

Article

Research on the Zooming Method for Determining the Flow, Heat Transfer, and Infrared Radiation of an Air-Breathing Hypersonic Vehicle Powered by a Scramjet

Xin Wei , Xiaojuan Shi *, Honghu Ji and Jinlong Hu

College of Energy & Power Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China; weixin_nuaa@163.com (X.W.); jhhpe@nuaa.edu.cn (H.J.); hjlnuaa@126.com (J.H.)

* Correspondence: shixiaojuan@nuaa.edu.cn

Abstract: In order to study the infrared radiation characteristics of an air-breathing hypersonic vehicle powered by a scramjet, it is necessary to solve the internal and external flow field of the air-breathing hypersonic vehicle. Owing to the complexity and difficulty of solving the three-dimensional flow and heat-transfer process in a scramjet combustor, a quasi-one-dimensional calculation method was established. Utilizing zooming technology, a combination of quasi-one-dimensional simulation within the combustion chamber and three-dimensional numerical simulation elsewhere on the vehicle was employed to obtain the flow field. The accuracy of the zooming method in determining flow, heat transfer, and infrared radiation was verified through comparison with experimental data. The results show that under the flight condition of $Ma = 6$, the gas temperature and wall heat flux in the scramjet combustor first increased and then decreased along the flow direction. The Mach number of the plume was smaller than that of the free flow, while the velocity of the plume was slightly larger. In the wavelength range of 3–5 μm , as the azimuth angle increased, the integrated radiation intensity of the air-breathing hypersonic vehicle demonstrated a characteristic pear-shaped distribution.

Keywords: hypersonic vehicle; zooming technology; scramjet; heat transfer; quasi-one-dimensional simulation; infrared radiation



Citation: Wei, X.; Shi, X.; Ji, H.; Hu, J. Research on the Zooming Method for Determining the Flow, Heat Transfer, and Infrared Radiation of an Air-Breathing Hypersonic Vehicle Powered by a Scramjet. *Aerospace* **2024**, *11*, 212. <https://doi.org/10.3390/aerospace11030212>

Academic Editor: Sebastian Karl

Received: 9 January 2024

Revised: 27 February 2024

Accepted: 6 March 2024

Published: 8 March 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

A hypersonic vehicle powered by a scramjet engine has obvious advantages over a traditional hypersonic vehicle [1]. Notably, the fuel-specific impulse of a scramjet is much higher than that of traditional rocket engines [2]; at the same time, scramjet engines use oxygen in the atmosphere as an oxidant, and the weight of the aircraft can be reduced when completing the same flight mission. At a flight Mach number range of 3–6, the scramjet engine stands out as the preferred propulsion system for air-breathing hypersonic vehicles [3].

Scramjet-powered hypersonic vehicles have a strong infrared radiation signature. Investigating the calculation methodology for the infrared radiation signature of air-breathing hypersonic vehicles offers a theoretical foundation for detecting and identifying hypersonic-vehicle targets. The main infrared radiation sources of hypersonic vehicles powered by scramjet engines comprise the high-temperature cavity wall, plume, and skin. As such, it is necessary to explore the calculation methods for flow and heat transfer in air-breathing scramjet-powered hypersonic vehicles.

At present, the primary focus of research on the infrared radiation signature of hypersonic vehicles has been on non-aspirated hypersonic vehicles such as space shuttles and boost-glide hypersonic vehicles. Wang [4] took the head of an aircraft with a blunt spherical shape as the object and numerically simulated the aerodynamic heating and aerodynamic heating radiation of the aircraft flying at a high Mach number. A series of remote infrared observations of the space shuttle Orbiter was carried out by NASA (National Aeronautics

