Chinese Journal of Aeronautics, (2022), 35(1): 318–339

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com

REVIEW

Theories and methods for designing hypersonic high-enthalpy flow nozzles

Yunpeng WANG [a,*], Zonglin JIANG [a,b]

[a] State Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China
[b] School of Engineering Science, University of Chinese Academy of Sciences, Beijing 100049, China

Received 12 August 2020; revised 7 September 2020; accepted 29 September 2020
Available online 20 March 2021

KEYWORDS

Boundary layer correction;
High-enthalpy flow;
Hypersonic wind tunnel;
Nozzle design;
Real-gas effects

Abstract Hypersonic high-enthalpy wind tunnels have been a challenge to ground tests in aerospace research area for decades and its test flow uniformity is one of the most important parameters for evaluating wind tunnel performances. Regarding to the performance requirement, theories and methods for designing hypersonic flow nozzles at high enthalpy conditions are quite difficult, but very interesting topics, especially when air molecule dissociations take place in wind tunnel test gas reservoirs. In this paper, fundamental theories and important methods for nozzle designs are briefly reviewed with the emphasis on two-dimensional axisymmetric nozzles for hypersonic high-enthalpy wind tunnels, including the Method of Characteristics (MOC), the graphic design method, the Sivells method, the theory for boundary correction, and the CFD-based design optimization methods. These theories and methods had been proposed based on several physical issues, respectively, which play important roles in nozzle flow expansion processes. These issues cover the expansion wave generation and reflection, the boundary layer development, the real gas effect of hypersonic high-enthalpy flows. Difficulties arising from applications of these methods in hypersonic high-enthalpy nozzle design are discussed in detail and the state of the art of the nozzle design technologies that have reached for decades is summarized with some brief comments. Finally, the prospect for the hypersonic nozzle design methods, and its numerical and experimental verifications are provided with from authors' viewpoint for readers' reference.

1. Introduction

Wind tunnels have been one class of the crucial testing facilities for aviation and aerospace technologies for more than one century. The wind tunnel testing for hypersonic vehicles requires even more higher performances than supersonic ones. To generate high-enthalpy test flows in a hypersonic wind tunnel, it is not only that the total temperature must be high enough to create obvious real gas effects, but also the flow uniformity

* Corresponding author.
E-mail address: wangyunpeng@imech.ac.cn (Y. WANG).
Peer review under responsibility of Editorial Committee of CJA.

Production and hosting by Elsevier

https://doi.org/10.1016/j.cja.2021.01.018
1000-9361

needs to reach a high level to ensure experimental data accuracy. Some scholars often use a word, "hypervelocity", to distinguish high-enthalpy flows from high Mach number flows at low static temperatures, which are produced usually by the conventional hypersonic wind tunnels.[1] In order to generate the high enthalpy flows the total temperature of which, ranging from 1500 K to 10000 K for simulating flight Mach numbers from 5 to 20, the shock-reflected wind tunnel is recognized as one of the best ground test facilities to investigate the real-gas effects in the opinion of Stalker. The shock tunnel operated in the shock-reflected mode creates the high-temperature air source in a manner of "what you produce is what you use", which would greatly reduce the heat load acting on the wind tunnel so that the high-enthalpy shock tunnel could be of engineering feasibility.

To improve shock tunnel performances, various shock tunnel drivers have been developed over the world to create strong incident shock waves since the mid-1960s. Of those driver techniques, the free-piston driver and the detonation driver are widely applied to the high-enthalpy shock tunnels. With the free-piston driver technique proposed by Stalker,[2] the test gas can be compressed and heated by the incident shock to obtain a test flow with a velocity as high as 7–8 km/s and an enthalpy up to 40 MJ/kg. However, the effective test duration is usually 1 to 3 ms, which results in a big challenge for flow measurement technologies. Nevertheless, the free-piston shock tunnel is one of the most commonly applied facilities because it can generate test flows with both high Reynolds number and high Mach number simultaneously. This feature is quite useful to reproduce the flight condition of the hypersonic vehicles reentering into the Earth's atmosphere. Bird[3] first tried the detonation driver for shock tubes to generate a high-enthalpy gas source, and conducted both computations and analysis on the detonation-driven shock tube, respectively. Yu et al.[4] proposed a concept that adding a damping tube at the end of the driver section to receive the detonation front for reducing the pressure load arising from the detonation reflection, so that the backward detonation-driven technique can be used to produce high-enthalpy flows. Jiang and Yu[5] presented a long-test-duration shock tunnel theory which can be used to extend the test duration of the backward detonation-driven shock tunnel and is named as the tailored condition for the backward detonation drivers. The JF-12 shock tunnel operated on the condition is capable of reproducing high-enthalpy air flows for Mach numbers from 5 to 9 at altitudes of 25 to 50 km with a test duration as long as 100 ms. The wind tunnel nozzle is a special devise to expand such the test gases at extremely high pressure and temperature to required flow speeds, but the boundary layer and real-gas effect impose a big barrier on the nozzle design. Therefore, developing special theories and methods for designing hypersonic high-enthalpy nozzles has been an important research direction in experimental aerodynamics for decades.

The first systemic theory on the wind tunnel nozzle design was proposed by Prandtl and Busemann[6] in the 1920s when more and more wind tunnels were built up in Europe and called as the Method of Characteristics (MOC) that has laid a primary foundation for developing modern wind tunnel nozzles. In the 1930s and 1940s, a variety of graphic methods based on the MOC were proposed for designing high-speed and supersonic nozzles.[7–9] However, to meet with the growing requirement on test flow quality, the nozzles designed with the graphic methods have some deficiencies. On the basis of the MOC, various analytic design methods were also developed.[10–15] For example, the nozzle contour can be created with the MOC algorithm under the radial flow assumption by using both the theoretical and empirical formulas. The axial Mach number distribution proposed by Sivells can greatly improve nozzle flow quality. This classic method for the high-speed nozzle design is still in use today.

Hypersonic flights, nominally the flight Mach above 5, became a serious issue in the 1940s and 1950s, and reach to a great success in both ballistic missiles and Apollo program in the 1960s. Several wind tunnel technologies were developed to test hypersonic vehicles and their common problem is how to deal with real gas effects in designing high-enthalpy flow nozzles to gain a good test flow uniformity. During this period, some methods were proposed by considering the change of the specific heat ratio γ,[16,17] and the MOC was adopted for computing nozzle contours. In practice application, some certain errors occur if only the change of γ is considered in nozzle design and these errors' effect on hypersonic high-enthalpy flow tests are unacceptable. So, the numerical method from the Computational Fluid Dynamics (CFD) is introduced into nozzle contour computations for optimization to meet quality requirements of hypersonic test flows. This algorithm combination including more features of the chemically-reacting flows works well for improving nozzle flow performances, and also leads to better understanding on limitations of the MOC application to hypersonic nozzle designs in the early 1990s. The MOC and its improved versions are effective to the nozzle flow Mach numbers equal to 8 or less, and it is recommended to improve the nozzle contour for even high Mach numbers by integrating the solver of the Parabolized Navier-Stokes (PNS) equations.[18] This is because that when the flight Mach number is higher than 8, air dissociations and boundary development play more and more important roles in hypersonic high-enthalpy nozzle flows. There are many papers published on this topic[19–30] and the CFD algorithms are used for the nozzle design optimization. These researchers coupled different optimization algorithms with various CFD solvers, and their codes can automatically iterate to solve a set of equations and get optimized nozzle contours. As a result, the nozzle design methods combining CFD solvers and optimization algorithms improve the nozzle flow quality significantly, but the computation code and the optimization process are extremely complicated and time-consuming, and also the code robustness is also not good enough. Gaffney[31] and Tang et al.[32] decoupled the CFD solver with the optimization method in their computational process, which greatly reduces the computational complexity and workload in the designing process without obvious effects on nozzle flow quality. Thus, their methods are quite convenient for engineering applications.

By recalling progresses in wind tunnel nozzle design, most of the theories and methods were published before the 1970s and these works can be classified as analytic theories and the semi-analytic methods. With development of computer technologies and CFD algorithms, integrating the CFD solver into the nozzle design process can improve significantly nozzle flow quality. This is because that the governing equations of gas dynamics including all the physical phenomena that behave obviously in nozzle flows. Furthermore, integrating the computer-aided optimization algorithms into nozzle design with CFD solvers is also a significant step to obtain the high quality nozzle, and it is a kind artificial intelligence that is helpful to improve design efficiency and accuracy. Being different from subsonic and supersonic nozzle designs, these integrated methods could be more effective for high-enthalpy nozzles than analytic methods. Since then, several improved design theories and correction methods considering the high-temperature gas effects have been brought up, however, designing a high-enthalpy nozzle with superior quality still remains a challenge because there exist some difficulties both in modeling the high-temperature reacting gas flow and solving the multicomponent reaction equations of the non-equilibrium flows. The higher the enthalpy flow, the larger the deviation of the actual flow parameters from design requirements. Hypersonic technology is the core of the future aviation and aerospace industries and the hypersonic wind tunnel is one of its main research means, therefore, the research on the design theories and methods for the hypersonic high-enthalpy flow nozzles is still of great significance in the foreseeable future.

2. Fundamentals for designing high-enthalpy flow nozzles

In shock tunnels, a nozzle is placed between the driven section and the test section. Test gas is heated by an incident shock waves in the driven section. The generated high-temperature and high-pressure test gas expands through the nozzle shortly after the incident shock reflection and reaches a uniform test flow in the test section (see Fig. 1). The nozzle contour is a key factor for test flow uniformity and the nozzle design is a process how to determine the nozzle contour. This paper is dedicated to the design theories and methods of two-dimensional axisymmetric nozzles for hypersonic and the high-enthalpy flows.

The test gas reservoir is the original place where the nozzle flow starts its expansion and the reservoir condition at the end of the driven section can be determined with the incident shock wave. At hypersonic high-enthalpy conditions, the test gas in the reservoir undergoes a physical–chemical reacting process governing by local gas temperature and its flow expansion through the nozzle endures high-temperature gas effects. The expanding gas flow is very difficult to accurately evaluate with available computational simulations or experimental measurements.

Therefore, the early nozzle design methods adopt a quasi-one-dimensional chemical equilibrium flow model for the nozzle flow analysis. By supposing that the high-enthalpy flow is inviscid and adiabatic, and the nozzle flow of chemical equilibriums is isentropic. So, the conclusion about isentropic flows can be accepted to supersede the momentum equation or the energy equation. Generally speaking, the chemical reaction equilibrium does not cause the system to be irreversible. If the chemical equilibrium nozzle flow is reversible, any chemically-reacting nozzle flow that is heat-insulated without shock waves and viscosity will be isentropic and the two-dimensional axisymmetric nozzle flow can be simplified as a quasi-one-dimensional flow.[33] The cross-sectional area of the nozzle can be expressed by a function $A = A(x)$ with a variable x, and all the flow parameters are of constant values in a given section. The flow state parameters at any given location and the area ratio A/A^* depend on three parameters, i.e. the pressure p, the temperature T and the local speed u, and the following equations exist:[34]

$$\frac{A}{A^*} = g_1(p_0, T_0, u) \tag{1}$$

$$\frac{T}{T_0} = g_2(p_0, T_0, u) \tag{2}$$

$$\frac{p}{p_0} = g_3(p_0, T_0, u) \tag{3}$$

where p_0, T_0 are the total pressure and total temperature, respectively.

For a calorically perfect gas (specific heat ratio $\gamma = 1.4$), the nozzle flow characteristics are governed by the local Mach number (Ma) only, namely:

$$\frac{A}{A^*} = f_1(Ma) \tag{4}$$

$$\frac{T}{T_0} = f_2(Ma) \tag{5}$$

$$\frac{p}{p_0} = f_3(Ma) \tag{6}$$

So, the equilibrium flow through the nozzle can be simplified as an isentropic quasi-one-dimensional flow for predicting expansion flow characteristics. This was demonstrated in detail by Anderson,[34] but the above equations do not apply to non-equilibrium flows. The non-equilibrium flow phenomenon leads to some uncertainties in the high-enthalpy flow nozzle design.

Considering non-equilibrium flow effects, Hall and Treanor[35] limited their nozzle flow analysis to steady state. In 1970, a new solution was proposed by Anderson,[36,37] and the unsteady state was involved. By solving the equations with a time-advancing finite-difference method, the problem arising from the steady state analysis was avoided.[38] In many literatures, the analysis of the non-equilibrium nozzle flows was based on assumption that the local equilibrium state is reached at the throat, and the non-equilibrium calculation is started downstream of the throat. Doing so can avoid dealing with the problem of the saddle-point singularity and the unknown mass flow, and Erickson,[39] Harris and Albacete[40] did their contributions. The equations for the two-dimensional axisymmetric non-equilibrium flows were generally solved by using the MOC[6] or CFD for nozzle design. The entropy increase along streamlines in the non-equilibrium flow is a result of the irreversible factor in the finite-rate chemical reaction processes.

When the MOC method is used to non-equilibrium flows, it is noted that the characteristic curves that go through any point in the non-equilibrium supersonic flow is based on the Mach line and the streamline with the frozen sound speed. The detailed MOC solution can be found in the book by

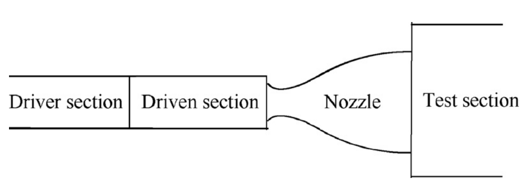

Fig. 1 Schematic of a shock tunnel in its basic configuration.

Zucrow and Hoffman.[41] In the hypersonic nozzle design, the MOC presents a theoretical limitation when the Mach number is high. The CFD method was widely used in the nozzle design to get the whole flow field solutions. But for the high-enthalpy flow of high Mach numbers, the limitations on the accuracy of the current chemical reaction models and the CFD techniques are also observable. Therefore, there are still some problems that had not been effectively solved for the nozzle design of the high-enthalpy hypersonic wind tunnels. In high-enthalpy expansion flows, the specific heat capacity ratio of the test gas is a variable, the high-temperature affects boundary layer development, chemical reactions and non-equilibrium processes also play an important role in it. These high-temperature gas effects must be considered to obtain good quality nozzle designs.

Hypersonic nozzles are usually quite long because of large area ratio and its dimension includes the length of the uniform flow area (generally called diamond-shaped area) at the nozzle exit. The current length-shortening method can do nothing to reducing the length without performance loss. The extra-long nozzle causes the increase of the boundary-layer thickness, and the thickness can occupy even 50% of the cross-sectional area of the nozzle exit. In addition, it is difficult to accurately evaluate the boundary-layer development (mainly turbulent) of the high-enthalpy flows with the current CFD solvers, which make it impossible to accurately amend the boundary layer thickness. Moreover, it is also difficult to accurately predict and evaluate high-enthalpy flows with the available chemical reaction models and its reliability can be hardly verified using current experimental data. Therefore, the development of the theories and methods for designing hypersonic high-enthalpy flow nozzles to gain high performance is still a challenge as before.

