}{b}\right) \tag{14}$$

$a + b \le x \le a + b + c$ (conical):

$$r(x) = R_I + \left(a + \frac{b}{2}\right)\tan\theta - x\tan\theta \tag{15}$$

$a + b + c \le x \le L$ (downstream quartic)

$$r(x) = \frac{(L-x)^2}{12R^*}\left[6 - \left(\frac{L-x}{d}\right)^2\right] + r^* \tag{16}$$

The variables are defined in Fig. 7. The designer specifies R_I, R^*/r^* (CONTUR variable RC, viscous value if boundary layer is used), r^*, θ, a, and L.

D. Procedural Recommendations for Using CONTUR

The previous section has described the basic original CONTUR code as released by Sivells. The following section summarizes application experience and gives recommendations for ameliorating certain problems encountered in the past.

1. Complex Thermodynamics

CONTUR admits only a perfect gas (sometimes referred to as “flat-earth” thermodynamics). The user may specify the specific-heat ratio, the gas constant, and constants for a Sutherland-type viscosity law; however, there is no provision for more complex thermodynamics. Further, the MOC solution, the transonic solution, the model for radial flow, and the boundary-layer solution all assume perfect gas and are tightly integrated in the code. If deviations from perfect gas are critical, then CONTUR is not applicable or easily modifiable. Still, this author knows of no other design code with CONTUR’s capabilities but with better thermodynamics. Thus, much capability would be lost if CONTUR were to be abandoned. For this reason, compromise measures have been sought to extend the code beyond its regime of obvious applicability.

One such extension is to use an “effective” perfect gas. If a better thermodynamic model is available and appropriate, say, a model for the gas in chemical equilibrium, then the equilibrium model can be used to establish the correct inviscid nozzle exit area ratio based on quasi-1-D flow. The perfect-gas area ratio/Mach number relationship [e.g., Ref. 8, Eq. (80)] can be used to obtain an effective value of γ by specifying the equilibrium exit Mach number and area ratio and computing the corresponding γ. If CONTUR is run with this effective γ value, it will ensure that the final nozzle will have the correct exit area ratio. Similarly, if the equilibrium exit speed is used in Ref. 8, Eq. (82), then an effective gas constant can be computed so that the perfect-gas design will yield the equilibrium exit speed. A more complete description of this idea is given in Ref. 9. Note that this approach does not guarantee that all exit thermodynamic properties will be accurate or that intermediate properties between the stilling chamber and the nozzle exit will be accurate. However, as illustrated in Ref. 9, the results will be better than simply assuming γ=1.4. When such approximations are clearly inappropriate, one can use an MOC design code that will admit arbitrary equilibrium thermodynamics, e.g., that of Johnson and Boney.[5]

2. Arbitrary Exit Cross-Section Shape

A second important MOC extension is the design of nozzles of arbitrary cross section.[10,11] This extension is based on the simple idea that any arbitrary cross-section nozzle can be designed as a subset of a larger axisymmetric nozzle. The desired exit shape is traced out as a closed curve on the axis-normal exit plane of the larger axisymmetric nozzle. Points along this curve define streamlines that can be traced upstream to the throat through the characteristic net, thus defining the contour. Note the remarkable fact that the noncircular nozzle contains an axisymmetric flow field in the inviscid approximation. It is important also to note that even though the test section and throat may be square, for example, the intermediate cross sections will have walls that bow outward slightly. Almost any MOC code would be suitable for the outer axisymmetric design. The present author has created such a design using the Johnson-Boney code.[5] The original idea may have been first published by Beckwith et al.[10] Those authors,in turn, credit a Morton Cooper (apparently unpublished) with the first application of the method in 1947.

3. Geometric Constraints

It is common to specify requirements on geometric parameters, such as nozzle length or viscous exit diameter, that are not available as input variables to the design code. A typical requirement is for a nozzle with a specified length to fit an existing facility with fixed upstream and downstream components. It is also typical to design several nozzles with viscous effects that all have the same exit diameters to permit use of one test section with all nozzles. These situations require manual iteration on CONTUR input variables. If the number of input parameters available for variation happens to match the number of geometric constraints, a multidimensional Newton iteration can be carried out. Otherwise, a least-squares approach may be more appropriate.