and Space Administration) [5,6]. Kennedy [7] et al. compared and analyzed the flow-field parameters of a general rocket exhaust plume at a 40 km altitude. Mu [8] took the small blunt cone of NASA's RAM-CII flight test as the research object, and obtained the optical radiation characteristic data for the flow field around the blunt cone in the 2–5 μm band by numerical calculation. Levin et al. [9] predicted the infrared shock-layer radiance from a slender hypersonic vehicle. It was stated that radiation from shock-heated ambient CO_2 is an important contribution at most altitudes and at speeds slower than 3.5 km/s. Niu [10–12] conducted a more in-depth study on the infrared radiation characteristics of the booster-glide hypersonic vehicle HTV (hypersonic technology vehicle)-2, and obtained the point-source radiation signature of the HTV-2 aircraft based on the LOS (line-of-sight) method to solve the radiation-transfer equation. Yang [13] simulated the effects of aerodynamic heating, surface temperature field, and infrared radiation characteristics of HTV-2 under typical flight conditions on the flight trajectory. The abovementioned research studies have reference significance for the infrared radiation characteristics of the air-breathing hypersonic vehicle skin. However, due to the high interconnectivity of the internal and external flow fields of the air-breathing hypersonic vehicle, the integration of the aircraft and the scramjet engine, and the complex physical and chemical processes of the internal flow of the scramjet engine, research on the infrared radiation characteristics of the air-breathing hypersonic vehicle is still relatively lacking. At present, it is still a challenge to calculate the infrared radiation signature of the air-breathing hypersonic vehicle.

The combustion chamber of a scramjet typically utilizes regenerative cooling. Resolving the flow and heat-transfer processes within this chamber entails addressing various physical and chemical phenomena, including radiation heat transfer, convective heat transfer, combustion, and heat conduction [14–17]. Three-dimensional numerical simulation of the combustion chamber is difficult and complicated. At present, there are some quasi-one-dimensional and one-dimensional calculation methods for the rapid calculation of the scramjet combustion chamber parameters [18–22]. These calculation methods can be summarized into two categories. One is to first calculate the Mach-number distribution of the gas flow and then to calculate other variables, ignoring friction, heat exchange, and fuel mass [18]; the other is to directly solve the quasi-one-dimensional Euler equation [19–22], which takes into account factors such as area change, mass addition, and wall friction. Nonetheless, these methods do not incorporate convective heat transfer on the coolant side, and the coupling of convective heat transfer between the gas, combustion chamber wall, and cooling channel is not addressed in the solution. In the present study, a convective heat-transfer model for the cooling channel, based on a straight rectangular fin, was established. Subsequently, the quasi-one-dimensional Euler equation was solved, incorporating source terms such as area change, friction, and heat release from chemical reactions. This method allows for a more accurate and efficient representation of the flow and heat-transfer processes in the scramjet combustion chamber.

In order to more fully and rapidly evaluate a potential component design in the context of an engine system [23], zooming technology was first proposed in the NPSS (numerical propulsion system simulation) program [24]. Zooming technology, also known as variable complexity analysis, is a technology that combines the calculation of a low-order accuracy model (zero-dimensional or one-dimensional) of the entire engine flow part with the calculation of a high-order accuracy model (three-dimensional) in a local component. It allows for customization of the resolution and fidelity of engine models to meet analytical needs. This reduces the demand for computing resources, as high-fidelity analysis is only applied to components of interest [23]. In this paper, for the calculation of infrared radiation characteristics of the air-breathing hypersonic vehicle, the flow and heat transfer of the combustion chamber were found to have a secondary effect on the infrared radiation characteristics of the aircraft. Consequently, there was no stringent demand for high accuracy in calculating the flow and heat transfer within the combustion chamber. Further, conducting three-dimensional numerical simulations of the combustion chamber is challenging and resource-intensive. Hence, employing a lower-order accuracy model,

such as a one-dimensional approach, for calculating the flow and heat transfer within the combustion chamber was deemed viable. Meanwhile, other components could be modeled and calculated in three dimensions.

The internal and external flow fields of the air-breathing hypersonic vehicle powered by a scramjet were obtained using zooming technology. This approach integrated a quasi-one-dimensional simulation method within the scramjet's combustion chamber with three-dimensional numerical simulations conducted outside the combustion chamber. In parallel, the accuracy of the calculation method for flow, heat transfer, and infrared radiation was verified through comparison with experimental data. Based on this method, the flow, heat transfer of the air-breathing hypersonic vehicle, and infrared radiation signature at different azimuth angles were thoroughly investigated.

2. Air-Breathing Hypersonic Vehicle Model

Figure 1a is a schematic of an air-breathing scramjet-powered hypersonic-vehicle model. The hypersonic vehicle features a waverider configuration, comprising components such as the forebody/inlet, isolator, combustor chamber, nozzle, and other associated elements. The area ratio of the nozzle outlet to the inlet is 3.9, and the aspect ratio of the nozzle outlet is 2.2. The combustion chamber of the scramjet adopts regenerative cooling technology, and the coolant inside the cooling channel is kerosene, as shown in Figure 1d. The combustion-chamber channel takes the form of a divergent channel, with a solid wall covering that includes rectangular cooling channels. The structure of the cooling channel is shown in Figure 1e. The material of the combustion chamber is Inconel X-750, the wall thickness of the hot side of the cooling channel is a fixed value of 8 mm, and the width w and spacing s of the cooling channel are design parameters.