3. Design methodology of hypersonic flow nozzles

The hypersonic vehicle tests impose much strict requirements on hypersonic tunnel design techniques, especially, the test flow must be of high-enthalpy, hypervelocity and high uniformity. These requirements highlight the importance of the high-enthalpy flow nozzles in engineering. The chapter is focused on the conventional theories and methods for designing the hypersonic wind tunnel nozzles, which are the bases for the current hypersonic nozzle design.

3.1. Principles for hypersonic wind tunnel nozzles

Wind tunnel nozzles can be divided into four types according their configurations: axisymmetric, two-dimensional, three-dimensional and asymmetric nozzles.[10] Generally speaking, as the Mach number is increasing, the total temperature of the test flows is also increased to match with flight conditions, at least, to prevent the test gas liquefaction to ensure gaseous flows.[13] By comparing with the axisymmetric nozzle, it is difficult for the two-dimensional nozzle to maintain the flow stability at the throat section, and improve its expansion efficiency under the same design condition, therefore, most of the hypersonic nozzles are made axisymmetric. So, the theories and methods for two-dimensional axisymmetric nozzles are selected as a base for reviewing.

In the early stage, the hypersonic nozzle was designed with the MOC developed for supersonic nozzles so that the expansion curve from the throat to the exit can ensure that the test flow at the nozzle exit is parallel and uniform.[9] The expansion curve can be divided into the front section (from A to B, initial expansion section) and the back section (from B to C, supersonic expansion section), as shown in Fig. 2. In this figure, the cross section, OA at the throat is of the smallest section area. The line from point A, thru B, to C indicates the nozzle contour that governs test flow quality. β is an oblique angle; $\tan\beta_B$ is the nozzle contour slope at point B and defined as the maximum expansion angle. If ν_1 presents the Prandtl-Meyer angle corresponding to the Mach number at the test section, the value of β_B is determined with the nozzle flow Mach number and the design method.[33]

$$\beta_B \leqslant \frac{\nu_1}{2} \tag{7}$$

The axial distance from cross-section A to B is called the initial expansion section. In this section, the test gas flow gradually expanding and accelerating deflects outward along the wall AB, and a series of expansion waves in the nozzle flow are generated before it reaches the inflection point B. The axial distance from cross-section B to C is called the supersonic expansion section or the wave-damping section where the flow continues its expansion, but expands mainly inward. The slope of the nozzle curve BC decreases gradually and becomes zero finally at point C. The flow expansion process generates a series of compression waves along the back section, which can offset the reflected waves generated by the expansion waves at the front section. The Mach angle corresponding to the Mach number at the test section is μ, namely the included angle between the line EC and the axis (see Fig. 2). When reaching the EC line, the flow expansion and acceleration are completed, and the flow moves in parallel to the axis, and achieve the Mach number as required in the nozzle design.

In designing conventional hypersonic nozzles, the diameter of the nozzle exit at the test section is a constant. The Mach number at the exit depends on how to change the throat diameter according to the area ratio. The formula presents its relation:[33]

$$\frac{A}{A^*} = \frac{r^2}{r^{*2}} = \frac{1}{Ma}\left(\frac{\gamma-1}{\gamma+1}Ma^2 + \frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{2(\gamma-1)}} \tag{8}$$

where, A is the cross-section area of the nozzle; A^* is the cross-section area of the throat; r is the distance from the origin to any point along the nozzle contour; r^* is the diameter of the throat section; Ma is the local Mach number at the throat; γ

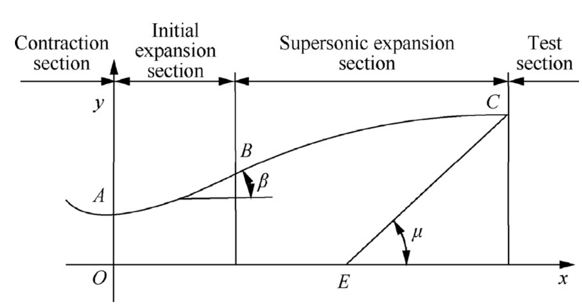

Fig. 2 Structure of axisymmetric nozzles.

is the specific heat capacity ratio of the test gas, which is 1.4 for ideal gas. This formula is effective for designing hypersonic nozzles at low total-temperature conditions. Obvious errors could be produced for designing high-enthalpy flow nozzles because the change of the specific heat capacity ratio due is not taken account.

In order to apply conventional methods in the hypersonic nozzle design at the high-enthalpy flow condition, the real-gas effects need to be appropriately corrected. Three steps are basically involved in the designing process. First, supposing that the nozzle flow is ideal, the potential flow contour (inviscid) can be obtained according to the isentropic relationship. Then, the boundary-layer development is predicted to estimate the displacement-thickness of the boundary layer. Finally, superimpose the estimated displacement-thickness onto the potential flow nozzle contour in the normal direction to gain the corrected nozzle contour. In addition, the design methods discussed mainly focus on the expansion section of high-enthalpy flow nozzles. As to the contraction section of the nozzle, the polynomial fitting method widely used is relatively simple and works well so that it would be not discussed here.

3.2. Conventional design methods for hypersonic nozzles

As mentioned in the above chapter, the MOC has laid a theoretical foundation for modern nozzle designs. The nozzles for supersonic and hypersonic wind tunnels were mostly designed based on the MOC since the MOC were successfully verified for supersonic wind tunnel nozzles.[12,13] Although some special problems were encountered in the hypersonic wind tunnel development (such as the real-gas effects, the heat dissipation and deformation of the throat, and the thicker boundary-layer), a considerable amount of the nozzle design work produced satisfactory results after necessary corrections were made.

A variety of theoretical methods derived from the MOC includes the graphic method and the analytic method. The nozzle contours designed with these methods can be basically divided into three types: (A) Curved throat contours (with a large "curvature radius") designed with the maximum expansion angle $\beta_B = 0.5v_1$; (B) Curved throat contours (with a large "curvature radius") designed with the maximum expansion angle $\beta_B < 0.5v_1$; (C) "Abrupt change" throat contours (with a small "curvature radius") designed with the maximum expansion angle $\beta_B = 0.5v_1$.[33,35,42]

Among these three types, the third one produces the shortest nozzle, followed by the first, and then the second. The third type of the nozzles had an abrupt expansion near the throat and a large axial pressure gradient, which often causes an abrupt change in the boundary-layer thickness on the nozzle wall, and even flow separation. The nozzle flow quality is poor in uniformity. The second type of the nozzles uses a smaller maximum expansion angle than the third one, therefore, it was too long to produce a thin boundary-layer for hypersonic flows. Thus, the method for the first type of the nozzle was commonly used. In conclusion, with the conventional methods of the MOC family, the nozzle boundary-layer will become too thick to ignore when the hypersonic test flow is to generate (in particular, the modern nozzles are generally designed for large wind tunnels). So, the boundary layer effect must be corrected to according to the design Mach number at the nozzle exit.

3.2.1. Graphic design method

Various graphic methods were proposed in the 1940s and 1950s, and the Busemann method[7,8] is the significant one of these MOC based methods. By assuming that the flow at the throat is uniform, the method according to the fundamental nozzle design theory and experience is used to set an initial curve AB where the slope is gradually increasing and $\beta_B = 0.5v_1$ is satisfied at point B. Curve AB, as shown in Fig. 3[8], is replaced with polylines $AA_1A_2\ldots A_NB$, where A_i is the inflexion point. Each polyline turns outward slightly at the same (or variable) angle $\Delta\beta$. The smaller $\Delta\beta$ is, the more accurate the result will be. According to the characteristic theory, there exist the expansion waves originating from point A_i, the reflected waves from the axis or walls, and these waves generated from flow expansion are intersected with the reflected waves. All of these waves can be predicted with the characteristic theory and all the areas surrounded by the wave lines are of the uniform flow. Finally, points A_i are connected in sequence to form a continuous smooth nozzle contour. The supersonic flow travels around the outer obtuse angle to produce the expansion wave system which is only approximate to a single wave. This method is very useful for designing the non-conventional nozzles, while the analytic method developed later is more convenient and computationally accurate for the conventional hypersonic nozzles.

Puckett[9] modified the conventional Busemsmn method, and the new method has several advantages. Using the simplified initial design curve can save almost half of the designing time or effort. The characteristic curve is calculated from the middle of the nozzle contour to its two ends. It is assumed that the flow passing through the nozzle at the maximum expansion (the inflection point) has a uniform speed and a uniformly-varying flow direction (see Fig. 4). The condition setting according to design experiences is reasonable. With these parameters, the nozzle exit diameter can be determined in the same way as the Busemann method. Similarly, the initial expansion section can be constructed. In addition, if β is less than β_B, one or multiple inflection points must be added to reflect expansion waves. Since the reflected waves can be selected, several initial curves AB can be also selected so that the design Mach number can be reached at the contour of the corresponding damping section. If the mesh size can be infinitely small, there will be an infinite number of the initial curves for selection in theory. Likewise, there will be countless damping section curves corresponding to the specific initial expansion curves. Despite of these, it does not indicate that any contour satisfying the area ratio expressed with Eq. (8)

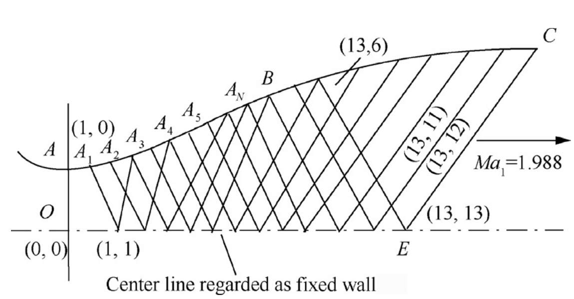

Fig. 3 Characteristics mesh for nozzle design.[8]

is appropriate. In addition, for most of the engineering applications, the errors arising from initial curve selecting can be ignored as long as an appropriate correction is made.

In the design process with the Busemann method, expansion waves are generated on certain points along a smooth initial curve. This means that the expansion waves are spaced in an orderly way though they are not required to be uniformly distributed. When the mesh size is large, the expansion waves are sometimes reflected from the wall at some points, which thus interferes the order of the spaces of the subsequent expansion wave system. As a result, the final nozzle damping section contour shows slight irregularity. Theoretically speaking, such the irregularity seems to disappear as the mesh becomes infinitely dense, but the problem cannot be avoided in practice when the Puckett method is applied. So, it is generally assumed that the damping curve is not affected by the waveform of the initial curve and the meshes are made dense enough so that the error could be ignorable to obtain the smooth contour as much as possible. However, due to limitations on the computational technology at the early stage, the design efficiency is decreased as the computational workload increases when the mesh becomes denser and denser. Besides, the analytic method comes out to be far superior to the graphic method in terms of the nozzle flow quality so that the graphic method had not been widely applied to the high-speed nozzle design. However, the clear physical phenomena dealt with the graphic method is helpful in development of the nozzle design methods.

3.2.2. Analytic design method and Foelsch method

The analytic method is similar to the graphic method and its special features will be explained by referring back to the nozzle contour as shown in Fig. 5[10]. Point B is the inflection point, and the slope of contour ABC reaches to the largest value at point B; and point A is the starting point of the nozzle contour while C is the end point. Generally, the slope is zero at points A and C. The analytic method adopts a radial flow assumption. The design principle of the initial expansion section is to use different approximation methods to transform the sonic flow at the throat into the radial flow at point B. The damping section contour is to transform the supersonic or hypersonic radial flow into a uniform flow moving parallel to the axis at the design Mach number. In addition, all the expansion waves originating from walls must be dampened. Various analytic methods were developed based on the MOC theory for the nozzle design and their common feature is that the flow at the inflexion point B is assumed to be a supersonic radial flow. The contour curve after point B is obtained with the analytic methods. The key point for designing this curve is to dampen all the expansion waves that is reflecting along the curve. Although different methods are used to obtain the contour curve before point B, there is a shared idea that the curve at the initial expansion section should be able to transform the sonic flow at the throat into the supersonic radial flow at the inflexion point.

In the design process of the nozzle contours, a potential flow nozzle contour is first obtained with a rigorous theoretical method, then the boundary-layer correction is carried out to count viscosity effects. However, one problem that needs to point out is how to evaluate and calculate the boundary layer development with sufficient accuracy, especially for the hypersonic boundary layer of the high-enthalpy flows in which the gas flows are reacting. Both theoretical and numerical methods hardly ensure accurate evaluations, therefore, a very accurate contour design seems to be impossible. In addition, the theoretically-calculated wave system may be quite different from the actual flow structure. So far, most of the potential flow nozzle contours have been designed with acceptable accuracy by using the simple approximation method, which can greatly reduce nozzle design workloads.

For the nozzle design, especially for low-velocity nozzles, the radial flow assumption of the analytic method is reasonable to a certain extent and the design results basically meet wind tunnel requirements. For designing the nozzles at high Mach numbers, various optimization techniques had been applied to obtain a high-quality flow. Several classical analytic methods with optimization process for nozzle design were developed and some methods are still in use today.

Foelsch[10,11] proposed his approximation method in 1947 and the certain flow area of the nozzle is assumed as the radial flow. The initial expansion section is designed by using the empirical formula and the flow is transited to the uniform area with the MOC. However, due to the limitation on the radial flow assumption, the nozzles designed with this method are generally very long. The direct connection between the radial flow area and the uniform flow area results in a discontinuous axial velocity gradient, which makes the flow quality unsatisfactory. The Foelsch method does not guarantee that the radial flow assumption can be met at the inflection point. Instead, an empirical curve is used with the curvature changing monotonically and the area ratio formula is satisfied at point B. The curve equation is given as below:

$$y = y^* + x^2 \frac{\tan\beta_B}{x_B}\left(1 - \frac{x}{3x_B}\right) \tag{9}$$

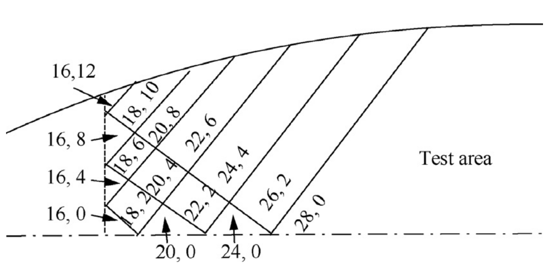

Fig. 4 Schematic of demonstrating Puckett method for nozzle design.[9]

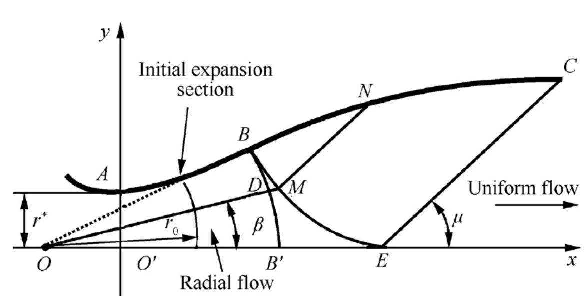

Fig. 5 Schematic of demonstrating Foelsch method for nozzle design.[10]

where,

$$x_B = \frac{3}{2}(y_B - y^*)\cot\beta_B \quad (10)$$

In the initial expansion section, the actual flow at the inflexion point differs from the radial flow, which has certain impacts on the flow uniformity in the test section. The damping section contour connects the starting and end points (B, C) with discontinuous curvature, but has no impact on hypersonic nozzles with fixed walls.

