Development and verification of a supersonic nozzle with a rectangular cross section at a Mach number of 2.8 for a scramjet model combustor

Tatsuya YAMAGUCHI*, Tomohiro HIZAWA*, Taro ICHIKAWA*, Taku KUDO*, Akihiro HAYAKAWA* and Hideaki KOBAYASHI*
* Institute of Fluid Science, Tohoku University
2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577, Japan
E-mail: yamaguchi@flame.ifs.tohoku.ac.jp

Received: 2 September 2018; Revised: 17 October 2018; Accepted: 5 November 2018

Abstract
Development of a supersonic nozzle with a rectangular cross section for a scramjet model combustor was performed and verified experimentally and numerically. The newly-proposed combined design methodology is easier to use than the other design methodologies when the nozzle exit geometry is the constraint condition because it can get information about the displacement thickness before determining the supersonic nozzle contour. The velocity profile of the numerical simulation and that of LDV measurements are quantitatively in good agreement, which meant that the numerical simulations performed in the present study could quantitatively evaluate the flowfield in the supersonic nozzle and the isolator developed in the present study. The designed supersonic nozzle showed good uniformity of the Mach number and stream angle in the mainstream region at the nozzle exit. Also, the average Mach number in the mainstream region at the nozzle exit plane showed very good agreement with the design Mach number. The influence of airstream total temperature on the Mach number distribution and stream angle distribution in the mainstream region was investigated numerically and was acceptably small, which means that the supersonic nozzle designed in this study can create approximately equivalent supersonic flow in the range of airstream total temperature 300 K - 800 K. Additionally, The mainstream region was found to have a trapezoidal shape with the upper side as the short side and the lower side as the long side. This trapezoidal shape is considered to be caused by the pressure gradient along the side wall, indicating that it is difficult to prevent the mainstream region from growing to trapezoidal shape in case of two dimensional supersonic nozzle with a single contour wall.

Keywords: Scramjet engine, Supersonic combustion wind tunnel, Supersonic nozzle, Convergent–divergent nozzle, De Laval nozzle, Design methodology, Laser Doppler velocimetry, Computational fluid dynamics

1. Introduction

The scramjet engine, which is a type of air-breathing engine, is a promising propulsion system for hypersonic transportation. The combined use of rocket and scramjet engines is expected to be an effective way to reduce the cost of space transportation in the future. Because the velocity of the airstream passing through a scramjet combustor is greater than the speed of sound, the airstream residence time is only a few milliseconds, which is much shorter than the time required for combustion. This means that the airstream residence time must be extended adequately and efficiently to achieve flameholding in the scramjet combustor.

Many researchers have recently investigated the use of cavity flameholders to achieve flameholding in a supersonic airstream (Rasmussen et al., 2005, Hongo et al., 2013) because they create a low-speed recirculation zone in the recess, which can easily extend the residence time of entrained air without a large total pressure loss. Yamaguchi et al. (2016) proposed a cavity flameholder with a single injector at the bottom that supplies hydrogen-rich burned gas directly into the cavity flameholder for forced ignition. This burned-gas injection can provide fuel at a high temperature and adjust
Combustion experiments in the semi-freejet supersonic combustion wind tunnel have shown that there are two characteristic combustion modes: the jet-plume mode and the cavity mode. (Yamaguchi et al., 2016). Figure 1 shows direct photographs of the two combustion modes observed in combustion experiments in the semi-freejet supersonic combustion test facility. The jet-plume mode, which is characterized by a luminous region only along the jet plume, is considered not to achieve flameholding. The cavity mode is more preferable than the jet-plume mode because it is characterized by a large luminous region in the recirculation zone at the jet side and in the jet wake. Because the cavity mode can be achieved by increasing the enthalpy flow rate of the burned-gas jet, the combination of the burned-gas injection and the cavity flameholder with a single injector at the bottom are considered to have good compatibility in the supersonic flow at low total temperature. The cavity flameholder with a single injector at the bottom showed very good flameholding performance in the semi-freejet supersonic combustion test facility. However, it is not clear whether the cavity flameholder with a single injector at the bottom is also effective in the scramjet combustor, where the supersonic airstream has a relatively thick boundary layer.