4. Cold-Wall Contour Designs

A routine requirement is for a continuous-flow nozzle that must contain a flow at total temperature and pressure at which all known structural materials are liquid or gaseous. An example is an arc-heated facility, where the total temperature might be 10,000°R. For such a nozzle, the wall must be cooled to structurally stable temperatures, probably to less than 2000°R. When the wall is much “colder” than the gas, the boundary-layer displacement thickness may be very small, or even negative. This occurs because of the relatively high gas density at the wall. For such a case, it is typical to make the final contour an inviscid design.

5. Miscellaneous Issues in Applying CONTUR (Tips for Users)

In this section the author wishes to offer new users of CONTUR some suggestions for dealing with practical problems in using this code.

Suggestion 1: First, starting a contour design computation can be difficult. It is easy to guess values of input quantities that will cause the code to fail or to deliver absurd results (such as a nonmonotonic centerline Mach number distribution). The user must specify values for variables (e.g., FMACH or SFOA) for which one may have no physical feel. Suggestion 1 is to start with a previous, successful contour design that is similar to the desired new contour, and to alter the various input parameters to move the design toward the new requirements. One clear rule suggested by Sivells is to choose BMACH $\leq 0.8 * $ CMACH (or CMC). Some release versions of CONTUR include input files for several prior designs.

Suggestion 2: Another issue is choice of distribution of grid points on the centerline and on the various characteristics. Experience indicates that inviscid processing is not sensitive to these distributions. However, the distributions also determine the step sizes used for the boundary-layer calculation, which can become unstable if there are large changes in the marching step. Suggestion 2 is that, after the inviscid design is nearly final, the grid point distributions should be revised to ensure that variation of the wall step size, as defined by the left-running characteristics, varies smoothly from throat to exit. The primary print file contains adequate information to accomplish this refinement.

Suggestion 3: Sivells has advocated in personal communications with the author that the maximum curvature over the contour should be at the throat and that the curvature should decrease monotonically from the throat to the exit. Further, he has advocated using as large a throat radius ratio as possible for flow quality, with the absolute minimum ratio being RC = 1 for accurate machining.

Suggestion 4: A fourth issue is that the final contour design should be checked for smoothness independently of the design code. CONTUR computes distributions of slope and second derivative using splines, but the results of this procedure are sometimes unreliable indicators of adequate smoothness. Post-design smoothing may be necessary. Suggestion 4 is always to recompute two-point slopes from the final coordinate table and always to plot these slopes. This check will make visible any point-to-point slope wiggles. Further, the differences in the adjacent slopes, so computed, should be plotted and examined for wiggles. Also, the slope should increase monotonically from the throat to the inflection point and decrease monotonically from the inflection point to the exit. The slope differences should vary smoothly over the contour.

Suggestion 5: The contour used for these two checks should include the contraction. The contraction should have continuous curvature and should match that of the supersonic contour at the throat. Curvature discontinuities may cause characteristics to coalesce into shock waves and should be avoided. Sivells has recommended a contraction angle of 30 deg or less.

Suggestion 6: The contraction contour should be evaluated for possible (undesirable) flow separation using the Stratford criterion.[12] If separation is indicated, it may be necessary to reduce the contraction angle. Further, sharp corners anywhere on the contour between the stilling chamber and the throat should be avoided. However, to this author’s knowledge, the effect of contraction design on ultimate flow quality has not been fully researched.

Suggestion 7: Another design issue is severe length constraint. Length requirements are sometimes specified without regard to what is physically possible, though there is a minimum length less than which delivery of uniform flow is not possible. As discussed by Argrow and Emanuel,[13] the minimum-length nozzle is the sharp-throat nozzle. However, the sharp throat may be undesirable because of flow quality or heat-transfer considerations. If so, the minimum-length continuous-curvature nozzle from CONTUR can be obtained by choosing a single centerline Mach number distribution (set ETAD = 60 deg, i.e., no radial flow region) with a quintic polynomial. Single cubic or quartic polynomials will also yield relatively short nozzles.