Crown[42] improved the Foelsch method and proposed a new curve (see Eq. (11)) as the initial expansion contour, however, there is no substantial improvement on the Foelsch method.

$$y = y^* + x_B(\lg\beta_B)\left(\frac{x}{x_B}\right)^3\left(1 - \frac{x}{2x_B}\right) \quad (11)$$

where,

$$x_B = 2(y_B - y^*)\cot\beta_B \quad (12)$$

The Foelsch method is significant because it "reasonably" processes the flow near the nozzle throat by using the radial flow assumption, and greatly simplifies the design work of the wall contour at the throat which is the most complicated part of the nozzle design. As a result, a series of the nozzle design methods have been derived based on radial flow assumption so far, and are widely applied to both high and low-velocity wind tunnels that had been constructed in the last century.

3.2.3. Cresci method

The Foelsch methods were modified later by Cresci[12] and a damping area was added to the nozzle contour to reduce the axial velocity gradient discontinuity arising from application of the Foelsch method. In the Cresci method, the nozzle flow is divided into three areas, as shown in Fig. 6. Area $OABD$ is the throat expansion region which is considered as the radial flow from the origin. Area $BDEC$ is the transition area which rectifies and straightens the radial flow into the parallel one. The area indicated with "uniform flow" is the parallel flow region which has the uniform flow moving parallel at along the axis at the same Mach number.

The initial expansion section AB is designed with the empirical curve from the Foelsch method. It is assumed that the axial velocity distribution satisfies a polynomial in the damping area $BDEF$. According to experiences, the Mach number at point D is 0.2 being less than the Mach number at point E, i.e., the design Mach number. In this case, suppose that the velocity distribution of the DE satisfies a cubic polynomial expressed as

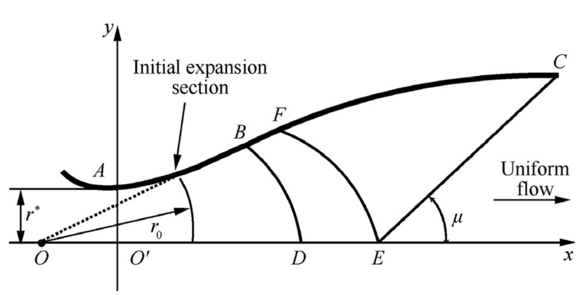

Fig. 6 Cresci method for nozzle design.[12]

$$F = c_0 + c_1\bar{x} + c_2\bar{x}^2 + c_3\bar{x}^3 \quad (13)$$

where, c_0, c_1, c_2, c_3 are the undetermined coefficients and $\bar{x} = (x - x_D)/(x_E - x_D)$, the cubic polynomial satisfies the following boundary conditions:

If $\bar{x} = 0, F = F_D$, then,

$$\left[\frac{dF}{dx}\right]_D = (x_E - x_D)\left(\frac{dF}{dx}\right)_D \quad (14)$$

If $\bar{x} = 1, F = F_E$, then,

$$\left[\frac{dF}{dx}\right]_D = 0 \quad (15)$$

According to

$$3(F_E - F_D) = (x_E - x_D)\left(\frac{dF}{dx}\right)_D \quad (16)$$

the value of x_E is selected. The positions of points D and E can be determined by substituting the boundary condition into the cubic polynomial. The leftward characteristic curve through point E is a straight line with the design Mach number so that the endpoint C of the characteristic curve can be determined. With BD, DE, and EC as the boundary condition, the characteristic curve mesh throughout the flow and the nozzle contour of the BC section can be obtained with the characteristic theory and the mass conservation law. Since the empirical curve from the Foelsch method is still used in designing the initial expansion section in the Cresci method, it is difficult to optimize the nozzle contour obtained on this basis. However, by comparing with the Foelsch method, the Cresci method is a more mature technology for nozzle contour designs.

3.2.4. Sivells method

Most of the previous methods focus only on the nozzle contour design, which leads to some undesired phenomena such as disturbance accumulations at the axis and flow separations on the nozzle wall. In the late 1960s and the 1970s, Sivells[13–15] proposed his design method that relies on the axial velocity and Mach number distribution specified according to the Cresci method and the Hall transonic theory.[43] With the Sivells method, the nozzle contour being of continuous curvature can be obtained by setting the axial Mach number distribution. As a result, there are no concentrated compression waves inside the nozzle, and the Mach number distribution at the exit becomes uniform, and the nozzle flow quality is greatly improved.

When using the Sivells method, the nozzle contour is determined with the characteristics mesh among the known boundary points, the given axial Mach number distribution, and the approximate analytic solution at the sonic throat. The nozzle is divided into three areas, as shown in Fig. 7 and these are the throat area $TIEG$, the radial area $GEBA$, and the downstream area $ABCD$. The flow is radial in the radial area $GEBA$ between the characteristic curves EG and AB, and the nozzle contour from point G to point A is a straight line inclined to the axis at an angle of ω. Therefore, determining the contour involves two steps. That is to say, two sets of characteristic equations are required to solve. First, the throat contour from point H to point G on the branch line is determined. Point H is located on the leftward characteristic curve starting from point I with the sonic speed. The flow in the throat area is calculated with the Hall transonic flow solution. Next, the downstream contour from point A to point D is determined, and as shown in Fig. 7, the characteristics CD is a straight line where point D is the theoretical endpoint of the nozzle contour, namely, the position of the nozzle exit. According to these assumed boundary conditions, the flows in $IHGE$ and $BADC$ areas can be solved by combining the characteristics mesh with the mass conservation law, thereby, the nozzle contours along HG and AD can be obtained. The contour of the TH section is obtained with the transonic solution where the velocity distribution from point I to point C is first assumed. The velocity distribution is made to match with theoretical transonic conditions at the throat, radial flow conditions through an intermediate region, and design flow conditions at the nozzle exit.

The axial velocity distribution from point I to point C can be divided into three parts. The first part is from point I with the sonic speed in the throat area to point E where the radial flow is assumed to start. Its velocity distribution is defined by a quartic polynomial. The second part is from point E to point B where the radial flow ends. Its velocity distribution is given with the condition under which the radial flow is generated. The third part is from point B to point C where the uniform flow starts. Its Mach number distribution is defined with a quartic polynomial. The coefficients of the polynomial are selected based on the principles that the second derivatives of the axial velocity are continuous all the time and the derivative at point C is zero.

As reported by Potter and Carden,[44] the conventional MOC had been used to design two nozzles (one with Ma = 9 and the other with Ma = 10) for AEDC-VKF (Arnold Engineering Development Center (von Kármán Facility) low-density hypersonic wind tunnels). The Cresci method[12] and the Sivells method[45] are combined to design the inviscid potential flow nozzle. The Cresci method based on the radial flow assumption is used to design the damping section area (Area III, as shown in Fig. 8). According to the Sivells idea, the Mach number distribution along the axis is given to complete the contour design. Because of ignoring the impact of the variable specific heat capacity ratio and mistaking the nozzle flow as the full laminar flow due to its low density, some design errors emerge. The test flow calibration shows the actual Mach numbers at the exits of two nozzles are 10.15 and 9.30, respectively, instead of the design Mach numbers of 9 and 10. Carefully checking numerical results, it is observable that the Sivells method can produce the better nozzle flow quality than the Foelsch and Cresci methods. Its flow quality can even be further improved as the design Mach number increases. Therefore, the Sivells method is more promising for designing the hypersonic nozzle that needs to meet strict requirements.

Because of the limitation on the power of computational technologies at that time when the Sivells method was proposed, the nozzle length, especially under the high-velocity flow condition, is quite long. It is okay for small diameter nozzles, but becomes unacceptable for large diameter nozzles. To solve this problem, a number of the design methods for shortening nozzles is proposed. For example, a widely-used one was developed by Ali et al.[46] based on the assumption of the straight sonic line and the curved sonic line at the throat. The nozzle-shortening method can meet design requirements at lower Mach numbers, but they can hardly ensure the flow quality when the flow Mach number gets high, therefore, the method is not applicable for designing hypersonic nozzles.

In practice, the potential flow nozzle contour designed with MOC can meet design requirements of hypersonic nozzles which are not required to take account of real gas effects. The Sivells method is a milestone since which the conventional MOC nozzle design techniques have become mature. So far, the Sivells method is still one of the internationally-recognized design methods for high-velocity nozzles that feature high accuracy and gain wide engineering applications.

3.2.5. Theory and method for boundary-layer correction

The viscous nozzle flow behaves different from the inviscid one and the viscosity effect plays an important role in two aspects. The first aspect is the boundary layer developing over the nozzle wall, and the boundary-layer thickness affects significant on the nozzle contour design. The thick boundary-layer will reduce the potential flow area, thereby changing the nozzle area ratio. As a result, the actual Mach number at the exit will be lower than the design Mach number. The second one is the wave system generated in the nozzle expansion and it interacts with the boundary layer, which results in some flow perturbations. Such the interference affects not only the boundary-layer development, but also the configuration of the wave system, therefore, leads to a big difficulty in completely eliminating both the expansion waves and its reflections in the potential nozzle flow. As a result, the flow reaching at the nozzle exit is not uniform.

These two problems become even more prominent and serious under the high-enthalpy flow condition. Therefore, the boundary-layer correction must be performed on the designed potential flow contour. Generally speaking, the displacement is calculated according to the boundary-layer thicknesses at each point of the potential flow nozzle contour and then superimposed onto the potential flow nozzle contour in the normal

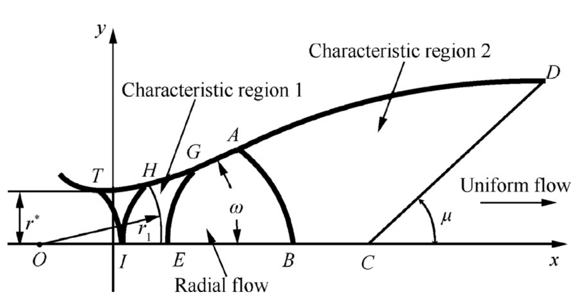

Fig. 7 Sivells method for nozzle design.

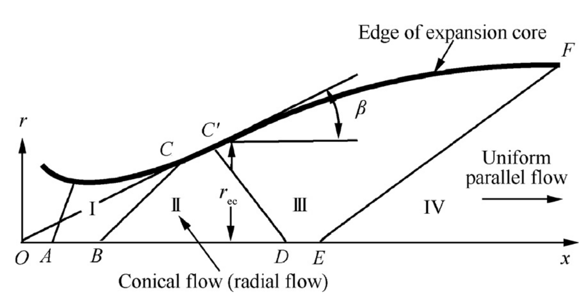

Fig. 8 Schematic diagram for design of inviscidnozzle contour.[44]

direction to obtain the new nozzle contour. Namely, the theoretical nozzle contour is moved outward by the equivalent displacement-thickness.

The boundary-layer thickness δ at the nozzle wall leads to the change of the nozzle cross-section area. The change can be denoted by the distribution of the displacement-thickness δ^*. The displacement-thickness, as shown in Fig. 9, is defined as:

$$\delta^* = \int_0^\delta \left(1 - \frac{\rho u}{\rho_1 u_1}\right) dy \tag{17}$$

where ρ and u are the density and flow speed in the uniform flow, respectively; ρ_1 and u_1 are the density and the flow speed in the boundary-layer, respectively. The potential flow eliminates the impact of δ^* on the nozzle contour, so the boundary-layer correction is that displacing the nozzle potential flow contour outward along the wall in the normal direction by δ^* is namely superimposing δ^* onto the potential flow contour in the normal direction. The boundary-layer correction of the nozzle contour is performed according to the following formula.

$$\begin{cases} x = x_{\text{potential flow}} - \delta^* \sin\theta \\ y = y_{\text{potential flow}} + \delta^* \cos\theta \end{cases} \tag{18}$$

where, $x_{\text{potential flow}}$ and $y_{\text{potential flow}}$ are the coordinates for the potential flow contour; θ is the tilt angle of the potential flow wall relative to the x-axis. The growth law of δ^* on the wall is correlated to the characteristics of the viscid flow in nozzles.

When the nozzle flow is accelerated in the axial direction, the boundary-layer on the wall develops generally in a favorable pressure gradient. The distribution of the boundary-layer thickness δ^* can be determined with any theoretical, empirical or numerical methods. In the nozzle design for low-velocity wind tunnels, the computation and evaluation of the displacement-thickness of the boundary layer are relatively simple. A theoretical approximation or an empirical formula is often used because the nozzle is short and a boundary-layer is thin. This simple correction will not cause obvious deviations in test flows at the nozzle exit and quite satisfactory results can be obtained. However, the deviations will become ignorable for hypersonic nozzles as well as high-enthalpy nozzles. This is because that the hypersonic boundary layer is getting thick and the high-temperature makes it develop faster. Therefore, no matter which method is used, the boundary-layer correction must be carried out with the accurate evaluation of the boundary layer.

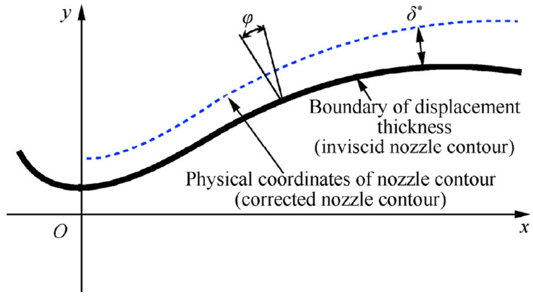

Fig. 9 Schematic diagram for boundary-layer and displacement-thickness line.

The boundary layer over the nozzle wall of the hypersonic wind tunnels (except for low-density wind tunnels) is generally developed into a turbulent state. In order to prevent the test gas liquefaction in the conventional hypersonic wind tunnels, the test gas is heated from several hundred to a thousand K. The nozzle walls require cooling to operate continuously (for example, in the electric arc wind tunnels). The heat exchange crossing the nozzle wall makes the boundary-layer computation complicated, especially for the high-enthalpy boundary layer where the temperature is high and the test gas is reacting. The computation and evaluation methods for hypersonic boundary layers are still not mature at present and the methods for low-velocity turbulent boundary-layers are still in use for hypersonic nozzle design. The Sivells-Payne method[47] is commonly used because it is relatively simple in computation. Of course, the real-gas effect on the boundary-layer development is not taken into account. The Persh-Lee method proposed by Preiswerk[8] sets up its computation based on the axisymmetric momentum equations including real gas effects.

On a whole, the conventional nozzle design methods utilize different techniques to evaluate the thickness of the boundary layer and determine the displacement-thickness for correction, and are known as the combination of the Method of Characteristics and Boundary-Layer approach (MOC/BL). The working core is to make proper boundary layer corrections on the designed potential flow nozzle contours.

3.2.6. Limitations on the MOC method family

The nozzle design methods discussed above are called as the conventional MOC-based analytic methods or the MOC method family. The contour design of hypersonic nozzles with the ideal gas assumption has become quite mature since the 1940s. By the 1980s, the test flow generated from the nozzles designed with the MOC/BL algorithms becomes easy to verify and evaluate with the CFD computation.[48] Doing so, Benton et al.[18] had analyzed limitations on the MOC method family in designing axisymmetric hypersonic nozzles in detail. He pointed out that these class of the MOC/BL nozzle design methods become unreliable for the hypersonic nozzle at the Mach number higher than 8, but is effective for designing the nozzles with Mach numbers less than or equal to 8.

