Design and analysis of a rectangular cross-section hypersonic nozzle

R.S.M. Chue[1], **D. Cresci**[1], **and P. Montgomery**[2]

[1] *ATK GASL, 77 Raynor Avenue, Ronkonkoma, NY 11779, USA*
[2] *Aerospace Testing Alliance/AEDC, Arnold AFB, TN 37389, USA*

Summary. A technique for designing rectangular cross-section hypersonic nozzles where all four walls are contoured is presented. Two design variations are shown; one follows directly from a streamtracing method while the other is reshaped to enforce rectangular cross-sections. The former has slightly bulging walls while the latter is easier to manufacture. The application of the true-streamtraced nozzle when integrated with a Flight Acceleration Simulation Test (FAST) technique is also studied. In a FAST test, the simulated flight Mach number is varied by using a reflector plate downstream of the nozzle to generate either an oblique shock or an expansion fan. FAST requires a nozzle with rectangular exit, and therefore the current design method is particularly applicable. The flow quality of the rectangular nozzles as well as the effects of the reflector plate will be discussed.

1 Introduction

It is sometimes desirable to have a rectangular exit nozzle in a hypersonic ground-test facility or in a flight vehicle. Rectangular nozzles could be designed as two-dimensional nozzles where the area expands by having only two of the four walls contoured. However, since the expansion is two dimensional, the design would result in nozzles that are longer than round ones. In addition, when operating in a high temperature environment, such as that in a vitiation-heated test facility, the thin-slit nozzle throat that often results in two-dimensional hypersonic nozzles could lead to severe heating and structural deformation problems. An alternative approach to designing a rectangular exit hypersonic nozzle would be to have all four walls contoured so that the expansion is three dimensional.

This paper serves to demonstrate a technique for designing rectangular hypersonic nozzles that have all four walls contoured using a streamline tracing method. As an illustration, the design of a large-scale hypersonic nozzle with a 1.016 m × 1.778 m (40 in. × 70 in.) rectangular cross-section at the exit is shown. Two design variations can be produced from the tools developed at ATK GASL; one is generated using a streamtracing method, and another is further reshaped so that the cross-sectional shape is forced to be exactly rectangular at all axial locations.

As an example for the application of rectangular hypersonic nozzles, a numerical simulation of the true-streamtraced nozzle when integrated with a Flight Acceleration Simulation Test (FAST) configuration is also included in the study. In the FAST test operation, a test technique previously developed at ATK GASL [1], a reflector plate is placed downstream of the nozzle and the plate angle relative to the nozzle is varied during testing as a method to simulate the varying Mach number conditions along a flight trajectory. Note that a rectangular nozzle exit is preferred over a round one for this operation. If a round nozzle is used, the interaction of the circular nozzle flow, which includes the boundary-layer nonuniformity from the nozzle, with the reflector plate would result in a highly three-dimensional flow and would severely affect the test flow quality.

The flow simulations performed include a positive plate rotation to generate an oblique shock to decrease the test flow Mach number, and also a reverse rotation to generate expansion waves and a higher flow Mach number.

2 Rectangular nozzle design

The underlying concept of the design is to use a "parent" nozzle flow field that produces a high-quality nozzle flow while resulting in a compact (i.e., short length) nozzle. An axisymmetric nozzle designed using an analytical method is best suited as the parent flow field. From this, an arbitrarily shaped streamtube would then be "carved" out to obtain the surfaces of any desired nozzle geometry, which in the present case is rectangular. The streamtube is constructed using a streamline tracing procedure where streamlines that would result in a rectangular exit are traced along the axisymmetric flow field. Since no flow can cross a streamline, the carved-out flow field behaves as though the nozzle were still circular. The axisymmetric nozzle design and the streamtracing procedure are applied to an inviscid flow, and the obtained streamlines are corrected for displacement thickness effects as a final step.