Suggestion 8: If other measures do not yield a short enough nozzle, the nozzle may simply be truncated short of its theoretical length. Experience indicates that most contours contain several inches (depending on nozzle size) near the exit where the radius variation, to machining accuracy, is indistinguishable from a cylinder, and this length may be truncated. The nozzle might even be truncated further upstream with only a small sacrifice in flow quality, but the flow profiles then should be computed at the truncation station from the characteristic net and explicitly examined. For nozzles with a severe length constraint, it is generally better to minimize the theoretical length (e.g., by choosing a single centerline Mach number distribution) than to significantly truncate a longer nozzle. The important point of Suggestion 8 is that MOC design yields a nozzle with a clear theoretical length at which point the inviscid flow is precisely uniform. However, the goal is a practical piece of hardware subject to real-world limitations such as machining accuracy. Truncation to a meaningful length will produce hardware that is cheaper and that may satisfy a length constraint well short of the theoretical length while still providing adequate flow quality.

There are a few processing issues that the user of CONTUR should note. The program is coded in Fortran IV and makes extensive use of the three-branch IF statement. Although the code is well debugged, the coding is often referred to as "spaghetti Fortran." Not surprisingly, a few users have reported compilation difficulties, probably associated with IF equality checks of two real variables resolving differently on different machines. Compilation for 64-bit arithmetic is highly recommended. A compilation option should be set to ensure that subroutine-peculiar variable values are saved for later reentry into the subroutine. There is, of course, no graphical user interface to accompany the code. All processing is batch-style with the primary output on the unit 6 print file. Compute times are typically a few seconds on modern PCs and workstations.

E. Example CONTUR Applications

This section describes, as an example, a family of four Mach number 6 nozzles (Fig. 8) designed for a combustion-heated blowdown facility. The contours were designed for stilling chamber conditions of 666.5 psia (4.595 MPa) and 2865°R (1592 K) and with an exit diameter of 42 in. (1.07 m). The gas model in CONTUR was an effective perfect gas model with γ = 1.34 based on equilibrium thermodynamics and the CEA96 code.[14] The design had a length requirement of 159 in. (4.04 m). As summarized in Table 1, three CONTUR alternatives (Designs 1, 2, and 4) were examined to try to satisfy the length requirement. These three designs all used a single centerline distribution. The three designs differ in throat details and the chosen centerline distribution. An additional design (Design 3) was developed as a "good aerodynamic" design without length constraint as a standard of comparison for flow quality. Two of the short designs (1 and 2) were truncated about 30 in. to meet the length constraint. The final viscous contours were analyzed using a thermochemical nonequilibrium Navier-Stokes solver (DPLR, Data-Parallel Line Relaxation[15] to examine exit flow uniformity. All four designs produced very good and nearly identical flow uniformity in terms of nozzle exit velocity. (The Design 1 profile is shown in Fig. 9.) However, for static pressure profiles over the core region, the designs show distinct differences (Fig. 10). The two heavily truncated nozzles, Designs 1 and 2, clearly have more nonuniformity than Design 4; and the long nozzle, Design 3, is the best. A primary conclusion of this study is that choices made for the inviscid design have a significant effect on the ultimate flow quality.

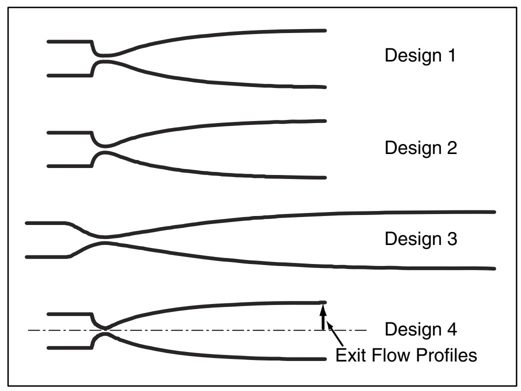

Figure 8. Candidate Viscous Characteristics-Based Contours for Mach Number 6 Nozzle

Table 1 Summary of Mach Number 6 Nozzle MOC-Designed Contours

Nozzle Designation	Centerline Mach Number Distribution	Radius Ratio RC	Inviscid Inflection Angle ETAD (deg)	Theoretical Length, Throat to Exit (in.)
Design 1	quintic	18	19.2	185.9
Design 2	cubic	4	21.4	185.5
Design 3	quartic-radial-quartic	12	9	281.6
Design 4	quintic	4	27.1	160.8