As the Mach number increases, the boundary layer gets thicker and thicker and its thickness becomes comparable to the nozzle diameter. The nozzles designed with the MOC/BL method hardly meet the design requirements. By solving Navier-Stokes (N-S) equations at Mach numbers of 13.5 and 17 respectively, Benton[48] found that the CFD results are less consistent with experimental results. The numerical evaluation is not improved satisfactorily even when changing of the specific heat capacity ratio was considered. As a result, the nozzle flow quality is not acceptable or the Mach number distribution of the nozzle flow does not meet the design parameters. So, the thin boundary-layer assumption is considered to be unreasonable for high Mach number flows.

Compression waves are present in all the hypersonic nozzles that were analyzed by Benton, but always occur near the inflection point, which results in irregular flow rates. He recommends that a design method should be improved by using the PNS equations. Candler and Perkins[49] analyzed the reflection of the characteristic curves in the thick boundary layer of the hypersonic nozzles. It is recalled that the MOC/BL method is based on a basic assumption that the boundary-layer flow and the nozzle core flow are not coupled with each other. For the case of the low Mach number nozzle with a thin boundary layer and a short geometrical nozzle shape, this assumption is reasonable. The reflection position of flow characteristic curves is approximately located on the inviscid contour. However, after the reflection from the thick boundary layer, there is a certain lag between the actual reflection characteristic curves and the calculated characteristic curves reflected through the potential flow boundary. This wave process means the reflection of the characteristic curves actually occur inside the thick boundary layer instead of at the potential flow boundary line, which results in that the actual reflected characteristic curves lag behind the design reflected characteristic curves, as shown in Fig. 10, thereby the flow quality of the MOC/BL-based nozzles is reduced.[30,50]

The nozzle designed with the conventional MOC/BL methods has a deviation in the characteristic reflection, which makes the expansion waves in the damping section incompletely eliminated, and thus results in a non-uniform test flow at the nozzle exit. Benton et al.[18] believed that this effect appears visible when the Mach number is about 7 if the MOC/BL methods are used to design hypersonic nozzles. When the turbulent boundary layer transition takes place, the proportion of the boundary-layer thickness to the nozzle radius rises, and becomes larger and larger as the Mach number increases. Therefore, to improve the flow quality of hypersonic nozzles, the advanced nozzle design methods need to be developed, with which the boundary layer development could be accurately evaluated.

3.3. CFD-based design-by-analysis method

In order to break through limitations on the conventional MOC/BL methods for the hypersonic nozzle design, Korte et al.[19–25] proposed a new design method in which the CFD numerical technique is introduced into the nozzle design process for optimization in the early 1990s. Their optimization algorithm based on the nonlinear least square method[51] is coupled with a high-accuracy CFD solver,[52] and the nozzle contour can be created with least changes in the flow angularity and the Mach number at the nozzle exit. The CFD solver is applied to the PNS equations and ensures high-quality flow solutions at the exit of the MOC/BL-based nozzle. The Korte method is also called as the "design-by-analysis" method by Shope.[29] The main advantage of the Korte method is that the CFD solver of the N-S equations is used to have a correct physical modeling to handle the interaction between the core flow (inviscid flow) and the boundary layer in hypersonic nozzles. During the design process as shown in Fig. 11[22], the CFD solutions of the nozzle flow and the nozzle contour are substituted into the optimization algorithm. The algorithm iterates until a uniform flow is obtained at the nozzle exit. The accuracy of the Korte method depends on the performance of the CFD solver and the optimal solution that the optimization algorithm can achieve. In addition, this design method can be used to design not only the axisymmetric nozzles but also the 3D nozzles by using a three-dimensional CFD solver.

Subsequently, Korte et al.[21,23] improved the CFD-based design method for hypersonic nozzles and the improved method makes it possible to complete the entire design process with the CFD technology without utilizing the MOC/BL-based initial nozzle contour. In the design process with the improved Korte method, the Mach number distribution along the axis is optimized according to the parameters, such as the Mach number and the flow angularity at the exit. The Mach number distribution along the axis in the optimization objective function remains the usage of the formula of the Sivells method.[14] The Korte method removes the constraints of the conventional MOC and the designed nozzle flows are of higher quality than that generated by the theoretical methods. However, it is obvious that the "CFD-based design-by-analysis" method highly depends on the numerical simulation accuracy. Therefore, the CFD-based numerical method needs to be further validated for the high-enthalpy nozzle flows with non-equilibrium chemical reaction models.

In 2000, Korte[26] developed his code to deal with the high-temperature flow and the specific heat capacity ratio is changeable with the flow temperature through Sivells' irrotational flow MOC.[14] This code is used by Gaffney[31] to design the nozzles for the HYPULSE shock tunnels which are the high-enthalpy ground test facilities at the NASA Langley Research Center in the United States. The tunnel nozzles can generate high-enthalpy flows at $Ma = 15$ and are to use for evaluating scramjet engine performances. This code is written by combining the rotational flow MOC with a solver of Euler equations. Its advantages are that the code is not limited by the structure (or programming language) of the existing codes, does not need to go through the program line by line for possible modifications, and runs relatively fast. And also, the number of the points defining the nozzle contour depends on the number of the specified initial points. The viscosity effect is not

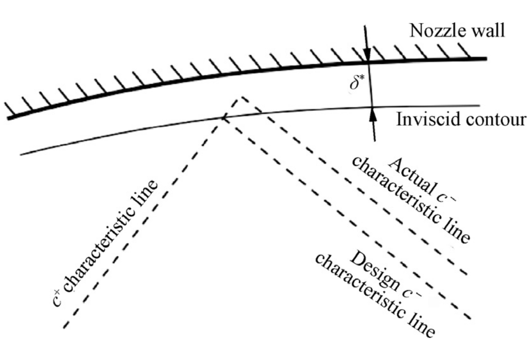

Fig. 10 Lagging between actual characteristic curves reflection and design characteristic curves reflection.[30,50]

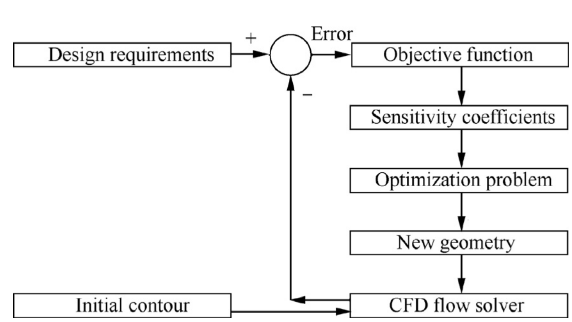

Fig. 11 CFD-based design optimization process.[22]

considered in the Gaffney design method and the non-uniform distribution of the flow properties of the HYPULSE nozzles is induced by the viscosity effect because that only the rotational equations are solved. Final viscosity-corrected nozzle contour is generated by iterating the MOC solution with the solutions of full N-S equations.[53] Theoretically, the Korte method allows any combination of the CFD solvers with optimization algorithms in its design process. Keeling,[27] Tolle,[28] Shope[29] and Chan et al.[30] did their improvements on the Korte method by trying different CFD solvers and optimization algorithms. In his early work, Korte et al. developed a nozzle design program called as CAN-DO.[22] The cubic spline interpolation was applied in the program for contour design, the optimal test flow was explored by adjusting the spline interpolation points, and evaluated with a PNS solver.

During the recent years, Chan et al.[30] adopted an open-source RANS (Reynolds Averaged Navier-Stokes) solver, and Eilmer[54] combined it with the simplex optimization method.[55] The goal of the design optimization is to obtain the nozzle flow with the minimum flow angularity and the minimum deviation from the design Mach number at the nozzle exit. With this program, Chan et al.[30] designed the nozzles of Ma = 4, 7, 10 for the T4 shock tunnel[50,56] at the University of Queensland, Australia, and carried out the comparison between numerical simulations and experimental data. The numerical simulations of the nozzle flow showed that the test flow in the core area has a good uniformity; the discrepancy of the Mach number distribution at the exit is less than 0.5%; the flow angularity is less than 0.05°; the differences in both the static temperature and the flow rate are less than 1%; and the changes in both the pitot and the static pressure are less than 2%.

In addition, the measured pitot pressures on several stations downstream from three optimized nozzle exits match well with the CFD numerical simulations. These comparisons demonstrated that the "CFD-based design-by-analysis" method is effective for the nozzle design at hypersonic high-enthalpy flow conditions. Figs. 12 and 13 present both the computational and optimized results produced by Chan et al.[30] for the nozzle of Ma = 7, including a comparison of the grid effect and the MOC design results.

The good results for the design of hypersonic high-enthalpy nozzles were achieved with the "CFD-based design-by-analysis" methods, but the design process turns out to be complicated. In particular, the design process adopts the CFD-coupling with optimization algorithm, and a suitable optimization algorithm has to select because different N-S solvers are used, which make the method difficult to be popularized in engineering applications.

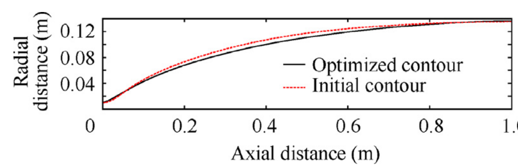

Fig. 12 Comparison of initial contour and final contour of nozzle optimized with method proposed by Chan et al. (Ma = 7).[30]

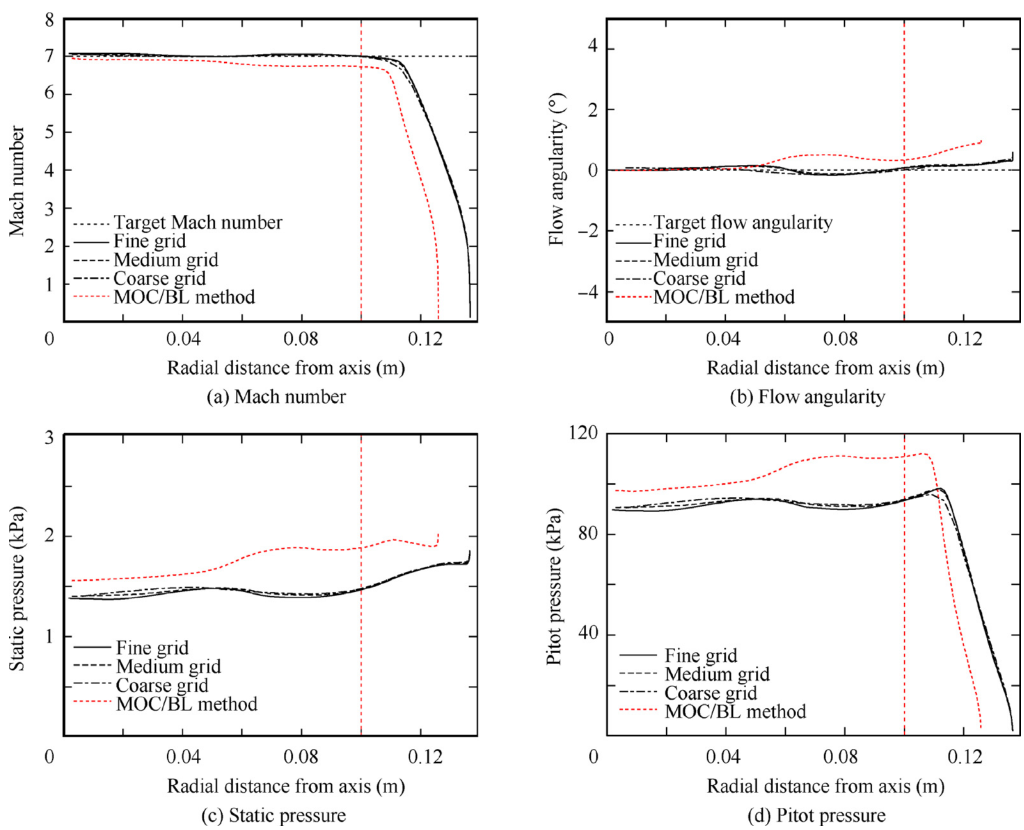

Fig. 13 Comparison of flow properties at nozzle exit between the "CFD-based design-by-analysis" method of Chan et al. and MOC/BL.[30]

Tang et al.[32] had completed the optimization of several hypervelocity nozzles using the CFD-based MOC design method. Being different from Korte and Gaffney methods, their method does not couple the optimization algorithm with the N-S solver to optimize the nozzle contour. Instead, based on the Sivells' irrotational flow MOC method, they correct and optimize the inviscid flow contour (potential flow) and the high-temperature boundary-layer simultaneously by using the CFD solutions from solving the N-S equations which include the high-temperature, chemical reactions and non-equilibrium effects. It is the separately-solving of the CFD equations and the conventional MOC as well as the optimization algorithm for the nozzle contour optimization makes the design process simple and convenient to use. The boundary-layer displacement-thickness that is obtained by using the CFD results from each iteration step is used as the basis for the boundary-layer correction in the next iteration step. Besides, the flow information along boundaries is also applied to the optimizing computation of the next iteration step. All the initial and boundary conditions used in the MOC method are derived from the nozzle flow data of the CFD simulations. Generally, a satisfactory exit flow can be created within two or three iterations. The evaluation can be carried out by examining whether the flow angularity and the Mach number distribution in the core flow area of the nozzle exit meet the design requirements. Fig. 14[32] shows the flow chart during the CFD-based iterative optimization.

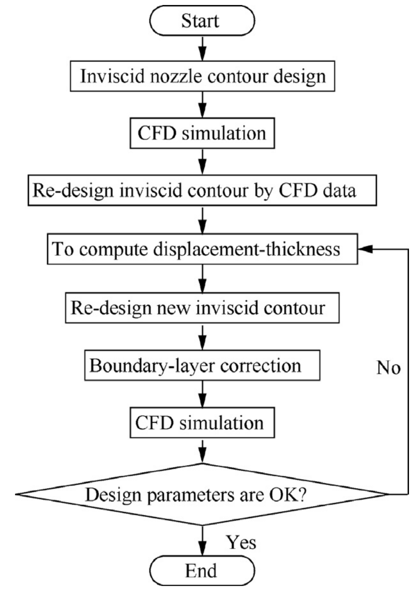

Fig. 14 Flow chart for CFD-based iterative optimization in Tang's work.[32]

In general, the "CFD-based design-by-analysis" methods for the nozzles are roughly divided into two types. One is the Computational Fluid Dynamics/Optimization Methods (CFD/OM) in which the CFD solver is deeply coupled with the optimization algorithm. These methods abandon the MOC analytic design methods and accepts only the MOC design contour as its initial input condition. Compared with the conventional MOC/BL analytic methods, the CFD/OM for the hypersonic nozzle design includes the viscosity effect into the optimization process so that the boundary-layer correction would be unnecessary. The uniform flow can be created and the flow quality can be improved to a certain extent. This is true especially for the nozzle flow where the boundary layer is thick and the Mach number is large. Of course, several criticisms on the method were also reported. First, due to limitation of the number of the control points used in optimization process, much higher computational cost is taken than the MOC/BL method. Second, the code must be re-modified for a given situation, and it is not very convenient for other users to apply. Third, the designed nozzle quality entirely depends on the accuracy of the CFD results and the optimization algorithm. For hypersonic nozzles, especially the high-enthalpy flow nozzles, the chemical reacting non-equilibrium flow occurs and its complexity increases as the flow Mach number increases. Therefore, improving the accuracy of CFD simulations has been a challenge for decades. The robust of the optimization algorithms also need to be improved further according to the actual design situation. The other type is the CFD/MOC/OM in which the CFD solvers, the conventional MOC analytic methods, and optimization methods are combined in the designing process, but not coupled with together. Separately-solving the CFD equations, the mature MOC design methods together with nozzle contour optimization could reduce technical difficulties in the computational process, and produce a high robustness in the combined algorithms and greatly reduce the computation workload. In the following chapter, the CFD/MOC/OM will be discussed in detail.