The semi-freejet supersonic combustion test facility at the Institute of the Fluid Science was developed by Niioka et al. (1992). Because the semi-freejet test facility was developed for fundamental experiments to investigate mixing, ignition, and combustion phenomena, it is difficult to investigate the phenomena that occur inside the scramjet combustor, such as operation mode transition, thermal choking, pseudo shockwave, and boundary layer interaction. To investigate the combustion characteristics of the burned-gas injection cavity flameholder in a scramjet model combustor, a direct-connect supersonic combustion test facility, which can simulate the airstream inside the scramjet combustor, is required. In this study, a direct-connect supersonic combustion test facility was newly developed. Additionally, a supersonic nozzle, which is the most important component in a supersonic wind tunnel, was developed using a newly proposed design methodology combining an approach to the design of the supersonic nozzle contour and a method for correcting the boundary layer effect. The objective of this study was to experimentally and numerically verify the performance of the newly proposed design methodology and the supersonic nozzle developed by applying this methodology.

2. Concept of direct-connect supersonic combustion test facility

Figure 2 shows a schematic of the direct-connect supersonic combustion test facility newly developed in this study. The facility is a blowdown supersonic wind tunnel consisting of an air tank, an air heater, a stilling chamber, a supersonic nozzle, a test section, a supersonic diffuser, and a silencer. The supersonic nozzle and the test section were newly developed in this study. The other parts are the same as those used in the semi-freejet supersonic combustion test facility at the Institute of Fluid Science, Tohoku University (Niioka et al., 1992). The cross-sectional area of the supersonic nozzle throat was reduced to approximately 1/6 that of the conventional supersonic nozzle in the semi-freejet test facility. This reduced the air consumption rate and extended the maximum test duration to approximately 120 s. In preparation for the combustion experiments, the air was dried by an air dryer, pressurized by an air compressor, and stored in the air tank at 4.5 MPa prior to the experiments. Additionally, the air heater, which was filled with approximately 240,000 ceramic balls.
of approximately 13 mm in diameter, was heated with a propane/air gas burner in advance. The maximum temperature of the air heater exit was 870 K. The total pressure and temperature of the airstream were measured in the stilling chamber. The airstream total pressure was controlled by the main regulating valve, which was installed just downstream of the air tank. However, it is not easy to control the airstream total temperature because it strongly depends on the amount of heat remaining inside the air heater.

Figure 3 shows a cross-sectional view of the supersonic nozzle and the test section. To facilitate methods of optical observation and measurement, such as schlieren photography, laser Doppler velocimetry (LDV), particle image velocimetry, and laser-induced fluorescence, the supersonic nozzle and test section were designed so that the flow path had a rectangular cross section. As shown in Fig. 3, because the supersonic nozzle was inserted into the stilling chamber, it was possible to prevent the boundary layer from influencing the upstream side of the airstream in the supersonic nozzle. Also, since the temperature difference between the inside and the outside of the supersonic nozzle which is inserted into the stilling chamber is considered small compared to the common supersonic nozzle which is placed in the atmosphere, it was assumed that the wall of the supersonic nozzle is close to adiabatic wall. The length, width, and height of the supersonic nozzle are 300, 90, and 30 mm, respectively. The details of the supersonic nozzle design methodology will be discussed in Section 3.

The test section consists of an isolator, an expansion duct, and a constant-area duct. The length, width, and height of the isolator are 200, 90, and 30 mm, respectively. The top wall of the expansion duct is inclined at an angle of 2.5° with respect to the horizontal. The expansion duct prevents thermal choking by suppressing the decrease in the Mach number caused by heat release in the supersonic flow. It also has quartz glass windows in one of its side walls for observation and in the top wall for laser incidence. There is a burned-gas injection type cavity flameholder in the expansion duct 30 mm downstream from the duct entrance, which is also the isolator exit. The depth, length, width, and aft ramp angle of the cavity flameholder are 10 mm, 22.07 mm, 90 mm, and 22.5°, respectively. The length of the cavity flameholder was determined using the definition proposed by Gruber et al. (1999). The cavity flameholder has a single injection hole at its bottom. The distance between cavity leading wall and the center axis of the injection hole is 2.5 mm. The burned gas is supplied to the cavity flameholder by the hydrogen/air burned gas torch igniter developed by Yamaguchi et al. (2018). This igniter can control the injection gas temperature by generating burned gas at equivalence.
ratios ranging from 1.0 to 10.0. The diameter of the injection hole is 3.0 mm. The length, width, and height of the constant-area duct are 389, 90, and 45 mm, respectively. Although Fig. 3 includes expansion duct with a cavity flameholder, which corresponds to a combustor, the influence of the combustion phenomena is not discussed in the present study since the interaction among the supersonic nozzle, the isolator and the combustor is not strong except when the back pressure is high. In the following chapters, only the supersonic nozzle and the isolator will be considered.