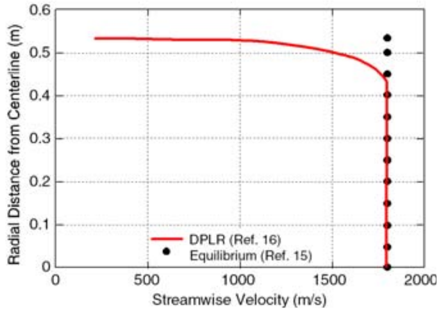

Figure 9. Computed Exit Velocity Profile for Design 1

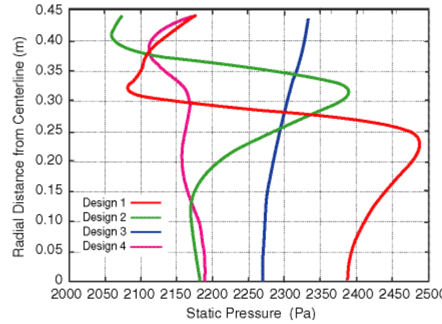

Figure 10. Computed Static Pressure Profiles at Nozzle Exit for Four Designs

III. Design by Analysis – DBA

As hypersonic flow conditions become increasingly severe, eventually the assumption of a perfect gas will become inaccurate; and MOC codes, even those admitting thermochemical equilibrium, will have to be abandoned or, at a minimum, corrected with design procedures admitting more complex thermodynamics. When nozzle flow involves thermochemical nonequilibrium or phase change (e.g., condensation), a fundamental change in the flow physics occurs: multiple definitions of sound speed arise, and Mach number becomes ambiguous. It becomes impossible to characterize the nozzle flow with a single Mach number. Mach number remains well defined for a vehicle moving through the atmosphere, but not for the flow in a nozzle that simulates that flight. The reason for the difference is the source of the energy: for flight, the energy is the kinetic energy of the vehicle, which does not vitiate the free stream ahead of the bow shock; in the nozzle, the energy must be added to the flow, which always vitiates the free stream to some degree. For these reasons, it is necessary to abandon the concept of Mach number to characterize nozzle flow. Mach number ambiguity is a significant issue when choosing flow variables for the objective function of DBA procedures. In the following discussion (as well as above), references to a “Mach number 6 nozzle” are references to a nozzle that simulates flight in the atmosphere at Mach number 6.

DBA methods couple an optimization procedure with a flow solver that computes the flow field through a given contour. The contour is represented by a mathematical function that depends on a series of coefficients, which are the design parameters. The method iteratively alters the contour by perturbing the design parameters to drive some measure of flow nonuniformity (the objective function) toward zero. A typical objective function would be the sum (over grid points on a line across the test section) of the square differences of flow properties from target values. Usually, each perturbation requires a flow-solver solution. At each iteration, the design parameter values are computed with least-squares optimization. Most DBA procedures require computation of a Jacobian matrix, i.e., a matrix of partial derivatives of the components of the objective function with respect to the design parameters. In the most primitive DBA approach, the Jacobian is formed with finite differences between the various flow-solver solutions. More sophisticated DBA methods use a sensitivity equation or adjoint equation to form the Jacobian derivatives as part of the flow solution. Sensitivity equation and adjoint equation methods are much less computationally expensive than primitive methods, but they require very invasive modifications to the flow solver. A direct-design code will usually execute in a minute or less on a personal computer, whereas design by analysis may require hundreds of hours on a parallel supercomputer. The primary advantage of DBA over direct design is that any necessary physics – such as 3-D viscous and reacting flow effects – can be included in the optimization procedure by proper choice of flow solver, while direct-design methods are generally limited to inviscid flow, perhaps with an ad hoc boundary-layer correction, and simpler thermodynamics. A brief survey of DBA nozzle design methods is given in Ref. 9.

The first documented DBA nozzle design method was reported by Korte et al.[16] For this technique, the objective function included not only the test rhombus Mach number, but also the centerline Mach number distribution for which target values were determined by Sivells’ method of polynomials. The design parameters were the slope coefficients of the spline fit to the contour rather than the coordinates of the contour. Use of the slopes was said to greatly improve the convergence. The flow solver was an efficient parabolized Navier-Stokes solver. The solver initially assumed perfect-gas thermodynamics, but later versions included certain high-pressure effects. The Korte procedure is named CAN-DO, CFD-based

Aerodynamic Nozzle Design and Optimization. It has been used successfully in several nozzle design and redesign efforts.