4. Design technologies of high-enthalpy flow nozzles

4.1. Real-gas effects on nozzle design

In the 1950s and 1970s, a series of papers were published on the design methods of supersonic and hypersonic nozzles with ideal gas assumption[57–60] and some books presented systematical reviews on the progresses.[61,62] When the nozzle exit radius is selected as the characteristic length and the boundary-layer thickness is much smaller than the characteristic length, the nozzle flow can be regarded as an inviscid flow. So, the potential flow nozzle contour can be taken as the initial step and the hypersonic nozzle design can be completed by estimating the boundary-layer displacement-thickness and correcting with the boundary layer effect.

For hypersonic nozzles at low Mach numbers, satisfactory results can be achieved with the conventional design methods. However, the real-gas effects must be considered at the high-enthalpy flow condition and it is the case for the high-enthalpy shock tunnels. If the reservoir gas temperature exceeds 600 K, the gas molecule vibrations will be excited and the specific heat capacity ratio γ becomes a function of the static temperature rather than a constant. When the temperature reaches 2000 K, the oxygen will begin to dissociate. The oxygen will be almost completely dissociated into oxygen atoms and the nitrogen begins to dissociate into nitrogen atoms after the temperature rises over 4000 K. Therefore, for the high-enthalpy flow nozzle design, obvious errors will occur and the conventional optimization methods will no longer be effective if the ideal gas assumption is still used. The specific heat capacity ratio γ is the parameter that reflects the ratio of the flow enthalpy to the internal energy, and varies with the local temperature. Air at the ambient condition is of $\gamma = 1.4$, and its γ can remain unchanged as long as the temperature and pressure do not go up too much. At the early development stage of the high-enthalpy flow nozzle design, the γ's impact varying with the temperature and pressure was considered, but the air dissociations and ionizations were ignored. This assumption defines a semi-perfect gas that is still suitable for the state equation that takes into the account the thermodynamic properties (such as the specific heat capacity ratio) varying with temperature and pressure.

Fig. 15 shows the specific heat capacity ratio (c_v is the specific heat at constant volume) varies with the temperature and pressure, and the pressure effect vanishes after the temperature reaches to 600 K as shown in the left figure, but it varies with the temperature as shown in the right figure. The gas state parameters in compressible gas flows are closely related with γ, therefore, the wind tunnel test is generally required to maintain the same γ as in the real flight condition. For the hypersonic nozzle design, the γ's variation must be considered when the Mach number reaches 7 and becomes necessary when the test flow velocity reaches $Ma = 10$ while γ of the test gas varies between 1.15 and 1.2. The large γ variation will affect not only wind tunnel experimental results, but also the nozzle design of hypersonic wind tunnels.

Johnson and Boney[63] clearly explained the effect of γ on the nozzle contour and Fig. 16 presents his comparison of the nozzle contours at different Mach numbers and different specific heat capacity ratios. Obviously, the flow Mach numbers at the nozzle exit are quite different even for the nozzles of the same area ratio if the specific heat capacity ratios are different. However, the work done by Johnson et al. was dedicated to that the fixed area ratio is a comparing object for the change in the nozzle contour, and there are some simplifications in their work. Actually, the area ratio of the high-enthalpy flow nozzles is also affected by the real-gas effects, therefore the difference could be even large. This physical issue had been explained in Tang's work.[32]

4.2. MOC with a variable specific heat capacity ratio

In the 1950s, the real-gas effect on the hypersonic nozzle design was investigated by Guentert and Neumann,[16] Enkenhus and Maher,[17] Erickson and Creekmore,[64] Johnson et al.[60,63] The inviscid flow nozzle contour is computed first with the MOC method and the change of thermodynamic properties of the test gas was considered later. For example, the variable γ is taken as an impact parameter and the nozzle contour is finally determined by superimposing the boundary layer displacement-thickness onto the inviscid flow nozzle contour. Because the variable γ is only a parameter considered regarding to real gas effects, it is indeed a big simplification, however, such the design algorithm is a realistic and practical method for the high-enthalpy flow nozzle design in the early time.

Guentert and Neumann[16] proposed a set of the design methods to gain the inviscid flow nozzle contour of the high-enthalpy axisymmetric condition. Their methods calculated the test flow gas with chemical reactions by applying the MOC with variable isentropic index. Subsequently, Enkenhus

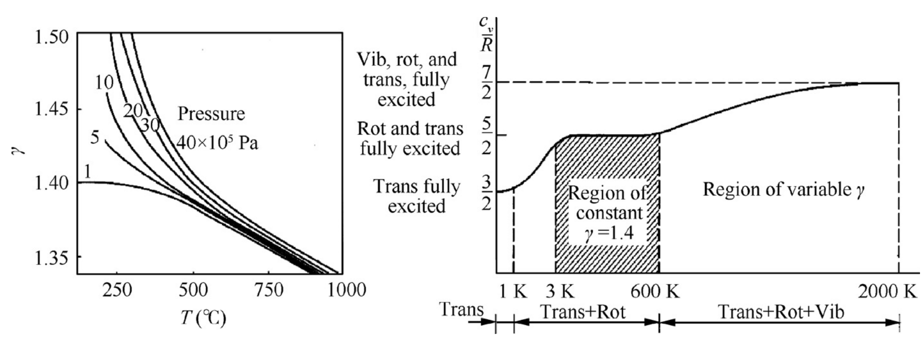

Fig. 15 Effect of temperature and pressure changes on specific heat capacity ratio.[38]

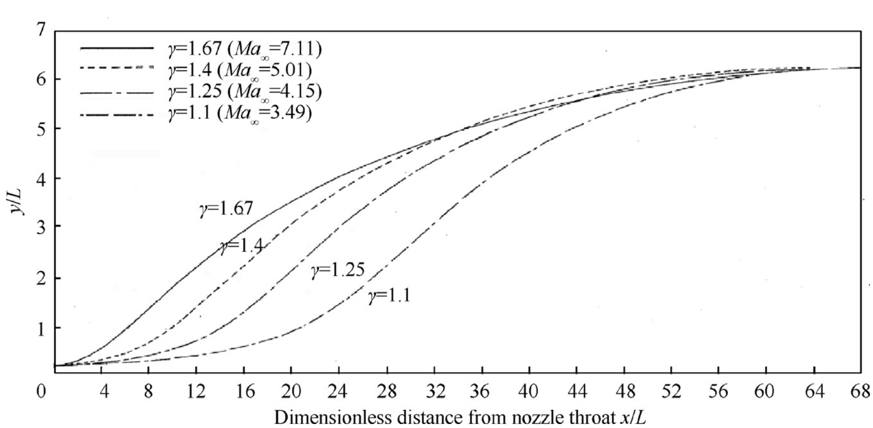

Fig. 16 Effect of γ on nozzle contour in case of same area ratio (L is length of nozzle).[63]

and Maher[17] proposed a similar method for predicting the axisymmetric nozzle contour of the high-enthalpy flows, and independently developed a computation program including the real-gas effect. Johnson et al.[60] improved the above-mentioned methods by directly applying the thermodynamic data of the isentropic expansion to the MOC computational code, and designed a high-enthalpy axisymmetric nozzle at the Mach number of 17 with total-temperature of 2800 K. Korte[26] developed a simplified design method with a variable γ and the potential flow nozzle contour is designed by the Sivells method. In his nozzle design process with variable γ as shown in Fig. 7, the flow area $HIEG$ was first calculated and the specific heat capacity ratio γ_1 was taken to be the average value of this area so that the initial expansion contour HG can be created. Then, the flow area $ABCD$ is calculated, and the specific heat capacity ratio γ_2 is the average value of this area, and then, the damping section contour AD is created through computation. Finally, the area ratio is calculated with a changed γ, so that the length of straight-line GA can be determined. Making the initial expansion section contour HG, the damping section contour AD, and the linear segment GA match with each other results in that the requirements of the following area ratio formula are met. In the formula, the equilibrium flow assumption is adopted so the area ratio formula can be derived based on the isentropic relation:[65]

$$\frac{A^*}{A} = \left(\frac{A^*}{A}\right)_{\mathrm{i}} F'(T_0, Ma) \tag{19}$$

$$\left(\frac{A^*}{A}\right)_{\mathrm{i}} = Ma\left(\frac{\frac{\gamma_{\mathrm{i}}+1}{2}}{1+\frac{\gamma_{\mathrm{i}}-1}{2}Ma^2}\right)^{\frac{\gamma_{\mathrm{i}}+1}{2(\gamma_{\mathrm{i}}-1)}} \tag{20}$$

where A^* and A are the nozzle throat area and the nozzle exit area, respectively; $(A^*/A)_{\mathrm{i}}$ is the area ratio under the ideal gas assumption, the subscript "i" indicates the ideal gas; Ma is the design Mach number. Without taking the variable γ into account, we have

$$F'(T_0, Ma) = 1 \tag{21}$$

If the variable γ is considered, $F'(T_0, Ma)$ expressed with the formula below:[65]

$$\begin{aligned} F'(T_0, Ma) = {} & 1 + B_4(Ma)b_{\rho_0} + C_4(Ma)\tfrac{c_{\rho_0}}{RT_0^2} \\ & + D_4\left(\tfrac{\theta}{T_0}, Ma\right)\left(\tfrac{\theta}{T_0}\right)\mathrm{e}^{-\left(\frac{\theta}{T_0}\right)} \end{aligned} \tag{22}$$

where,

$$B_4(Ma) = \frac{\gamma_{\mathrm{i}}+1}{2(\gamma_{\mathrm{i}}-1)}\left[B_1(Ma) - B_1(Ma^*)\right] \tag{23}$$

$$\begin{aligned} C_4(Ma) = {} & \tfrac{\gamma_{\mathrm{i}}+1}{2(\gamma_{\mathrm{i}}-1)}\left[C_1(Ma) - C_1(Ma^*)\right] \\ & + \tfrac{\Gamma_{\mathrm{i}}-\gamma_{\mathrm{i}}}{\gamma_{\mathrm{i}}}\left[\left(\tfrac{\rho}{\rho_0}\right)_{\mathrm{i}}^{3-2\gamma_{\mathrm{i}}} - \left(\tfrac{\rho^*}{\rho_0}\right)_{\mathrm{i}}^{3-2\gamma_{\mathrm{i}}}\right] \end{aligned} \tag{24}$$

$$\begin{aligned} D_4\left(\tfrac{\theta}{T_0}, Ma\right) = {} & \tfrac{\gamma_{\mathrm{i}}+1}{2(\gamma_{\mathrm{i}}-1)}\left[D_1\left(\tfrac{\theta}{T_0}, Ma\right) - D_1\left(\tfrac{\theta}{T_0}, Ma^*\right)\right] \\ & + \mathrm{e}^{-\left(\frac{\theta}{T_0}\right)\left[\left(\frac{T_0}{T}\right)_{\mathrm{i}}-1\right]}\left\{\left(\tfrac{T_0}{T}\right)_{\mathrm{i}} + \tfrac{T_0}{\theta}\left[1 - \tfrac{(\gamma_{\mathrm{i}}-1)^2}{2\gamma_{\mathrm{i}}}\left(\tfrac{\theta}{T_0}\right)^2\left(\tfrac{T_0}{T}\right)_{\mathrm{i}}\right]\right\} \\ & - \mathrm{e}^{-\left(\frac{\theta}{T_0}\right)\left[\left(\frac{T_0}{T^*}\right)_{\mathrm{i}}-1\right]}\left\{\left(\tfrac{T_0}{T^*}\right)_{\mathrm{i}} + \tfrac{T_0}{\theta}\left[1 - \tfrac{(\gamma_{\mathrm{i}}-1)^2}{2\gamma_{\mathrm{i}}}\left(\tfrac{\theta}{T_0}\right)^2\left(\tfrac{T_0}{T^*}\right)_{\mathrm{i}}^2\right]\right\} \end{aligned} \tag{25}$$

$$B_1(Ma) = 2\left[\left(\frac{p}{p_0}\right)_{\mathrm{i}} - \left(\frac{\rho}{\rho_0}\right)_{\mathrm{i}}\right] + \frac{\gamma_{\mathrm{i}}-1}{\gamma_{\mathrm{i}}}\left[1 - \left(\frac{p}{p_0}\right)_{\mathrm{i}}\right] \tag{26}$$

$$\begin{aligned} C_1(Ma) = {} & 2\tfrac{\Gamma_{\mathrm{i}}}{\gamma_{\mathrm{i}}}\left(\tfrac{\rho}{\rho_0}\right)_{\mathrm{i}}^{2-\gamma_{\mathrm{i}}}\left[1 - \left(\tfrac{\rho}{\rho_0}\right)_{\mathrm{i}}^{1-\gamma_{\mathrm{i}}}\right] \\ & - 3\tfrac{\gamma_{\mathrm{i}}-1}{\gamma_{\mathrm{i}}}\left[1 - \left(\tfrac{\rho}{\rho_0}\right)_{\mathrm{i}}^{2-\gamma_{\mathrm{i}}}\right] \end{aligned} \tag{27}$$

$$\begin{aligned} D_1\left(\tfrac{\theta}{T_0}, Ma\right) = {} & \tfrac{\gamma_{\mathrm{i}}-1}{\gamma_{\mathrm{i}}}\Big[1 - \mathrm{e}^{-\left(\frac{\theta}{T_0}\right)\left[\left(\frac{T_0}{T}\right)_{\mathrm{i}}-1\right]} \\ & \cdot\left\{1 - (\gamma_{\mathrm{i}}-1)\left(\tfrac{\theta}{T_0}\right)\left(\tfrac{T_0}{T}\right)_{\mathrm{i}}\left[\left(\tfrac{T_0}{T}\right)_{\mathrm{i}} - 1\right]\right\}\Big] \end{aligned} \tag{28}$$

$$\Gamma_{\mathrm{i}} = \gamma_{\mathrm{i}}^2 - 3\gamma_{\mathrm{i}} + 3 \tag{29}$$

where, subscripts "0" and superscript "*" are for the stagnation parameters and the flow parameters at nozzle throat, respectively. b and c are the characteristic parameters of flow. θ is the vibrational characteristic temperature that depends on the gas components. p/p_0, ρ/ρ_0 and T/T_0 are obtained with a series of isentropic relations of the one-dimensional flow. The detailed derivation process is introduced by Eggers.[65] In fact, the effect of the variable γ on the upstream flow in the radial area (especially, near the throat) is much larger than that on the downstream flow in the damping section. Because the downstream flow is expanding to low temperature and is approximate to the ideal gas flow, it is a proper way to focus on the radial flow area. As a result, a great deal of simplifications was achieved for the high-enthalpy flow because the downstream flow is basically frozen while the test gas near the throat has a complicated composition of the chemically-reacting non-equilibrium states.