3. Design methodology of two-dimensional supersonic nozzle
3.1 Design approach based on a combination of previous methods

The design of the supersonic nozzle contour is the most important step in the development of the supersonic wind tunnel. In the case of a quasi-one-dimensional flow, the airstream Mach number at the nozzle exit is simply determined by the ratio of the cross-sectional area at the nozzle exit to the cross-sectional area at the nozzle throat. However, in an actual nozzle, such as a two-dimensional supersonic nozzle or an axisymmetric supersonic nozzle, the nozzle contour must have a shape that counteracts the expansion wave caused by the acceleration of the supersonic flow to generate a uniform supersonic flow at the nozzle exit. Additionally, the supersonic nozzle contour should account for the displacement thickness of the boundary layer that develops along the nozzle wall to prevent the airstream Mach number at the nozzle exit from decreasing. The design methodology developed in this study consisted of a combined approach of nozzle contour design and boundary layer correction based on methods proposed in previous studies.

Two-dimensional supersonic nozzles can be classified into symmetric and asymmetric nozzles. Symmetric supersonic nozzles consist of two flat surfaces and two symmetric contour surfaces. Although symmetric supersonic nozzles are expected to have a symmetric flowfield, they have a higher production cost than asymmetric nozzles because it is technically difficult to machine two equivalent contour surfaces with high accuracy. On the other hand, asymmetric supersonic nozzles require only one contour surface and three flat surfaces, so it is relatively easier to accurately machine the necessary nozzle contours. Moreover, it is possible to easily change the airstream Mach number by changing only one contour surface (Yoshida and Tsuji, 1977). Therefore, an asymmetric two-dimensional supersonic nozzle design was considered in this study. There are various design methodologies for supersonic nozzle contours. Prandtl and Buseman (1929) first proposed a graphical solution called the Prandtl–Buseman method based on the method of characteristics. Later, Foelsch (1946) proposed an analytical solution based on the Prandtl–Buseman method; this method, which is superior in terms of simplicity and ease of application, was used in the present study. Other previously proposed design methodologies have been compiled in a bibliography by Wolf (1990).

Boundary layer correction is also important in supersonic nozzle design because the effective cross-sectional area of the airstream is reduced by the development of the boundary layer. Tucker (1950) proposed a well-known method of estimating the boundary layer thickness. Miyazato et al. (1994) have experimentally verified this method and shown that it can be used to easily obtain the boundary layer thickness in a flow field with a zero or forward pressure gradient by selecting an appropriate velocity distribution and coefficient of friction. Although Tucker’s method is an excellent method for boundary layer thickness estimation, another estimation method given by Burke (1961), which is superior in terms of simplicity and ease of application, was applied in this study. The equation proposed by Burke (1961) has been applied by Fiore et al. (1975) for the development of the Aerospace Research Laboratories (ARL) Mach 3 high Reynolds number facility and by Gruber and Najed (1994) for the development of a large-scale supersonic combustion test facility. The details of the method developed by Burke (1961) will be presented in Section 3.2. Other previously proposed estimation methodologies have been compiled by Stratford and Beavers (1961).

3.2 Design of the supersonic nozzle contour and correction of the boundary layer effect

In this study, the supersonic nozzle design procedure was divided into four stages: (1) the estimation of the displacement thickness for boundary layer correction, (2) the design of the contour of the initial expansion section, (3) the design of the contour of the terminal section to create a uniform supersonic flow by neutralizing expansion waves, and (4) the design of the contour of the convergence section.