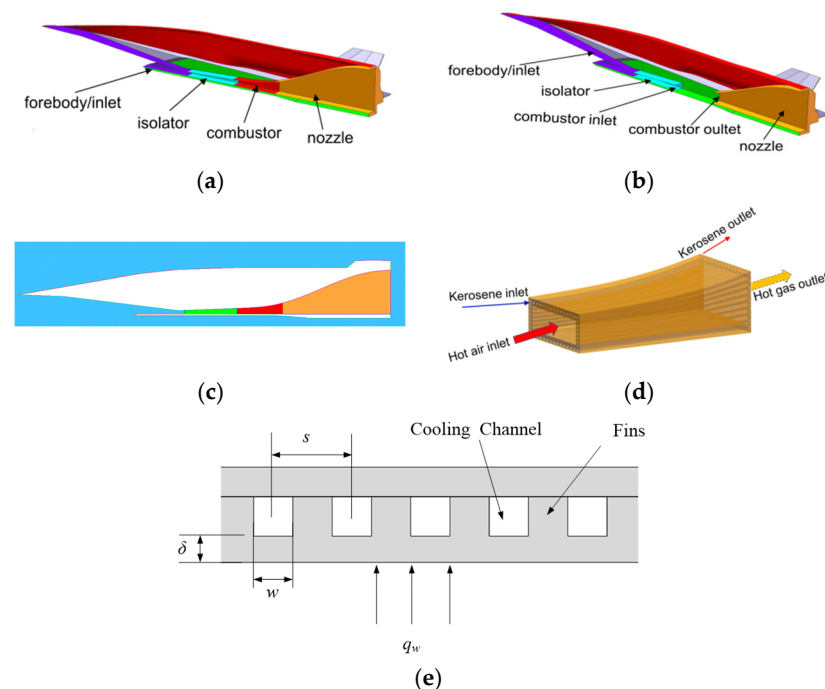


Figure 1. Physical model and computational model of an air-breathing hypersonic vehicle powered by a scramjet. (a) Schematic of the air-breathing hypersonic vehicle model powered by a scramjet (1/2 model); (b) Three-dimensional computational model of a hypersonic vehicle without a combustion chamber (1/2 model); (c) Two-dimensional computational model of an air-breathing hypersonic vehicle; (d) Quasi-one-dimensional computational model of the combustion chamber; (e) Schematic of the cooling channel.

3. Zooming Strategy for Calculating the Infrared Radiation of an Air-Breathing Hypersonic Vehicle

The internal and external flow fields of an air-breathing hypersonic vehicle powered by a scramjet are obtained by the zooming technology, which combines the quasi-one-dimensional simulation method in the combustion chamber of the scramjet with the three-dimensional numerical simulation outside the combustion chamber. On this basis, the infrared radiation characteristics of the air-breathing hypersonic vehicle are calculated. The process of the zooming strategy for the infrared radiation calculation for an air-breathing hypersonic vehicle is as follows:

- (1) Divide the entire calculation domain into the three-dimensional computational model of the hypersonic vehicle without a combustion chamber (as shown in Figure 1b) and the quasi-one-dimensional computational model of the combustion chamber (as shown in Figure 1d).
- (2) Carry out the two-dimensional numerical simulation of the flow field on the symmetry plane of the hypersonic vehicle (Figure 1c) to obtain the pressure, temperature, velocity, and other parameters of the air at the outlet of the isolator and the static pressure at the outlet of the combustion chamber, and use these as the inlet boundary condition and the outlet boundary condition of the quasi-one-dimensional calculation for the combustion chamber, respectively.
- (3) Perform the quasi-one-dimensional calculation for the combustion chamber (as shown in Figure 1d) and extract the total temperature, total pressure, and static pressure of the air outlet for the combustion chamber as the boundary conditions of the nozzle inlet for the three-dimensional numerical simulation.
- (4) Perform the three-dimensional numerical simulation on the internal and external flow fields for the hypersonic vehicle without a combustion chamber (as shown in Figure 1b). Combine the quasi-one-dimensional calculation results to obtain the internal and external flow fields of the entire calculation domain for the air-breathing hypersonic vehicle.
- (5) Perform the infrared radiation simulation for the air-breathing hypersonic vehicle at different azimuth angles.