Zonars[66] provided a comparison between the theoretical evaluation and the experimental measurements of the nozzle wall pressure in both the equilibrium and the non-equilibrium flows, as shown in Fig. 17 (here, P_{T}, H_{T}, and T_{T} are the total pressure, total enthalpy, and total temperature). Notably, as the total temperature increases, the non-equilibrium effect will become more and more obvious. As a result, the evaluation and analysis of the non-equilibrium nozzle flow becomes more important than the equilibrium one. For the design of the high-enthalpy flow nozzles, using the MOC technology in which only the variable γ is considered will also cause certain design errors. The design error for the hypervelocity nozzles will become so large that it would be imperative to develop new design methods coupling with the CFD technique to meet nozzle design requirements.

4.3. Research on CFD-based design of high-enthalpy nozzles

As the rapid development of the CFD technologies, the numerical solutions of the high-enthalpy flows gain the higher and higher accuracy. The feasibility study on the high-enthalpy flow nozzle design was conducted by using the CFD solvers. The evaluation on the real-gas effect on nozzle design becomes easy to implement by using the CFD technique, and this is a significant advantage of the CFD-based nozzle design method. Actually, if computational resources allow, we can use the advanced CFD solver to solve N-S equations implemented with appropriate chemically-reacting models in which the real gas effects are fully considered and such the high-accuracy

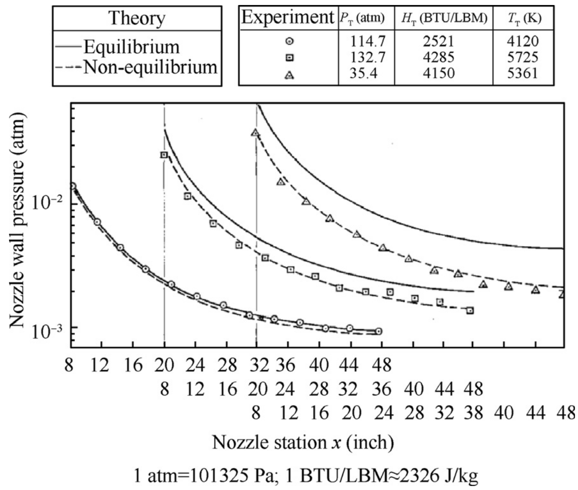

Fig. 17 Comparison of nozzle wall pressure.[66]

solutions can play a significant role in the high-enthalpy flow nozzle design.

Korte[52] first applied the CFD simulation to the optimization design of high-enthalpy flow nozzles and his CFD-based optimization method is a combination of a CFD solver with an optimization algorithm, therefore, the design process becomes much easier than before. What we need to do next is to replace with the CFD solver implemented with a non-equilibrium chemical reaction model. Then, high quality nozzle contour can be achieved. Korte designed a Mach 15 nozzle for a helium-driven shock tunnel with the nozzle design program called CAN-DO,[22] in which the PNS solver is used for the nozzle flow evaluation but the real-gas effect is ignored. Subsequently, Korte et al.[21] and Hollis[67] designed the 22-inch helium tunnel nozzle using the CAN-DO program at the NASA Langley Research Center. They simply modified only the state equation in the CFD solver and the design results obtained meet the application requirements in that time. In 2000, Korte[26] further improved the area ratio of the high-enthalpy flow nozzle in his design method to address the real-gas effect. The main design process is to solve a steady one-dimensional flow at a given Mach number, and iterate until a desired nozzle exit condition can be reached (see Fig. 18).

Shope and Tatum[68,69] applied the CFD-based optimization design methods to high-enthalpy flow nozzles by calculating chemically-reacting flows, and developed their new characteristic tracking code. Multi-optimization algorithms, such as the least-squares optimization[70] and the spline-interpolation optimization[71] were integrated in it, controlled and shifted through scripts. The Shope's nozzle design code calculates the expansion of the chemically reacting flows with the modified spline points from the code of Korte et al. His CFD solver accepts a computation method called the Data-Parallel Line Relaxation (DPLR) to solve the N-S equations with chemical reactions.[72–74] When the DPLR method is used to simulate the calorimetric perfect gas flow, the frozen gas components include CO_2, H_2O, N_2, O_2, and NO. The percentage of each component is obtained with the CEA96 program[75] of the NASA Glen Research Center (GRC).

The Shope optimization design process can be divided into two steps. In the first step, the MOC-based Sivells method is adopted to design the initial nozzle contour by assuming that the gas flow within the nozzle is the thermal-perfect gas or the calorically-perfect gas. The design parameters at the nozzle exit (such as Mach number and velocity), being approximate to the thermodynamic equilibrium states, can be obtained by setting the γ and other gas constants. In the second step, the obtained initial nozzle contour is further optimized by removing the application limitation on the MOC. The key design process is to correct the spline interpolation in the nozzle contour coordinates. The spline curve goes through only a few of the defined points along the initial nozzle contour, rather than interpolates all the coordinate points of the nozzle contour. The spline interpolation correction allows a sufficient number of the coordinate points to be used for defining the high-accuracy nozzle contour, while only a few nodes (design parameters) are used in such the correction. Then, these design parameters were used in the repeated optimization process (namely the MOC design process) to further improve the quality of the high-enthalpy test flow. Finally, Shope designed a Mach 6 nozzle for the Aerodynamic and Propulsion Test Unit (APTU) at the Arnold Engineering Development Center (AEDC) using this code, and the obtained results as shown in Fig. 19 look good. However, this method was not verified further for high-enthalpy flow nozzles at even high Mach numbers.

Gaffney[31,76] designed a nozzle for NASA's HYPULSE Shock-Expansion Tunnel (SET). The nozzle flow at Ma = 15 was simulated with a Euler equation solver by adjusting γ. Although the conventional MOC design process is adopted, the design results obtained meet well the

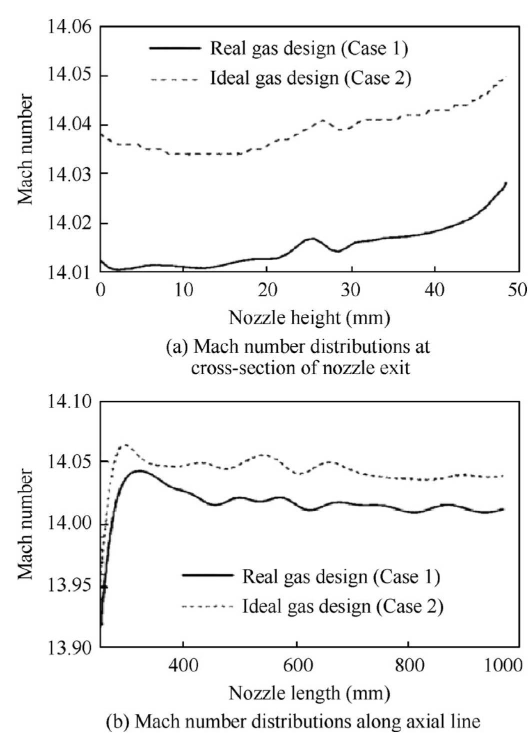

Fig. 18 Numerical comparison for nozzle design with real-gas and with ideal-gas.[26]

requirements. His optimization design code implemented with chemical non-equilibrium CFD solver proposed by Chan et al.[30] is also applied into the nozzle design of $Ma = 10$ for the T4 high-enthalpy shock tunnel in Australia, and relatively satisfactory test flows were obtained. However, it was found from their comparison that the design results, numerical computations and experimental data of the nozzles at the Mach numbers of $Ma = 4, 7$ agree better than the case at the Mach number of $Ma = 10$. This indicates that the real-gas effect plays a more and more important role in the hypersonic nozzle design when the Mach number of high-enthalpy ground facilities is increasing.

The combination of the CFD solvers with Optimization algorithms (CFD/OM) and the conventional MOC implemented with the Boundary-Layer correction (MOC/BL) are two methods widely used in designing nozzles for hypersonic wind tunnels. When the boundary layer of the nozzle flows becomes quite thick, the high-quality nozzle can be obtained with the CFD/OM methods. For impulse high-enthalpy test facilities, the nozzle flow has a cold wall boundary and the extremely-high pressure, and a thinner boundary-layer is also developed. Satisfactory design results can be still achieved by using the classic MOC/BL algorithms. This is because that the finer grid can be accepted with the classic MOC/BL to accurately simulate the thin boundary-layer, and the grid is more time-consuming and expensive to use the CFD/OM methods than the MOC/BL.

Tang et al.[32] improved the classic MOC/BL method to design the hypervelocity nozzle and the real-gas effect was taken into account. The nozzle flow is simulated with the CFD code, and the numerical flow data is extracted and then substituted into the MOC/BL for optimization iterative. Therefore, their method is different from the conventional MOC/BL and can be defined as the CFD/MOC/OM algorithm that does not include the complex CFD embedding optimization format method CFD/OM, and just combines the CFD simulations, the MOC/BL and optimization algorithms together in the design process, and works like a simpler combined optimization method. Tang et al.[32] carried out the design for a high-enthalpy nozzle at a Mach number of 17 with the CFD/MOC/OM. The correction of the high-temperature gas effects is used by considering not only the variable γ but also the species of non-equilibrium flow gases. The average value of the flow area $ABCD$ (see Fig. 7) is selected in computation since the specific heat capacity ratio does not change in this area. In the flow area $TIEG$, the variable γ depending on the

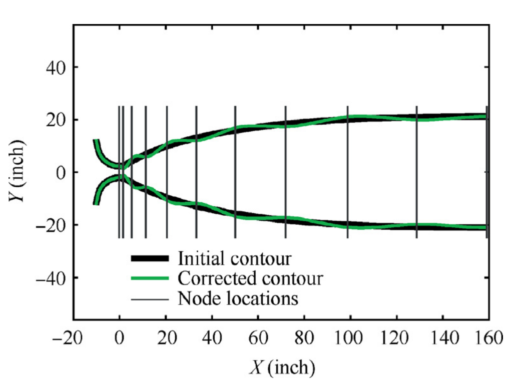

Fig. 19 Spline interpolation correction of nozzle contour.[69]

temperature is adopted in the CFD simulations and expressed with the NASA fitting polynomial[77,78]

$$\gamma = a_1 T^{-2} + a_2 T^{-1} + a_3 + a_4 T^1 + a_5 T^2 + a_6 T^3 + a_7 T^4 \tag{30}$$

where the coefficients in the polynomial (see Eq. (30)) can be obtained through the table look-up method and the flow information of gas species is extracted from the high-accuracy non-equilibrium CFD simulations. Calculating the area ratio requires the vibration characteristic temperature which is also affected by the change of gas species. The species components of the corresponding area are extracted from CFD simulations to calculate the vibration characteristic temperature. Then, the new area ratio can be worked out. The initial flowfield data and computational conditions are extracted from the simulated nozzle flowfield obtained by a CFD computation using the inviscid nozzle contour. In the iterative computation, the flow parameters will gradually converge. Generally, a satisfactory result can be achieved through two or three iterations. Even for the hypervelocity nozzle with a very high-enthalpy, the number of iterations may be increased, but is acceptable.

In Tang's work[32] on the design of large-scale high-enthalpy flow nozzles, two techniques adopted are critical. One is the fitting of the area ratio of the hypervelocity nozzles based on the CFD data. Since the conventional analytic formula for the nozzle area ratio is suitable for equilibrium flows, the actual Mach number calculated by the non-equilibrium CFD solvers has obvious deviations from the design Mach number. In order to obtain more accurate results of the potential flow, the area ratio obtained based on the equilibrium flow is corrected with the fitting method. The triangle mark in Fig. 20 is the area ratio obtained with Eq. (19) for different Mach numbers. The square mark is the actual Mach number at the nozzle exit, and the area ratio corresponds to the non-equilibrium CFD simulations. Curve fitting is carried out according to the relation between the theoretical and simulated Mach numbers at the nozzle exit. The new area ratio (marked with a round mark) corresponding to each Mach number is extracted from the fitted curve, and then substituted into the code to redesign the nozzle contour. After several iterations, the final area ratio obtained makes the design Mach number consistent with the actual Mach number. In fact, the fitting process of the area ratio is an accurate evaluation on the boundary layer development of the non-equilibrium nozzle flow.

Another technique is the nozzle contour optimization that is realized by using the endpoint slope-controllable Hermitian spline interpolation. On the whole, the obtained spline curve is very similar to the reference point given with the original design. The problem of the local fluctuation at the reference point is overcame in the optimized curve solvers. In fact, during the design process of high-enthalpy flow nozzles to ensure flow quality at the nozzle exit, the nozzle contour must meet the following conditions: (A) monotonically increasing; (B) smooth first-order derivative (continuous second-order derivative); (C) the nozzle contour going through the defined coordinate point, and the first-order and second-order derivatives being zero or controllable at the defined point; (D) the refined connection control at the throat section and the segmented connection points can be achieved to prevent the occurrence of any interruption point along the nozzle contour. When the existing optimization method is used for global fitting,

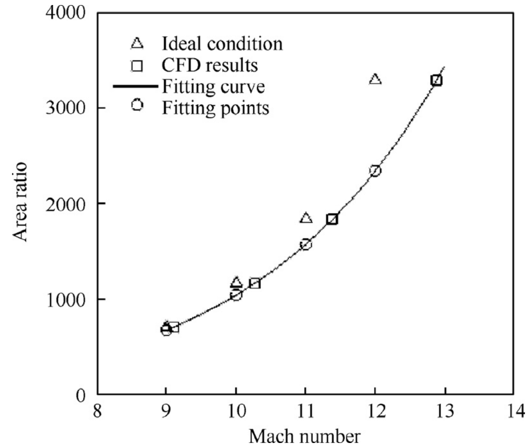

Fig. 20 Comparison of area ratios obtained by theoretical and CFD calculations.[32]

the local point-by-point fitting or interpolation is difficult to freely control the spline interpolation.

4.4. A potential application technology for CFD/MOC/OM: Data mining

In recent years, data mining technology has received widespread attention, and it has been applied in many fields. Data mining, also called knowledge discovery in databases, in computer science, the process of discovering interesting and useful patterns and relationships in large volumes of data. Therefore, data mining is defined as extracting information from huge sets of data.[79,80] At present, data mining technology has been successfully applied to CFD and has solved some related optimization problems.[81–84] As mentioned above, Tang et al.[32] carried out the design and optimization for the high-enthalpy nozzle at Mach number of 17 using the CFD/MOC/OM (see Fig. 21). In the process of optimization, the flow information of gas species, boundary and initial conditions are extracted from the high-accuracy non-equilibrium CFD simulations. The accurate extraction of flowfield information will greatly improve the optimization efficiency. Therefore, the author believes that data mining technology is a potentially effective tool, which can be used to optimize nozzle design based on CFD, i.e., the data mining approaches coupled with the CFD-based optimization method. Unfortunately, it seems that data mining technology has not been applied to the optimization design of high-enthalpy nozzles.