The estimation of the displacement thickness for boundary layer correction was conducted using Burke’s (1961) method under the following four assumptions. (1) The displacement thickness at the nozzle exit can be calculated using the equation proposed by Burke (1961). (2) The boundary layer develops from the nozzle throat. (3) The displacement
thickness at the nozzle throat is negligibly small. (4) The displacement thickness increases linearly with axial distance from the nozzle throat.

The displacement thickness at the nozzle exit $\delta_e^*$ is given by the following equation, as proposed by Burke (1961):

$$\delta_e^* = 0.0463 \frac{M_e^{1.311}}{Re_e^{0.276}} x_e,$$

(1)

where $x_e$ is the length of the divergence section, which is the distance between the nozzle throat and nozzle exit; $M_e$ is the design Mach number; and $Re_e$ is the Reynolds number at the exit of a supersonic nozzle of characteristic length $x_e$. Therefore, the correction height $y_c$ at the nozzle exit is given by

$$y_c = \frac{2\delta_e^*(W+H)-4\delta_e^{*2}}{x_eW} x,$$

(2)

where $W$ is the width of the supersonic nozzle and $H$ is the height of the nozzle at the exit. The contour of the initial expansion section is determined by the following quartic function:

$$y = \left[3(y_1 - y_{th}) - 2x_1 \tan \theta_l \right]^4 + \left[\frac{-8(y_1 - y_{th}) + 5x_1 \tan \theta_l}{x_1^3} \right]^3 + \left[\frac{6(y_1 - y_{th}) - 3x_1 \tan \theta_l}{x_1^2} \right]^2 + y_{th} + y_c \quad (0 \leq x \leq x_1),$$

(3)

where $y_{th}$ is the height of the supersonic nozzle throat and $x_1$, $y_1$, and $\theta_l$ are the $x$-coordinate, $y$-coordinate, and angle at the inflection point, respectively. These parameters are respectively given by

$$y_{th} = M_e(H - y_c) \left[\frac{1}{(y-1)M_e^2 + 2}\right]^\frac{\gamma+1}{\gamma-1},$$

(4)

$$\theta_l = \frac{\nu(M_e)}{2.6},$$

(5)

$$y_1 = \frac{y_{th}}{M_1} \left[\frac{\sin \theta_1}{\theta_1} \right] \frac{1}{\gamma+1} \left[\frac{(y-1)M_e^2 + 2}{\gamma+1}\right]^{\frac{\gamma+1}{\gamma-1}},$$

(6)

$$x_1 = \frac{3(y_1 - y_{th})}{2 \tan \theta_l},$$

(7)

where $M_1$ and $M_e$ are the Mach numbers at the inflection point and the nozzle exit, respectively; $\nu$ is the Prandtl–Meyer function in radians; and $\gamma (= 1.40)$ is the specific heat ratio of air.

In this study, the contour of the terminal section was designed analytically using the method proposed by Foelsch (1946). Figure 4 shows the geometrical definitions of the variables used in this method. The nozzle contours are given as functions of the Mach number $M$ along the Mach line emanating from the inflection point, as

$$x = x_1 - r \cos \theta_1 + r \cos [\nu(M_e) - \nu(M)] + l \cos \left[\nu(M_e) - \nu(M) + \sin^{-1} \left(\frac{1}{M}\right)\right] \quad (M_1 \leq M \leq M_e),$$

(8)

$$y = r \sin [\nu(M_e) - \nu(M)] + l \sin \left[\nu(M_e) - \nu(M) + \sin^{-1} \left(\frac{1}{M}\right)\right] + y_c \quad (M_1 \leq M \leq M_e),$$

(9)

where
Finally, the contour of the convergence section is determined using the following quadratic function:

$$y = \frac{y_i - y_{th}}{L_{CS}^2} x^2 + y_{th} \quad (-L_{CS} \leq x \leq 0),$$

where $y_i$ is the height of the nozzle inlet and $L_{CS}$ is the length of convergence section. In this study, the design Mach number, airstream total pressure, and airstream total temperature, which were used as the design inputs, were 2.80, 0.5 MPa, and 673 K, respectively. The design Mach number was determined by trial and error to yield an average Mach number in the mainstream at the isolator exit of 2.50.