A DBA procedure admitting reacting nonequilibrium thermodynamics is reported in Refs. 9 and 17. In this procedure, the contour is first designed with CONTUR including the boundary-layer correction. Next, the MOC contour is optimized using the DPLR solver to compute the objective function, where the design parameters are selected input variables to CONTUR. The chosen objective function is composed of the static pressure and density and the two velocity components. (Note the absence of Mach number.) The optimized MOC contour is then further altered with a correction distribution represented with cubic splines using a suitably small number of nodes along the nozzle. The corrections at the nodes are the design parameters. A formal least-squares optimization procedure is used to select the values of the design parameters that minimize the flow nonuniformity at the nozzle exit.

The DBA procedure of Ref. 17 was used to further optimize the Mach number 6, Design 4 contour shown in Fig. 8. The static pressure nonuniformity of four sequential optimization steps is shown in Fig. 11. The vertical scale in Fig. 11 was chosen as the radial grid point index to avoid obfuscating results where grid points were clustered. The Design 4 profile is for the unoptimized MOC contour including the boundary-layer correction. Design 9 is the MOC contour optimized for reacting flow effects, though the Design 9 contour is entirely defined by the MOC code. Design AI is the final optimized contour controlled by the least-squares optimization. The optimization process reduced the flow nonuniformity from about ±2 percent to ±0.5 percent. The objective function history is shown in Fig. 12. The objective function was reduced to about 26 percent of the value for the unoptimized MOC contour.

The above optimization process required 80 flow-solver solutions, including several failed attempts to reduce the objective function. These 80 solutions consumed approximately 200 serial processor hours on an HP 9000 Model 800 using four parallel processors. Of these 80 solutions, 27 successfully reduced the objective function and consumed approximately 70 serial processor hours. A portion of the procedure is automated with a Unix-like script, but the procedure is not a numerically well-behaved, fully automated process that will design a contour without significant user involvement. Details of the design procedure are documented in Ref. 9.

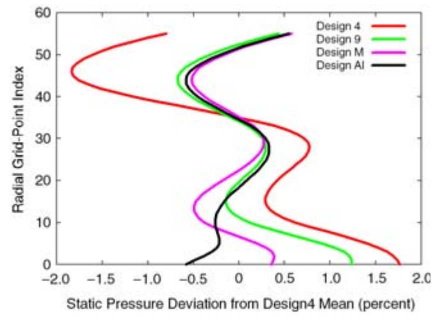

Figure 11. Computed Static Pressure Profiles for Design Optimization

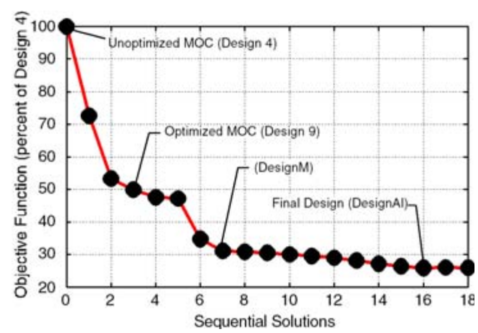

Figure 12. Objective Function History Optimization

IV. Summary

A brief survey of operational techniques to design nozzle contours for supersonic and hypersonic wind tunnels has been presented. The two primary approaches are direct design, based on the classical method of characteristics and design-by-analysis, where an optimization procedure is coupled with a CFD-based analysis code. Some previously unpublished application details on the use of the direct-design code by Sivells also have been presented. The primary foregoing observations on wind tunnel nozzle design include the following:

- For hypersonic nozzles, axisymmetric designs are usually chosen over the 2-D option because of throat-cooling issues and thermal/dimensional stability of the throat.
- An aerodynamically rigorous contour design for a nozzle of arbitrary exit cross-sectional shape can be achieved by extracting the desired shape from a flow field for a larger axisymmetric design.
- The nozzle contour should not be expected to clean up a nonuniform flow that originates upstream.
- Long nozzles with small inflection angles appear to offer the best flow quality.
- Extremely short nozzles may yield poorer flow quality, but flow uniformity can be improved with proper choices during the design and with formal optimization.