4. Quasi-One-Dimensional Calculation Method for the Flow and Heat Transfer in a Combustion Chamber with Regenerative Cooling

4.1. Governing Equations

The combustion chamber of scramjet can be simplified to a one-dimensional flow channel with a cross-sectional area that changes along the flow direction. Therefore, a quasi-one-dimensional Euler equation containing source terms such as area change, friction, mass addition, and chemical-reaction heat release is used to describe the flow and heat transfer in the combustion chamber:

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} = S \quad (1)$$

where $U = (\rho, \rho u, \rho E)^T$, $F = (\rho u, \rho u^2 + p, \rho u H)^T$, and the source term S at the right of the equation is:

$$S = -\frac{1}{A(x)} \cdot \frac{dA}{dx} \begin{bmatrix} \rho u \\ \rho u^2 \\ \rho u H \end{bmatrix} + \frac{1}{A(x)} \begin{bmatrix} dm/dx \\ 0 \\ dQ/dx \end{bmatrix} + \begin{bmatrix} 0 \\ -0.5\rho u^2 f/D(x) \\ 0 \end{bmatrix} - \frac{4}{D(x)} \begin{bmatrix} 0 \\ 0 \\ q_w \end{bmatrix} \quad (2)$$

where A is the cross-sectional area, D is the equivalent diameter, f is the friction coefficient, dQ/dx is the heat added per unit length, dm/dx is the mass added per unit length, and q_w

is the heat-flux density of the combustion chamber wall. H is the total specific enthalpy, and its expression is as follows:

$$H = \int_{T_0}^{T_t} c_p dT \quad (3)$$

The mass and energy conservation equations of kerosene in the cooling channel are shown as follows:

$$\rho_f u_f A = m_f \quad (4)$$

$$m_f \frac{d(h_f + \frac{1}{2}u_f^2)}{dx} = q_{wc} P_{wc} \quad (5)$$

where ρ_f is the kerosene density, u_f is the kerosene velocity, m_f is the flow rate of kerosene, P_{wc} is the circumference of a single cooling channel, q_{wc} is the average heat flux of the cooling channel, h_f is the enthalpy of kerosene, and A is the cross-sectional area of the cooling channel.

4.2. Parameter Models for Solving Quasi-One-Dimensional Governing Equations

(1) The heat-added term dQ/dx and mass-added term dm/dx .

The Rayleigh distribution is used to describe the law of heat release along the flow direction in the mainstream. It is similar to the heating law of the actual scramjet engine [25]. It has been applied in the one-dimensional simulation of many combustion-chamber flow fields [26,27]. The formula of the heating law is as follows:

$$dQ/dx = -\frac{x - x_0}{\mu^2} e^{-\frac{(x-x_0)^2}{2\mu^2}} Q \quad (6)$$

where Q represents the total heat release, x_0 is the starting position of the heating zone, and μ is a characteristic parameter of the Rayleigh distribution.

For the fuel mass-added term dm/dx , it is considered that it has the same form as dQ/dx :

$$\frac{dQ}{dx} = \frac{dm}{dx} \eta \Delta q \quad (7)$$

where Δq is the reaction-heat per unit mass, and η is the combustion efficiency.

(2) Friction coefficient.

The friction coefficient f adopts the empirical formula fitted by Wang [20], as shown in the following formula:

$$f = \frac{0.0592}{(Re_x^*)^{0.2}} \times 0.85 \quad (8)$$

(3) The aerodynamic-heating heat flux q_{aero} .

When the gas-flow velocity is extremely large, the physical parameters of the gas flow in the boundary layer change greatly and cannot be regarded as constants. At this time, the enthalpy difference is used to define the heat-transfer coefficient of the gas side [28]:

$$q_{aero} = \alpha_h (h_r - h_w) \quad (9)$$

$$h_r = h_\infty + \text{Pr}^{1/3} \frac{u_\infty^2}{2} \quad (10)$$

$$h^* = h_\infty + 0.5(h_w - h_\infty) + 0.22(h_r - h_\infty) \quad (11)$$

where α_h is the heat-transfer coefficient based on the reference enthalpy, h_r is the recovery enthalpy, h_w is the enthalpy at the wall temperature, h_∞ is the enthalpy of incoming gas flow, and u_∞ is the incoming flow velocity.