This combined supersonic nozzle design methodology can be used to obtain information about the displacement thickness before determining the supersonic nozzle contour. Thus, the proposed design methodology is easier to apply than other previously developed design methodologies that use the nozzle exit geometry as the constraint condition.

Figure 4 Variables used in Foelsch’s (1946) method for the design of the contour of the terminal section.

4. Validation of the two-dimensional supersonic nozzle design methodology

To validate the supersonic nozzle design approach discussed in Section 2, a three-dimensional steady Reynolds-averaged Navier–Stokes (RANS) simulation using Fluent v.14.5 computational fluid dynamics (CFD) software (ANSYS Inc. Canonsburg, PA) was performed. The governing equations were the continuity equation, the RANS equation, the conservation of energy equation, and the state equation for an ideal gas. These equations were implicitly solved using the finite volume method. The sheer stress transport (SST) $k$–$\omega$ model (Menter, 1994) was used to model the turbulence.

Figure 5 shows the numerical grids used to model the supersonic nozzle, isolator, and divergent section designed in this study. The supersonic nozzle contour was designed using the method discussed in the previous section. The numerical grids were composed of 567,000 hexahedral elements. The minimum grid size was 10 μm. The symmetry boundary condition was applied at the center plane in the width dimension to reduce the computational cost. The non-slip and adiabatic boundary conditions were applied at the wall boundaries. While the specific heat ratio was given as a constant value (=1.40) in the supersonic nozzle design approach discussed in chapter 3, it is given as a function of the static temperature in the numerical simulations. Therefore, the temperature dependency of the specific heat ratio is considered.
in the numerical simulations.

Figure 5 Numerical grids of the supersonic nozzle, isolator, and divergent section for the validation of the two-dimensional supersonic nozzle designed in this study.

LDV measurements were also conducted to support the numerical validation performed in this study. LDV was performed using backscattering LDV optics (FlowLite Dantec Dynamics, Skovlunde, Denmark) and a signal processor (Burst Spectrum Analyzer, Dantec Dynamics Skovlunde, Denmark). The laser wavelength, beam half angle, fringe spacing, and shift frequency are 632.8 nm, 2.72°, 6.67 \( \mu \)m, and 40 MHz, respectively. A corner cube prism was used to improve the signal intensity by reflecting the forward-scattered light to the receiving optics. SiO\textsubscript{2} particles with a mean diameter of 1 \( \mu \)m were selected as seeding particles.

Figure 6 shows simulated and experimental velocity profiles of a supersonic airstream between \( y = 0 \) and 15 mm at \( x = -9 \) mm and \( z = 0 \) mm. The total temperature and pressure of the airstream in the numerical simulation were 300 K and 0.5 MPa, respectively. The plotted numerical velocity profiles include the profile of the mean velocity \( u_{ave} \) as well as the velocity profiles obtained by as the mean velocity \( u_{ave} \) plus or minus the turbulence intensity \( u' \). The turbulence intensity \( u' \) is given as a function of the turbulent kinetic energy \( k \) as

\[
u' = \sqrt{\frac{2}{3} k}.
\]

The mean velocity profile obtained from the LDV measurements is also plotted in Fig. 6. The total temperature and pressure of the airstream in the LDV measurements were 293 K and 0.5 MPa, respectively. The error bars indicate the root mean square of the velocity fluctuation, which corresponds to the turbulence intensity \( u' \). Although the numerical simulation overestimated the turbulence intensity in the boundary layer, the simulated and experimental mean velocity profiles are quantitatively in good agreement. This confirms that the numerical simulations performed in this study can be used to accurately quantitatively evaluate the mean velocity profile in the supersonic nozzle and the isolator. Also, this result means that the adiabatic wall condition adopted in the present study is acceptable.

![Figure 6: Simulated and Experimental Velocity Profiles](image-url)
Figure 6 Velocity profiles of a supersonic airstream between $y = 0$ and 15 mm at $x = -9$ mm and $z = 0$ mm.