The design of a rectangular exit cross-section nozzle can be summarized in the following steps:

(a). An axisymmetric nozzle is designed with an exit diameter that encompasses the desired exit cross section of 1.016 m × 1.778 m (40 in. × 70 in.) This is done using ATK GASL's in-house inviscid method-of-characteristics code (*noz_code*), which generates the supersonic section nozzle wall contour.
(b). An inviscid axisymmetric CFD solution is obtained for the round nozzle geometry.
(c). The desired cross-sectional shape of the nozzle exit is prescribed on the CFD solution. The edges on the nozzle exit are discretized into points. To take advantage of symmetry, investigators need only one quarter of the nozzle geometry in the streamtracing and subsequent CFD analysis.
(d). Streamlines are traced through the inviscid axisymmetric flow field from the nozzle exit discretized points toward the upstream direction and end at the throat location. Note that since the parent flow field is axially symmetric, the CFD solution at any axial location is a function of the radius, but not of the angular orientation. When performing streamtracing, the angular orientation is therefore unchanged along each streamline although this orientation must be preserved in order to reconstruct the three-dimensional position of the final streamline.
(e). Each of the streamlines is corrected for the displacement thickness using ATK GASL's in-house *Sasman-Cresci* code [2].
(f). The true-streamtraced nozzle is then constructed using the family of viscous-corrected streamlines that form the side and top surfaces.
(g). To obtain a nozzle with fully rectangular cross-sections throughout the length, investigators further discretize the nozzle into small, axial increments. At each axial station, the shape is reformed into a rectangle where the cross-sectional area is preserved and the aspect-ratio is fixed to be the same as that at the nozzle exit.

To demonstrate the applicability of the technique for hypersonic nozzles, a design example is presented for a nozzle with a stagnation pressure and temperature of 19.3 MPa and 2610 K, respectively, and a nozzle exit Mach number of 6.25.

The designed nozzles have at least approximately rectangular cross-sections. This is exactly so for the all-rectangular nozzle, since it is enforced to have rectangular sections, but it is approximately true for the pure streamtraced nozzle as well. Furthermore, the throat and the nozzle exit are very close to rectangles even for the true-streamtraced case.

The main difference between the true streamtraced nozzle and the all-rectangular nozzle is that the walls of the former have concave and convex surfaces. This feature is a consequence of the inviscid streamline trajectories rather than of the viscous effects. The true streamtraced nozzle geometry is shown in Fig. 1 with a zoom-in view where the walls can be seen to bulge in and out at different locations. The geometry of the all-rectangular nozzle is shown in Fig. 2.

Fig. 1. Geometry of the true streamtraced nozzle. Left: Full view, right: zoom-in view

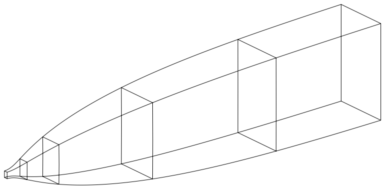


Fig. 2. Geometry of the all-rectangular nozzle

Since the true streamtraced nozzle is directly obtained from the exact streamtube of the parent axisymmetric nozzle, it is expected to adhere more accurately to the theoretical wave-cancelling feature of the method-of-characteristics design. However, the all-rectangular nozzle may be easier and less expensive to manufacture. In the following section, a CFD analysis is carried out to examine the flow quality of the nozzles to determine the best performing design.

3 Computational analysis of designed nozzles

The computational domain includes only the supersonic contours and starts at the nozzle throat plane, where a uniform sonic condition is assumed. The nozzle entrance conditions are obtained using a one-dimensional chemical equilibrium code. The fluid medium is vitiated air consisting of the combustion products from the vitiation heater. High temperature gas dynamics and finite-rate chemistry are captured using a 16-reaction, 12-species kinetics model. The boundary layer is assumed to be fully turbulent, and the two-equation Mentor Shear Stress Transport (SST) k-ω turbulence model is used. The computational code used is GASP version 4.2 [3].

The CFD solution for the true streamtraced nozzle at the design condition is shown in Fig. 3. In the figure, the Mach number contours are shown to illustrate the boundary layer thicknesses on the nozzle walls. Fig. 3a plots the contours on the $z = 0$ symmetry plane, which shows that the boundary layer thickening eventually decreases the core flow

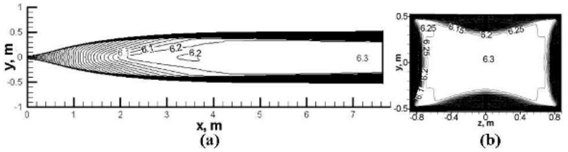


Fig. 3. Mach number contours for true streamtraced nozzle. (**a**) On $z = 0$ symmetry plane, (**b**) at nozzle exit cross section

size near the nozzle exit. Fig. 3b plots the Mach number contours at the nozzle exit and shows a clean flow area of 0.76 m × 1.52 m (30 in. × 60 in.) where the Mach number is very close to the design value of 6.25.