- Short nozzles are preferable for high-enthalpy facilities because they lose less flow energy through wall heat transfer.
- All direct-design methods assume some initial flow profile in the vicinity of the throat and tacitly assume that the contraction contour will deliver that profile. The contraction contour is often designed without guaranteeing that initial flow profile.
- Most hypersonic facility flows involve thermodynamics that are more complex than the perfect-gas model. Popular direct-design codes in which the perfect-gas assumption is deeply embedded may be extended slightly with proper choice of an effective perfect-gas model.
- Direct design based on method of characteristics remains viable as an initialization procedure for design-by-analysis methods.
- The primary advantage of design by analysis is that the effect of any important physics can be included in the design by proper choice of the flow solver.
- In a hypersonic nozzle flow, Mach number may be an ill-defined parameter.

Acknowledgements

The author wishes to acknowledge and thank the Peer Reviewers, Dr. Stan Powell and Mr. Bill Milam, Aerospace Testing Alliance, for their careful review and helpful suggestions.

References

[1]Anderson, J. D., "A History of Aerodynamics and Its Impact on Flying Machines," Cambridge University Press, 1997.

[2]Liepmann, H. W., and Roshko, A., "Elements of Gasdynamics," John Wiley & Sons, Inc., New York, 1966.

[3]Sivells, J. C., "A Computer Program for the Aerodynamic Design of Axisymmetric and Planar Nozzles for Supersonic and Hypersonic Wind Tunnels," AEDC-TR-78-63, Dec. 1978.

[4]Sivells, J. C., "Aerodynamic Design of Axisymmetric Hypersonic Wind Tunnel Nozzles," *AIAA Journal of Spacecraft and Rockets*, Vol. 7, No. 11, Nov. 1970, pp.1292-1299.

[5]Johnson, C. B., and Boney, L. R., "A Method for Calculating a Real-Gas Two-Dimensional Nozzle Contour Including the Effects of Gamma," NASA TM X-3243, Sep. 1975.

[6]Hall, I. M., "Transonic Flow in Two-dimension and Axially-Symmetric Nozzles," *The Quarterly Journal of Mechanics and Applied Mathematics*, Vol. 15, Pt. 4, Nov. 1962, pp. 487-508.

[7]Kliegel, J. R., and Levine, J. N., "Transonic Flow in Small Throat Radius of Curvature Nozzles," *AIAA Journal*, Vol. 7, No. 7, July 1969, pp. 1375-1378.

[8]Ames Research Staff, "Equations, Tables, and Charts for Compressible Flow," NACA Report 1135, 1953.

[9]Shope, F. L., "Design Optimization of Hypersonic Test Facility Nozzle Contours Using Splined Corrections," AEDC-TR-04-2, Mar. 2005.

[10]Beckwith, I. E., Ridyard, H. W., and Cromer, N., "The Aerodynamic Design of High Mach Number Nozzles Utilizing Axisymmetric Flow with Application to a Nozzle of Square Test Section," NACA TN 2711, 1952.

[11]Haddad, A., and Moss, J. B., "Aerodynamic Design for Supersonic Nozzles of Arbitrary Cross Section," *J. Propulsion and Power* (AIAA), Nov.-Dec. 1990, Vol. 6, No. 4, pp. 740-746.

[12]Stratford, B. S., "The Prediction of Separation of the Turbulent Boundary Layer," *Journal of Fluid Mechanics*, Vol. 5, 1959, pp. 1-16.

[13]Argrow, B. M., and Emanuel, G., "Comparison of Minimum Length Nozzles," *ASME Journal of Fluids Engineering*, Vol. 110, Sep. 1988, pp. 283-288.

[14]Gordon, Sanford, and McBride, Bonnie J., "Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications, I. Analysis, II. Users Manual and Program Description," NASA Reference Publication 1311, June 1996.

[15]Candler, G. V., "Hypersonic Nozzle Analysis Using an Excluded Volume Equation of State," AIAA-2005-5202, 38th AIAA Thermophysics Conference, Toronto, Ontario, Canada, 6-9 June 2005.

[16]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-92-4009, AIAA 17th Aerospace Ground Testing Conference, Nashville, TN, July 1992 (A92-56832).

[17]Shope, F. L., and Tatum, K. E., "Reacting Navier-Stokes Nozzle Contour Design Optimization," AIAA-2005-5081, 23rd AIAA Applied Aerodynamics Conference, 6-9 June 2005, Toronto, Ontario, Canada.