Figure 7 shows the Mach number distribution in the supersonic nozzle on the plane of symmetry and the nozzle exit plane. The airstream total temperatures in the cases shown in Fig. 7(a), (b), and (c) were 300, 673, and 800 K, respectively, and the airstream total pressure in all three cases was 0.5 MPa. The inlet conditions for the case shown in Fig. 7(b) are the same as the inputs of the supersonic nozzle design approach performed in this study. As shown in Fig. 7(b), the uniformity of the Mach number distribution outside the boundary layer was very good. The mean Mach number of the airstream outside the boundary layer on the nozzle exit plane was 2.79. The region outside the boundary layer was defined as the region in which the velocity was greater than 99% of the maximum mainstream velocity in the plane. It was shown that the mean Mach number outside the boundary layer on the nozzle exit plane was almost equal to the design Mach number of 2.80. Furthermore, the mean Mach number in the isolator exit plane was 2.50, which is equal to the target value. Additionally, there were no apparent differences among the Mach numbers in the three cases shown in Fig. 7. Therefore, the influence of airstream total temperature on the Mach number distribution in the supersonic nozzle is negligibly small in this range of temperatures (300–800 K), which means that the supersonic nozzle designed in this study can create approximately equivalent supersonic flows in this temperature range.

Figure 8 shows the distributions of the stream pitch angle $\alpha$, stream yaw angle $\beta$, and stream angle magnitude $\theta$ in the nozzle exit plane for the same three cases as shown in Fig. 7. These angles are defined respectively as

$$\alpha = \tan^{-1}\left(\frac{v}{u}\right),$$  \hspace{1cm} (16)

$$\beta = \tan^{-1}\left(\frac{w}{u}\right),$$  \hspace{1cm} (17)

$$\theta = \tan^{-1}\left(\frac{\sqrt{v^2 + w^2}}{u}\right),$$  \hspace{1cm} (18)

where $u$, $v$, and $w$ are the $x$-, $y$-, and $z$-components of the velocity, respectively. Black regions in the distributions are regions in which the value was larger than the upper limit of the color map. The white lines indicate the position where the stream angle is zero. As shown in Fig. 8, there were no apparent differences among the different stream angle values in the three cases. Therefore, the influence of the airstream total temperature on the stream angle in the supersonic nozzle is negligibly small in the considered temperature range (300–800 K). In the mainstream region, both the pitch and yaw angles of the stream showed good uniformity. However, in the boundary layer, the stream yaw angle was in the range of −1.0 to 1.0, but the stream pitch angle showed a relatively large non-uniformity near the side wall.
from numerical simulations.

Figure 8 Distributions of the stream pitch angle, stream yaw angle, and stream angle magnitude in the nozzle exit plane. The black regions in the distributions are regions in which the value was larger than the upper limit of the color map. The white line indicates the position where the stream angle is zero.

Figure 9 Distribution of the static pressure in the divergent section of the supersonic nozzle.

This non-uniformity can be explained by the distribution of the static pressure in the divergent section of the supersonic nozzle shown in Fig. 9. The top, left, right, and bottom boundaries of each cross-sectional distribution represent the upper flat wall (\(y = 30\) mm), side wall (\(z = 30\) mm), symmetry line (\(z = 0\) mm), and bottom nozzle wall (\(y = 0\) mm), respectively. Figure 9(f) shows the pressure distribution in the nozzle exit plane. Figure 9(a)–(d) shows the
presence of a pressure gradient in the y-direction. Because this pressure gradient on the side wall is indispensable for the acceleration of the mainstream in the two-dimensional supersonic nozzle, it is considered difficult to avoid an increase in the stream pitch angle in the boundary layer on the side wall. However, it may be possible to suppress this increase near the side wall by gradually accelerating the mainstream.

Figure 10 shows the displacement thickness $\delta^*$ along the upper flat wall of the supersonic nozzle in the plane of symmetry plotted against the $x$-coordinate. The black points indicate the numerical results in the supersonic nozzle and the isolator. The red point indicates the displacement thickness obtained by Eq. (1), and the red line indicates the displacement thickness obtained based on assumptions (2)–(4) in Section 2.1. Although the displacement thickness does not develop along the wall upstream of the nozzle throat, it is not negligibly small. Additionally, the displacement thickness does not increase linearly along the wall of the divergent section but does increase linearly along the wall downstream of $x = -250$ mm, where little decrease in the static pressure is induced by the expansion wave. Therefore, to more accurately estimate the development of the displacement thickness, it is necessary to properly evaluate the displacement thickness at the nozzle throat and consider the influence of the pressure drop induced by expansion waves on the development of the displacement thickness.