5. Other aided design techniques

5.1. Multi-throat nozzle design techniques

For the hypersonic wind tunnels operating in a wide range of Mach numbers, designing one nozzle for each Mach number requires high costs and large workloads. It is a good strategy that the multiple throats are designed to share one nozzle so that the test flows at different Mach numbers can be created only by replacing the nozzle throat. A Throat Changing Technique (TCT) was proposed to create a special part of the nozzle, in which the throat is located. This work can be done by adding a straight segment at the inflection point of the nozzle expansion section or by re-designing the initial expansion section, while keeping the original supersonic expansion section unchanged. In this way, the nozzle area ratio at a given design Mach number is achieved accordingly, thereby the flow Mach number is changed at the nozzle exit. In engineering applications, the nozzle with multi-throats can be designed in advance so that the test flow at different Mach numbers can be obtained by replacing with different throats.

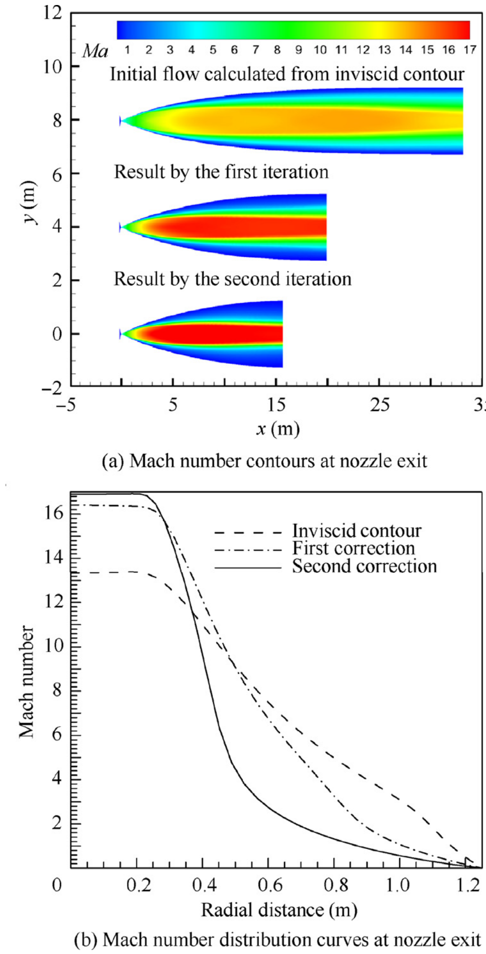

Fig. 21 Iterative optimizations for Ma = 17 nozzle.[32]

The TCT realizes the change of the flow Mach number by changing the throat area on the premise that the nozzle exit area is kept unchanged. For the nozzle flow under different enthalpy conditions, the flow Mach number at the nozzle exit can be generated by using different area ratios. For the ideal gas flow, the throat area is generally determined by Eq. (8). For the high-enthalpy flow, Eq. (19) can be used to determine the throat size. The good design results can be obtained for high-enthalpy flow nozzles at high Mach numbers by using the fitting area ratio processed from the CFD data.

To ensure that the multiple throats can share the same damping section (supersonic expansion section), we need to design several initial expansion sections as the throat parts to meet the area ratio requirement for different flow Mach numbers. The nozzle contour determined with the certain design method should have a straight segment near the inflection point (the turning point of the contour) as shown in Fig. 22. The straight segment on the nozzle contour is the place where the throat part can be replaced without disturbing the flow state in the supersonic expansion section. From the hypersonic nozzle design methods discussed above, the straight segment is short and the short length sometimes may leads to difficulties in throat replacement, especially in the case of the flow expansion from a high Mach number nozzle to a low Mach number one. After completing the initial nozzle contour, the inflection points (the turning points, namely point E or A in Fig. 22) or the straight segment of the nozzle contour must be first found if the multiple throat nozzle is going to design. Then the slope of the straight segment needs to be calculated and the spline interpolation can also be performed based on the nozzle contour data to obtain the slope at each point. Generally, the inflection point is the extreme slope point. Finally, the new throat radius for producing the required design Mach number is obtained by using the area ratio computation without any change in the nozzle exit area. So, a new nozzle design is completed by using the same expansion angle parameter and the inflection point of the new nozzle contour is obtained. At this stage, the slope of the inflection point should be consistent with the original nozzle contour and the length of the line EE' is adjusted so that the inflection point of the initial expansion section of the new nozzle is coincident with point E'.

In engineering applications, the multi-throat nozzle designed with TCT is generally used for high enthalpy shock tunnels. The real-gas effect cannot be ignored and the variable γ needs to be applied during the nozzle design. The design methods for such the high-enthalpy flow nozzle can be referred from the above chapters.

5.2. Truncation operation of the designed nozzles

The high-enthalpy flow nozzle designed under the hypersonic condition is usually quite long, and its length varies also with different nozzle design methods, but is determined mainly by the characteristics of the hypersonic flows. In particular, the prismatic uniform area of hypervelocity nozzle flows is very long itself because the expansion waves propagate downstream much faster than in the radial direction. However, for engineering applications it is desirable to make the nozzle as short as possible without sacrificing the uniformity of the flow. Aiming at shortening hypersonic flow nozzles, some attempts were made in terms of design methods[85–87] and some cases of low Mach numbers have been designed and analyzed. Few literatures have now been reported and published for the nozzle shortening with a Mach number of 10 or higher. Moreover, there are some discussions about the necessity and cost performance of shortening hypersonic nozzles.[88–93] The design methods working in nozzle-shortening are not reported.

After the inviscid flow nozzle contour is corrected with real gas effects it is the time to consider how to adjust the nozzle length to meet engineering requirements, therefore, the truncation of the designed nozzle near the exit is the last issue in the design process. The truncated position on the high-enthalpy flow nozzle can be determined under some acceptable conditions that the cross-sectional Mach number distribution should be kept uniform at the truncated position and the radial velocity component should meet with design index requirements. The nozzle-truncated steps are suggested as follows: Complete the CFD numerical evaluation on flow properties within the final nozzle contour; Create the potential flow boundary-line according to the boundary layer displacement-thickness; Obtain the first-order derivative of the potential flow boundary-line. If the first-order derivative at point C of the nozzle contour (at the end of the displacement-thickness boundary-line, see Fig. 23) is a negative, the position of the extreme value, named point Q can is determined. With the horizontal coordinate x_Q of point Q as the x-coordinate of the truncation point, the truncated position P_{cut} of the final nozzle contour can be determined. At this stage, the flow evaluation must be made again at the new nozzle exit (the nozzle cross-section at truncated position) to confirm whether it meets the design requirements. If the derivative at point C at the nozzle exit is zero, the truncation operation can be conducted on the premise that the test flow quality is met. The final step is carrying out CFD numerical revaluations on the new nozzle contour after nozzle truncation until its performance meets the design requirement.

Theoretically speaking, if there is no error in the boundary-layer correction, the derivative at point C should be zero. However, the correction errors may exist due to the real-gas effects. As a result, the displacement-thickness boundary-line in the high-enthalpy nozzle flow obtained by the CFD solver does not coincide with the initial inviscid flow contour, especially for the case where the derivative at point C is negative. So, it is necessary to perform the truncation operation to obtain the short nozzle and a large core area of uniform test flows.

6. Summary and prospective

The nozzle design theories and methods have reached to a significant level for a hundred years' development since the wind tunnel nozzle design was tried first by Prandtl and Busemann, and the designed nozzles could meet various requirements from all the wind tunnels that had been constructed for testing subsonic, supersonic and hypersonic vehicles from aviation and aerospace industries. Progresses on the theories and methods can be classified into three categories and summarized as follows:

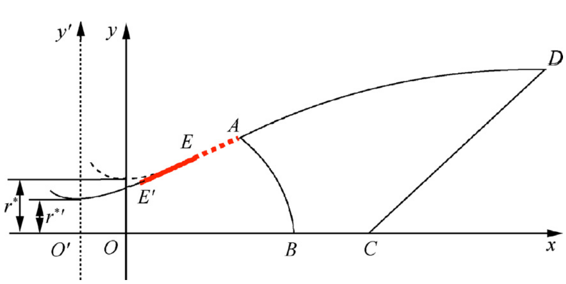

Fig. 22 Schematic diagram for TCT.

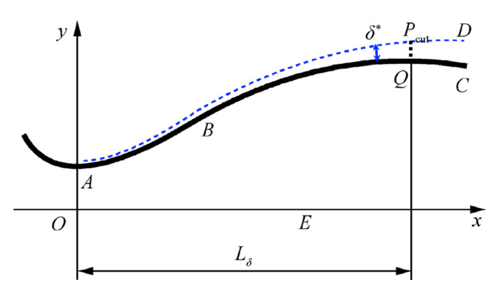

Fig. 23 Schematic diagram for nozzle truncation method.

(1) The first category is the family of the MOC/BL methods that combine the MOC with the boundary-layer correction. The conventional MOC proposed in 1929 is actually the foundation for nozzle design theories and many improved versions deduced from the MOC are still widely in use today. The MOC concept is based on the control of the expansion wave generation and elimination which are the key physical issues in nozzle flow expansion. The boundary-layer correction put the viscosity effect in consideration. Therefore, the good design results can be obtained for supersonic wind tunnel nozzles, even for conventional hypersonic wind tunnels that generate the cold test flows. Some improved methods had developed for the high-enthalpy flow nozzles, but hardly meet evaluation standards.
(2) The CFD-based optimization methods (CFD/OM) are taken as the second category in which optimization methods are closely coupling with CFD solvers for high-enthalpy flow nozzle designs. The real-gas effect is included with numerical solutions obtained by solving the nozzle flow governing equations implemented with chemical reaction models, and the nozzle contour further improved with the optimization algorithms. Since its first appearance in the early 1990s, the CFD/OM methods have been evolving with the continuous improvement of the CFD technologies and various optimization algorithms for decades. More satisfying design results can be obtained for hypersonic high-enthalpy nozzles than the MOC/BL methods, and actually, the design accuracy can reach a high level that the modern CFD technology could reach if the computational resources are powerful enough. However, some criticisms on the CFD/OM methods had been reported, such as the coupling technology of the CFD solver with the optimization algorithm is demanding, the computational cost is high and it is not easy to widely apply in engineering.
(3) The CFD/MOC/OM is considered as the third category in which the CFD/OM is embedded into the MOC design process to avoid the criticisms on the second category for high-enthalpy flow nozzles. The initial nozzle contour based on the potential flow is designed first with the MOC techniques, and the CFD simulations of the chemically-reacting non-equilibrium flows are then used to involve real gas effects and the boundary-layer correction. In its design process, the CFD data are obtained first in each iteration step, then the flow information is collected as the initial and boundary conditions of the MOC design, and finally one iteration step is completed with the optimization solver for the new nozzle contour. Satisfactory optimization results can be obtained with a few iterations. The CFD/MOC/OM does not require the coupling technology of the CFD solver with the optimization algorithm, therefore the method is simple and robust, easy to apply in engineering.

During the recent decades, the hypersonic flight vehicles have been recognized as the key technology for the future aviation and aerospace industries. Therefore, the increasing requirement on hypersonic ground test facilities is foreseeable because flight tests are much expansive and inefficient. This demand gives rise not only the performance improvement on most hypersonic high-enthalpy wind tunnels developed in the last century, but also motives the development of new hypervelocity wind tunnels generating high-enthalpy test flows of high quality. Therefore, developing new design methods to meet the ever-increasing demands on high quality nozzles is still a challenge. By reviewing the research progress achieved in the past hundred years, the following aspects seem worth of further exploring in future from our viewpoint.

(1) The MOC/BL design method alone can no longer produce satisfactory nozzle contours for hypervelocity and high-enthalpy wind tunnels, but high-quality nozzles cab be still obtainable with the aid of high-accuracy CFD simulations. Accurate experimental data of the high-enthalpy flows can be also used to provide to reasonable MOC conditions setting, thereby further improving or correcting the design deviation brought by real-gas effects. In addition, setting the axial velocity or Mach number distributions based on the Sivells method is useful for a nozzle design to obtain relatively satisfactory results.
(2) Great achievements have been gained in the CFD research area during the recent decades, however, it is still a great challenge to accurately simulate the hypersonic high-enthalpy flows in which the gas flow is chemically reacting, the boundary layer may be separated and transfers into turbulent. In particular, most of the thermochemical reaction models have its different limitations on modelling the hypervelocity flows where its total temperatures range from 1500 to 10000 K, the boundary layer is affected significantly by the flow temperature and its transition cannot be predicted with acceptable accuracy. Therefore, the flow modeling and high-order CFD solvers are significantly important for the future nozzle design.
(3) One aspect worth exploring further is to take high-accuracy experimental data of high-enthalpy flows as the basis for nozzle optimization design or boundary-condition setting. Two steps are involved in the exploring. The first step is to correct the nozzle contour at the throat based on the experimental data of the designed nozzle. The process is simple and straightforward, but not cost-efficient. The partial "re-design and re-correction" is necessary for the machined nozzle. The other one is to take the high-enthalpy flow field data as the design reference condition, and make correction and optimization with the CFD technique. In accordance with the flow parameters, such as the flow enthalpy and Mach number, this way can take the corresponding test measurements as the reference conditions for optimizing the nozzle design.
(4) The CFD/OM method is considered to be of the complex design process and high-computational cost, but it probably represents the one of the development trends in nozzle design because of the rapid development of the computer technology in near future. The high-accuracy CFD simulations with thermochemical non-equilibrium reaction models are also in urgent need in view of the hypersonic technology. This huge driving force will require that chemically-reacting models used

in the CFD solver must be further improved. Such the progress may bring a wide application to the CFD/OM method.

From a point view of methodology, the hypersonic nozzle design initiates from theoretical analysis with the MOC, and was improved with semi-empirical formula approximations, enhanced with the boundary-layer correction and fulfilled with the CFD-based optimization methods (CFD/OM and CFD/MOC/OM). The MOC is a foundation and the CFD introduction is a big milestone, and the two achievements are huge. These theories and methods have developed into a theoretic system so far, and occupy a big portion of the wind tunnel theories. In order to support the development of advanced hypersonic vehicles, the research on the theory and method for designing the high-enthalpy flow nozzle is still important and should receive great attention in future.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

This research was co-supported by the National Natural Science Foundation of China (Nos. 11672357 and 11727901).