The Mach number contours for the all-rectangular nozzle are shown in Fig. 4. Fig. 4a shows the contours on the $z = 0$ symmetry plane where the boundary layer thickness is seen to be more severe than for the true-streamtraced nozzle and the core flow size is decreased far more than in the true streamtraced nozzle. Fig. 4b plots the Mach number contours at the nozzle exit. Compared to the true streamtraced nozzle (Fig. 3b), the exit flow deviates more from the design Mach number of 6.25 and is more nonuniform. The most noticeable nonuniformities are two round-shaped, low Mach number regions at around $z = 0$ on the y_{max} and y_{min} walls, which decreased the core flow size in the y direction to about 0.508 m (20 in.). Since the only difference between the true streamtraced and the all-rectangular nozzles is that the former follows directly from the method-of-characteristics wave cancelling technique while the latter is reshaped to enforce rectangular cross sections, the poor performance of the latter is a consequence of reshaping that renders the wave mechanics inaccurate. The CFD results therefore demonstrate that the true-streamtraced nozzle can produce better flow quality than the all-rectangular nozzle.

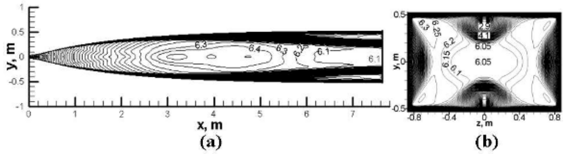


Fig. 4. Mach number contours for all-rectangular nozzle. (**a**) On $z = 0$ symmetry plane, (**b**) at nozzle exit cross section

4 Application of rectangular nozzle to FAST test operation

The Flight Acceleration Simulation Test (FAST) is a ground testing technique previously developed at ATK GASL [1] to simulate the variation of flight conditions along a trajectory in real time. It is used as an example for the application of rectangular exit nozzles. The basic principle behind the FAST technique is that the Mach number in a two-dimensional flow field can be varied by a simple wave process: a shock wave to reduce the Mach number, or an expansion fan to increase it. This process is accomplished by rotating a flat plate (the "reflector plate"), on which the test article is mounted, relative to the nozzle exit. The technique produces the correct flow conditions over the test article by simultaneously varying the test-section Mach number through the strength of the shock wave or expansion fan and the stagnation conditions by varying the pressure and the fuel-air ratio in the vitiation heater. In this manner, a selected segment of a flight trajectory – including both speed and altitude – can be produced in real time by preprogramming the facility operation [1].

The true streamtraced nozzle is selected for this analysis as its flow quality is superior to that of the fully rectangular nozzle. The nozzle is truncated at 5.4 m (212.56 in.) from the throat location to maximize the core flow size. A reflector plate is placed downstream of the nozzle. The plate is displaced 0.127 m (5 in.) normal from the nozzle wall and is translated 0.0635 m (2.5 in.) axially from the nozzle exit. A "jet stretcher" is assumed to extend the wall contour of the nozzle exit to minimize the disturbances from the edge of the nozzle exit. The jet stretcher walls are taken to be that of the untruncated nozzle contour so that the walls are perfectly continuous between the nozzle exit and the jet stretcher. Beyond the length of the original untruncated nozzle, the top (z) surface of the jet stretcher is open (no wall), whereas the side (y) surfaces are assumed to have fences throughout. The side fences are assumed to have the same wall tangency angles as those at the untruncated nozzle exit.

The nozzle operated with a 10° compression reflector plate is used to generate a test flow that simulates a flight Mach number that is lower than that for the nozzle design condition. The test section is to be located where the oblique shock generated by the reflector merges with the expansion waves generated from the opened top surface. At this location, the flow underneath the oblique shock has the largest core size and is then used as the test flow. The flow field on the $y = 0$ symmetry plane is shown in Fig. 5a.

The flow distributions on a transverse slice taken at where the oblique shock and the top expansion waves merge are shown in Fig. 5b. The useful core flow is in the lower center region, which has a Mach number of approximately 5.5. In the slice, there are two low Mach number distorted flow regions on the side (y_{max}, y_{min}) walls, which are generated by the interaction between the oblique shock and the boundary layers inherited from the nozzle. The distorted regions significantly reduced the core flow width to be about 0.41 m (16 in.) in the y direction. In the z direction, the core flow height based on the flow nonuniformity immediately above the reflector surface is about 0.38 m (15 in.)