Figure 11 shows velocity profiles and displacement thicknesses in the nozzle exit plane ($x = -200$ mm) along different 4-mm line segments originating at the wall. Figure 11(a) shows the distribution of the $x$-component of the velocity in the nozzle exit plane. The white arrows labeled A–K in Fig. 11(a) indicate the line segments used to evaluate the velocity profiles. Figure 11(b) and (c) shows the velocity profile and displacement thickness, respectively, along each of the line segments defined in Fig. 11(a). As shown in Fig. 11(a), the mainstream region has a trapezoidal shape, with the shorter and longer bases of the trapezoid lying along the upper and lower sides of the nozzle, respectively. Figure 11(b) and (c) reveals that the mainstream region develops this trapezoidal shape as a result of the relatively low boundary layer thickness along line segments F and G. Because this low boundary layer thickness is considered to be caused by the pressure gradient on the side wall observed in Fig. 9, it is difficult to prevent the mainstream region from taking on a trapezoidal shape in the case of a two-dimensional supersonic nozzle. However, it is considered to be possible to suppress the trapezoidal shape formation of the mainstream region by gradually accelerating the mainstream. It should be noted that the gradual acceleration of the mainstream increases the length of the supersonic nozzle, thereby also increasing the displacement thickness. As a result, although the displacement thickness estimation method performed in this study underestimated the development of the displacement thickness, the supersonic nozzle designed in this study showed good uniformity of the Mach number and stream angle in the mainstream region. Moreover, the average Mach number in the mainstream region at the nozzle exit, which was 2.79, showed very good agreement with the design Mach number of 2.80.

Figure 10 Displacement thickness along the upper flat wall of the supersonic nozzle in the plane of symmetry plotted against the $x$-coordinate.
Figure 11 Velocity profiles and displacement thickness in the nozzle exit plane. (a) Distribution of the $x$-component of the velocity in the nozzle exit plane ($x = -200$ mm) and definition of line segments for velocity profile assessment. (b) Velocity profiles along each line segment. (c) Displacement thickness of each velocity profile.

5. Conclusions

In this study, a design methodology for a two-dimensional supersonic nozzle for the development of a direct-connect supersonic combustion test facility was proposed and investigated experimentally and numerically. The following conclusions were reached in this study.

1) The supersonic nozzle design methodology proposed in this study can be used to obtain information about the displacement thickness before the supersonic nozzle contour is determined. Thus, the supersonic nozzle design methodology is easier to apply than other previously developed methodologies requiring the nozzle exit geometry as a constraint.

2) Although numerical simulations overestimated the turbulence intensity in the boundary layer, the mean velocity profile obtained by numerical simulation and that measured by LDV were quantitatively in good agreement. Therefore, the numerical simulations performed in this study can yield quantitatively accurate estimates of the mean velocity profile in the supersonic nozzle and the isolator.

3) The influence of the airstream total temperature on the distributions of the Mach number and stream angle in the mainstream region was negligibly small in the considered temperature range of 300–800 K, which means that the supersonic nozzle designed in this study can create approximately equivalent supersonic flows in this range of airstream total temperatures.

4) Although the displacement thickness estimation method proposed by Burke (1961) underestimated the development of the displacement thickness, the supersonic nozzle designed in this study showed good uniformity of the Mach number and stream angle in the mainstream region.

5) The mainstream region has a trapezoidal shape, with the shorter and longer bases of the trapezoid lying along the upper and lower sides of the nozzle. Because the formation of this trapezoidal shape is considered to be caused by the pressure gradient along the side wall, it is difficult to prevent it in the case of a two-dimensional supersonic nozzle.

6) The average Mach number in the mainstream region at the nozzle exit plane, which was 2.79, showed very good agreement with the design Mach number of 2.80.
References

Burke, A. F., Turbulent boundary layers on highly cooled surfaces at high Mach numbers, ASD Symposium on Aerothermoelasticity, AFASD TR 61-645 (1961), pp. 704-741.