References

1. Stalker RJ. Hypervelocity aerodynamics with chemical nonequilibrium. *Ann Rev Fluid Mech* 1989;**21**:37–60.
2. Stalker RJ. A study of the free-piston shock tunnel. *AIAA J* 1967;**5**(12):2160–5.
3. Bird GA. A note on combustion driven shock tubes. Paris: AGARD; 1957. Report No.: 146.
4. Yu HR, Esser B, Lenartz M, et al. Gaseous detonation driver for a shock tunnel. *Shock Waves* 1992;**2**(4):245–54.
5. Jiang ZL, Zhang ZJ, Liu YF, et al. Criteria for hypersonic airbreathing propulsion and its experimental verification. *Chinese Journal of Aeronautics* 2021;**34**(3):94–104.
6. Prandtl L, Busemann A. Näherungsverfahren zur zeichnerischen ermittlung von ebenen strömungen mit übershllgeschwindigkeit. Festschrift zum 70. Geburtstag von A. *Stodola. Zurich: Füssli* 1929;499–509.
7. Busemann A. *Gasdynamik handbuch der experimentalphysik*, Vol. 4.1. Leipzig: Akademische Verlagsgesellschaft; 1931. p. 341–460.
8. Preiswerk E. Application of the methods of gas dynamics to water flows with free surface I: Flows with no energy dissipation. Washington, D.C.: NACA; 1940, Report No.: NACA TM 934.
9. Puckett AE. Supersonic nozzle design. *J Appl Mech* 1946;**13**(4):A265–70.
10. Foelsch K. A new method of designing two-dimensional Laval nozzles for a parallel and uniform jet. Inglewood: North American Aviation, Inc.; 1947. Report. No.: NA-46-235.
11. Foelsch K. The analytical design of an axially symmetric Laval nozzle for a parallel and uniform jet. *Journal of the Aeronautical Sciences* 1949;**16**(3): 161-66, 188.
12. Cresci RJ. Tabulation of coordinates for hypersonic axisymmetric nozzles. Part I—Analysis and coordinates for test section Mach numbers of 8, 12, and 20. 1958. Report No.: WADD-TN-58-300.
13. Sivells JC. Aerodynamic design of axisymmetric hypersonic wind tunnel nozzles. Reston: AIAA; 1969. Report No.: AAIA-1969-0337.
14. Sivells JC. Aerodynamic design of axisymmetric hypersonic wind-tunnel nozzles. *J Spacecraft Rockets* 1970;**7**(11):1292–9.
15. Sivells JC. A computer program for the aerodynamic design of axisymmetric and planar nozzles for supersonic and hypersonic wind tunnels. Arnold AFB: Arnold Engineering Development Center; 1978. Report No.: AEDC-TR-78-63.
16. Guentert EC, Neumann HE. Design of axisymmetric exhaust nozzles by method of characteristics incorporating a variable isentropic exponent. Washington, D.C.: NASA; 1959. Report No.: TR R-33.
17. Enkenhus KR, Maher EF. The aerodynamic design of axisymmetric nozzles for high-temperature air. 1962. Report No.: 7395.
18. Benton J, Edwards A, Perkins J. Limitations of the method of characteristics when applied to axisymmetric hypersonic nozzle design. Reston: AIAA; 1990. Report No.: AAIA-1990-0192.
19. Korte JJ. Aerodynamic design of axisymmetric hypersonic wind-tunnel nozzles using least-squares/parabolized Navier-Stokes procedure. *J Spacecraft Rockets* 1992;**29**(5):685–91.
20. Korte JJ, Hedlund E, Anandakrishnan S. A comparison of experimental data with CFD for the NSWC hypervelocity wind tunnel Mach14 nozzle. Reston: AIAA; 1992. Report No.: AIAA-1992-4010.
21. Korte JJ, Kumar A, Singh DJ, et al. Least-squares parabolized Navier-Stokes procedure for optimizing hypersonic wind-tunnel nozzles. *J Propul Power* 1992;**8**(5):1057–63.
22. Korte JJ, Kumar A, Singh D, et al. CAN-DO, CFD-based Aerodynamic Nozzle Design and Optimization program for supersonic/hypersonic wind tunnels. Reston: AIAA; 1992. Report No.: AAIA-1992-4009.
23. Korte JJ. A CFD-based aerodynamic design procedure of hypersonic wind-tunnel nozzles. *AGARD conference proceedings*; 1993. p.36.1-10.
24. Korte JJ, Hodge JS. Prediction/measurement of flow quality in hypersonic wind-tunnel nozzles designed using CFD. Reston: AIAA; 1994. Report No.: AIAA-1994-2544.
25. Korte JJ, Hodge JS. Flow quality of hypersonic wind-tunnel nozzles designed using computational fluid-dynamics. *J Spacecraft Rockets* 1995;**32**(4):569–80.
26. Korte JJ. Inviscid design of hypersonic wind tunnel nozzles for a real gas. Reston: AIAA; 2000. Report No.: AIAA-2000-0677.
27. Keeling SL. A strategy for the optimal design of nozzle contours. Reston: AIAA; 1993. Report No.: AIAA-1993-2720.
28. Tolle R. A new optimum design code for hypersonic nozzles, utilizing response surface methodology. Reston: AIAA; 1997. Report No.: AIAA-1997-0519.
29. Shope FL. Contour design techniques for super/hypersonic wind tunnel nozzles. Reston: AIAA; 2006. Report No.: AIAA-2006-3665.
30. Chan WYK, Jacobs PA, Smart MK, et al. Aerodynamic design of nozzles with uniform outflow for hypervelocity ground-test facilities. *J Propul Power* 2018;**34**(6):1467–78.
31. Gaffney RLJ. Design of a pulse-facility nozzle using the rotational method of characteristics. *J Spacecraft Rockets* 2006;**43**(6):1216–24.
32. Tang B, Wang YP, Jiang ZL. Nozzle design for a large[HYPHEN]scale high[HYPHEN]enthalpy shock tunnel. *Sci Sin-Phys Mech Astron* 2019;**49** [in Chinese] 074701.
33. Tong BG, Kong XY, Deng GH. *Aerodynamics*. Beijing: Higher Education Press; 1990 [Chinese].
34. Anderson Jr JD. *Hypersonic and high-temperature gas dynamic*. 2nd ed. Reston: AIAA; 2006.
35. Hall JG, Treanor CE. Nonequilibrium effects in supersonic-nozzle flows. 1968. Report No.: AGARDograph 124.
36. Anderson Jr JD. A true-dependent analysis for vibrational and chemical nonequilibrium nozzle flows. *AIAA J* 1970;**8**(3):545–50.

37. Anderson Jr JD. Time-dependent solutions of nonequilibrium nozzle flows—A sequel. *AIAA J* 1970;**8**(12):2280–2.
38. Anderson Jr JD. Modern compressible flow: With historical perspective. 3rd ed. New York: McGraw-Hill Book Company; 2003.
39. Erickson WD. Vibrational nonequilibrium flow of nitrogen in hypersonic nozzles. Washington, D.C.: NASA; 1963. Report No.: TN D-1810.
40. Harris EL, Albacete LM. Vibrational relaxation of nitrogen in the NOL hypersonic tunnel No. 4. White Oak: Naval Ordnance Laboratory; 1964. Report No.: TR 63-221.
41. Zucrow MJ, Hoffman JD. *Gas dynamics, Volume 2. Multidimensional flow*. Hoboken: John Wiley & Sons, Inc.; 1997.
42. Crown JC. Supersonic nozzle design. Washington, D.C.: NACA; 1948. Report No.: TN 1651.
43. Hall IM. Transonic flow in two-dimensional and axially-symmetric nozzles. *Quarterly J Mech Appl Math XV* 1962;**4**:487–508.
44. Potter JL, Carden WH. Design of axisymmetric contoured nozzles for laminar hypersonic flow. *J Spacecraft Rockets* 1968;**5**(9):1095–100.
45. Sivells JC. Aerodynamic design and calibration of the VKF 50-inch hypersonic wind tunnels. Arnold AFB: Arnold Engineering Development Center; 1963. Report No.: TDR-62-230.
46. Ali MH, Mashud M, Bari AA, et al. Numerical solution for the design of minimum length supersonic nozzle. *ARPN J Eng Appl Sci* 2012;**7**(5):605–12.
47. Sivells JC, Payne RG. A method of calculating turbulent boundary growth at hypersonic Mach numbers. Arnold AFB: Arnold Engineering Development Center; 1959. Report No.: TR-59-3.
48. Benton J. Design and Navier-Stokes analysis of hypersonic wind tunnel nozzles [dissertation]. Raleigh: North Carolina State University; 1989.
49. Candler G, Perkins J. Effects of vibrational nonequilibrium on axisymmetric hypersonic nozzle design. Reston: AIAA; 1991. Report No.: AIAA-1991-0297.
50. Craddock CS. Design of the axisymmetric HyShot nozzle for T4. Brisbane: University of Queensland; 2000. Reptort No.: 2/00.
51. Huddleston DH. Aerodynamic design optimization using computational fluid dynamics [dissertation]. Knoxville: University of Tennessee; 1989.
52. Korte JJ. An explicit upwind algorithm for solving the parabolized Navier-Stokes equations. Washington, D.C.: NASA; 1991. Report No.: TP 3050.
53. Gaffney RLJ, Korte JJ. Analysis and design of rectangular-cross-section nozzles for scramjet engine testing. Reston: AIAA; 2004. Report No.: AIAA-2004-1137.
54. Gollan RJ, Jacobs PA. About the formulation, verification and validation of the hypersonic flow solver Eilmer. *Int J Numer Meth Fluids* 2013;**73**(1):19–57.
55. Nelder JA, Mead R. A simplex method for function minimization. *Comput J* 1965;**7**(4):308–13.
56. Hannemann K, Itoh K, Mee DJ, et al. Free piston shock tunnels HEG, HIEST, T4 and T5. In: Igra O, Seiler F, editors. *Experimental Methods of Shock Wave Research*. New York: Springer; 2016. p. 181–264.
57. Armstrong AH, Smith MG. Two-dimensional supersonic nozzle design, Part I: Theory. 1951. Report No. 5/51.
58. Ruptash J. Supersonic wind tunnels—Theory design and performance. Toronto: University of Toronto, Institute of Aerophysics; 1952.
59. Beckwith IE, Moore JA. An accurate and rapid method for the design of supersonic nozzles. Washington, D.C.: NACA; 1955. Report No.: TN 3322.
60. Johnson CB, Boney LR, Ellison JC, et al. Real-gas effects on hypersonic nozzle contours with a method of calculation. Washington, D.C.: NASA; 1963, Report No.: TN D-1622.
61. Lukasiewicz J. *Experimental methods of hypersonics, gasdynamics*. New York: M. Dekker; 1973. p. 80–6.
62. Shapiro AH. *The dynamics and thermodynamics of compressible fluid flow*, ***Vol. 1***. New York: The Ronald Press Company; 1953.
63. Johnson CB, Boney LR. A method for calculating a real-gas two-dimensional nozzle contour including the effects of gamma. Washington, D.C.: NASA; 1975. Report No.: TM X-3243.
64. Erickson WD, Creekmore HS. A study of equilibrium real-gas effects in hypersonic air nozzles, including charts of thermodynamic properties for equilibrium air. Washington, D.C.: NASA; 1960, Report No.: TN D-231.
65. Eggers Jr AJ. One-dimensional flows of an imperfect diatomic gas. Washington, D.C.: NACA; 1949. Report No.: TN 1861.
66. Zonars D. Nonequilibrium regime of airflows in contoured nozzles: Theory and experiments. *AIAA Journal* 1967;**5**(1):57–63.
67. Hollis BR. Real gas flow parameters for NASA Langley 22-inch Mach 20 helium tunnel. Washington, D.C.: NASA; 1992. Report No.: CR-4462.
68. Shope FL. Design optimization of hypersonic test facility nozzle contours using splined corrections. Arnold AFB: Arnold Engineering Development Center; 2004. Report No.: TR-2005-2.
69. Shope FL, Tatum KE. Reacting Navier-Stokes nozzle contour design optimization. Reston: AIAA; 2004. Report No.: AIAA-2005-5081.
70. Scales LS. *Introduction to non-linear optimization*. Berlin: Springer-Verlag; 1985.
71. Conte SD, Boor C. *Elementary numerical analysis*. 2nd ed. New York: McGraw-Hill Book Company; 1972.
72. Wright MJ, Candler GV. Data-parallel line relaxation method for the Navier-Stokes equations. *AIAA J* 1998;**36**(9):1603–9.
73. Candler G. *APTU nozzle code manual, Version 3.0*. 2004.
74. Candler G. Hypersonic nozzle analysis using an excluded volume equation of state. Reston: AIAA; 2005. Report No.: AIAA-2005-5202.
75. Gordon S, McBride BJ. Computer program for calculation of complex chemical equilibrium compositions and applications, II. User's manual and program description. Washington, D.C.: NASA; 1996.
76. Gaffney RLJ. Erratum on design of a pulse-facility nozzle using the rotational method of characteristics. *Journal of Spacecraft and Rockets* 2007;**44**(2):496–504.
77. McBride BJ, Gordon S, Reno MA. Coefficients for calculating thermodynamic and transport properties of individual species. Washington, D.C.: NASA; 1993. Report No.: TM 4513.
78. McBride BJ, Zehe MJ, Gordon S. NASA Glenn coefficients for calculating thermodynamic properties of individual species. Washington, D.C.: NASA; 2002. Report No.: NASA/TP-2002-211556.
79. Witten IH, Frank E, Hall MA, et al. *Data mining: Practical machine learning tools and techniques*. 4th ed. Amsterdam: Elsevier; 2017.
80. Jeong S, Shimoyama K. Review of data mining for multi-disciplinary design optimization. *Proc Inst Mech Eng, Part G: J Aerospace Eng* 2011;**225**(5):469–79.
81. Huang W. Design exploration of three-dimensional transverse jet in a supersonic crossflow based on data mining and multi-objective design optimization approaches. *Int J Hydrogen Energy* 2014;**39**(8):3914–25.
82. Huang W, Wang ZG, Ingham DB, et al. Design exploration for a single expansion ramp nozzle (SERN) using data mining. *Acta Astronaut* 2013;**83**:10–7.
83. Huang W, Pourkashanian M, Ma L, et al. Effect of geometric parameters on the drag of the cavity flameholder based on the variance analysis method. *Aerosp Sci Technol* 2012;**21**:24–30.
84. Ou M, Yan L, Huang W, et al. Design exploration of combinational spike and opposing jet concept in hypersonic flows based on CFD calculation and surrogate model. *Acta Astronaut* 2019;**155**:287–301.

85. Ahlberg JH, Hamilton S, Migdal D, et al. Truncated perfect nozzles in optimum nozzle design. *ARS J* 1961;**31**(5):614–20.
86. Hoffman JD. Design of compressed truncated perfect nozzles. *J Propul Power* 1987;**3**(2):150–6.
87. Chue RSM, Cresci D, Montgomery P. Design and analysis of a rectangular cross-section hypersonic nozzle. *The 26th international symposium on shock waves*, 2007.
88. Takahashi M, Ueda S, Tomita T, et al. Transient flow simulation of a compressed truncated perfect nozzle. Reston: AIAA; 2001. Report No.: AIAA-2001-3681.
89. Verma S. Study of flow separation in truncated ideal contour nozzle. *J Propul Power* 2002;**18**(5):1112–21.
90. Verma S, Haidn O. Flow separation studies in a subscale truncated ideal contour nozzle. Reston: AIAA; 2007. Report No.: AIAA-2007-1320.
91. Stark R, Wagner B. Experimental flow investigation of a truncated ideal contour nozzle. Reston: AIAA; 2006. Report No.: AIAA-2006-5208.
92. Ladeinde T, Chen H. Performance comparison of a full-length and a truncated aerospike nozzle. Reston: AIAA; 2010. Report No.: AIAA-2010-6593.
93. Ruf J, McDaniels D, Brown A. Details of side load test data and analysis for a truncated ideal contour nozzle and a parabolic contour nozzle. Reston: AIAA; 2004. Report No.: AIAA- 2010-6813.