The 10° expansion case is similar to the compression case but with the reflector plate tilted downward to generate expansion waves to simulate a flight Mach number that is higher than that for the nozzle design condition. The test section is to be located approximately where the expansion waves generated by the plate merge with the location, the core size is the largest and is used as the test flow. The flow field on the $y = 0$ symmetry plane is shown in Fig. 6a.

The flow distributions on a transverse slice taken near the end of the reflector plate are shown in Fig. 6b. The useful core flow is represented by the lower center region, which has a Mach number of approximately 7.6. Unlike the compression case, there is no shock wave generated on the plate and thus no shock interaction with the inherited viscous flow. As a result, the viscous flow nonuniformity on the y_{max} and y_{min} walls do not

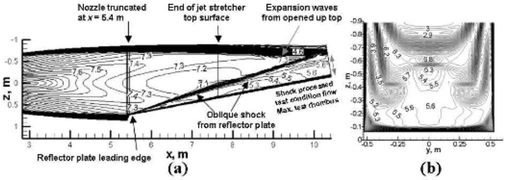
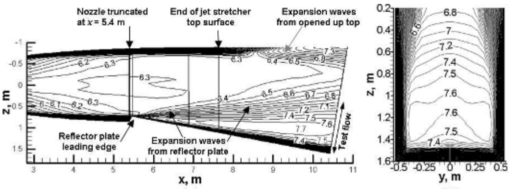


Fig. 5. Mach number contours over reflector plate, 10° compression. (**a**) On $y = 0$ symmetry plane, (**b**) $x \simeq 4.45$ m (175 in.) from nozzle exit



(a) (b)

Fig. 6. Mach number contours over reflector plate, 10° expansion. (**a**) On $y = 0$ symmetry plane, (**b**) $x \simeq 5.16$ m (203 in.) from nozzle exit

show any sign of distortion or significant growth; therefore, the flow uniformity is largely preserved. The core flow size is also found to be relatively insensitive to the location of the transverse slice. The high Mach number region is seen to occur just outside the wall boundary layer over the plate, and it gradually decreases in the z direction away from the plate. However, the variation in Mach number is weak and a core flow size of approximately 0.91 m × 0.76 m (36 in. × 30 in. ($y \times z$)) can be obtained.

5 Concluding remarks

A technique is presented for designing rectangular exit nozzles using a streamtracing method. Two nozzle variations are produced, one adheres to the streamtracing procedure and the other is further enforced to be rectangular. Both nozzles have very similar shapes, although the true-streamtraced nozzle has slightly bulging walls. However, the flow qualities delivered by the two nozzles are significantly different from each other. The all-rectangular nozzle suffers from larger boundary layer growth and flow distortions, and thus the core flow size is severely reduced. The true-streamtraced nozzle is therefore a superior design.

The true-streamtraced nozzle is also applied to the FAST facility operation where a reflector plate is added to the nozzle exit. The analysis for the plate rotated 10° into the flow to generate a shock showed a large decrease in core flow area due to the interaction between the shock and the viscous flow inherited from the nozzle. On the other hand, the 10° expansion operation yielded a more uniform flow. This suggests that the test facility should be configured with a nozzle that is designed below the mid-value in the operating range so as to take advantage of the better uniformity in the expansion operations.

Acknowledgement. The authors would like to thank the Test Resource Management Center (TRMC) Test and Evaluation/Science and Technology (T&E/S&T) Program for their support. This work is funded by the T&E/S&T Program through the Arnold Engineering Development Center, Arnold Air Force Base, TN.

References

1. Engers R., Erdos J., Roffe G., Swartwout W.: Flight testing on the ground in the *FAST* facility. ISABE Paper 99-7170, 14th ISABE (1999)
2. Sasman P.K., Cresci R.J.: Compressible turbulent boundary layer with pressure gradient and heat transfer. AIAA J., **4**, 11, Jan. (1966)
3. GASP version 4.2, Aerosoft, Inc, Blacksburg, Virginia, USA (2002)

Approved for public release; distribution is unlimited