Fiore, A. W., Moore, D. G., Murray, D. H. and West, J. E., Design and calibration of the ARL Mach 3 high Reynolds number facility, ARL 75-0012 (1975).

Foelsch, K., A new method of designing two-dimensional Laval nozzles for a parallel and uniform jet, North American Aviation Report, No. NA-46-235 (1946). – Cited by: Crown, J. C., Supersonic nozzle design, National Advisory committee for Aeronautics, TR-1691(1948).

Gruber, M. R., Baurle, R. A., Mathur, T. and Hsu, K.-Y., Fundamental studies of cavity-based flameholder concepts for supersonic combustors, 35th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, A99-31094 (1999).

Gruber, M. R. and Nejad, A. S., Development of a large-scale supersonic combustion research facility, 32nd Aerospace Science Meeting & Exhibit, AIAA-94-5044 (1994).

Kobayashi, K., Tomioka, S., Mitani, T., Supersonic flow ignition by plasma torch and H2/O2 torch, Journal of Propulsion and Power, Vol. 20 (2004), pp. 294-301.

Menter, F.R., Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal, Volume 32, No. 8 (1994), pp.1598-1608.

Miyazato, Y., Kashitani, M. and Matsuoka K., Development of turbulent boundary layer in a supersonic nozzle (in Japanese), Engineering Science report, Kyushu University, Vol. 16, No. 3 (1994), pp.315-322.

Niioka, T., Hasegawa, S., Tsukamoto, T., Kobayashi, H. and Terada, K., Supersonic combustion flow test facility (in Japanese), Report of the Institute of Fluid Science Tohoku University, Vol. 3 (1992), pp.1-13.

Prandtl, L. and Busemann, A., Näherungsverfahren zur zeichneris chen ermittlung von ebenen strömungen mitÜberschallgeschwindigkeit, Stodola Festschrift (Zurich) (1929), S. 499-509. – Cited by: Evvard, J. C., Marcus, L. R., Achievement of continuous wall curvature in design of two-dimensional symmetrical supersonic nozzles, NACA TN-2616.

Rasmussen, C. C., Driscoll, J. F., Hsu, K.-Y., Donbar, M., Gruber, M. R., Carter, C. D., Stability limits of cavity-stabilized flames in supersonic flow, Proceedings of the Combustion Institute, Vol. 30, Issue 2 (2005), pp. 2825-2833.

Stratford, B. S. and Beavers, G. S., The calculation of the compressible turbulent boundary layer in an arbitrary pressure gradient – A correlation of certain previous methods, Aeronautical Research Council Reports and Memoranda No. 3207 (1961).

Tucker, M., Approximate turbulent boundary-layer development in plane compressible flow along thermally insulated surfaces with application to supersonic-tunnel contour correction, NACA Technical Note 2045 (1950).

Wang, H., Wang, Z., Sun, M. and Wu, H., Combustion modes of hydrogen jet combustion in a cavity-based supersonic combustor, International Journal of Hydrogen and Energy, Vol. 38, Issue 27 (2013), pp. 12078-12089.

Wolf, W. D., A selected annotated bibliography to aid in the development of quiet wind tunnel technology, NASA Contractor Report 4294 (1990).

Yamaguchi, T., Hizawa, T., Ichikawa, T., Kudo, T., Hayakawa, A. and Kobayashi, H., Total temperature estimation of a hydrogen/air burned-gas torch igniter for a scramjet combustor, Journal of Thermal Science and Technology, Vol.13 No. 2 (2018), JTST0030.

Yamaguchi, T., Murata, K., Kudo, T., Hayakawa, A. and Kobayashi, H., A study of cavity combustion for pre-burnt injection gas from a single hole in supersonic air stream, Proceedings of the First Pacific Rim thermal Engineering Conference, PRTEC-14721 (2016).

Yoshida, A. and Tsuji, H., Design and operation of the supersonic combustion wind tunnel (in Japanese), Report of the Institute of Space and Aeronautical Science University of Tokyo, Vol. 13 (1977), pp.487-515.