According to the definition of the Stanton number [28] and the Chilton–Colburn analogy [29], α_h is obtained according to the following formula:

$$St = \frac{\alpha_h}{\rho u_\infty} = \frac{1}{2} f Pr^{-2/3} \quad (12)$$

(4) The radiation heat flux.

Since the radiation between the high-temperature gas and the combustion chamber wall cannot be ignored, in addition to the aerodynamic heat, the radiation heating of the gas to the wall should also be considered when calculating the wall heat transfer. The radiative heat flux is calculated using the zero-dimensional model proposed by Lefebvre [30]:

$$q_{rad} = \left(\frac{1 + \varepsilon_w}{2} \right) \sigma (\varepsilon_g T_g^4 - \alpha_g T_w^4) \quad (13)$$

(5) A convective heat-transfer model of the cooling channel based on a straight rectangular fin.

In this paper, the following assumptions are made to simplify the convective heat-transfer model for the internal cooling channel of the combustion chamber wall: The heat conduction of the solid wall along the cooling channel is neglected, and only the heat conduction perpendicular to the wall is considered. The outer wall of the combustion chamber is an adiabatic wall. Therefore, the solid wall between the adjacent cooling channels in Figure 1e can be assumed to be a straight rectangular fin, and the fin efficiency of a single straight rectangular fin is as follows:

$$\eta_f = \frac{th(mw')}{mw'} \quad (14)$$

where $w' = w + (s - w)/2$ (the definition of w and s are shown in Figure 1e), th is the hyperbolic function, and the expression of m is as follows:

$$m = \sqrt{\frac{2h_c}{k(s - w)}} \quad (15)$$

where h_c is the convective heat-transfer coefficient of the kerosene side and k is the thermal conductivity of the solid wall.

The convective heat-transfer coefficient h_c of kerosene is obtained by the kerosene correlation of the internal forced convection [31] and the definition of the Nusselt number, as shown below:

$$Nu_f = 0.0065 Re_f^{0.89} Pr^{0.4} \left(\frac{\mu_f}{\mu_w} \right)^{0.1} \quad (16)$$

$$Nu_f = \frac{h_c w}{\lambda_f} \quad (17)$$

where μ_w is the dynamic viscosity of kerosene at the average wall temperature, μ_f is the dynamic viscosity of kerosene at the average fluid bulk temperature, and λ_f is the thermal conductivity of kerosene.

According to the conservation of energy, the local heat-transfer coefficient h_c of the cooling channel and the heat flux q_w of the combustion-chamber wall satisfy the following formula:

$$q_w L(x) = N h_c (T_f - T_{wc}) (w + 2w\eta_f) \quad (18)$$

where $L(x)$ is the perimeter of the inner surface of the combustion chamber at the axial location x , N is the number of cooling channels, T_f is the temperature of kerosene, and T_{wc} is the wall temperature of the cooling channel.

According to Fourier's law of heat conduction, the inner wall temperature in the combustion chamber T_w and the wall temperature T_{wc} of the cooling channel satisfy the following relationship:

$$T_w = q_w \delta / k + T_{wc} \quad (19)$$

4.3. Quasi-One-Dimensional Calculation of Boundary Conditions

The boundary conditions of the quasi-one-dimensional calculation of the combustion chamber can be determined by means of a simplified calculation of the two-dimensional cold-state simulation with the symmetry plane. Figure 1c presents a two-dimensional calculation model of the air-breathing hypersonic vehicle. The air is compressed by the inlet, enters the combustion chamber through the isolator, and is then accelerated by the nozzle. Table 1 shows the boundary conditions of the two-dimensional cold-state calculation, with reference to Ref [32]. Figure 2a shows a schematic of the computational domain of the two-dimensional cold-state calculation. The grid of the two-dimensional model is shown in Figure 2b, and the number of cells of the used grids was 54,000. A structured grid was used, and the thickness of the first boundary layer was 0.01 mm, the number of boundary layers was 15, and the growth ratio was 1.1. Figure 2c presents the static-pressure contour of the two-dimensional cold-state simulation. An observation can be made that there was an obvious shock-wave system when the air flowed through the intake and the isolator. The shock wave was continuously attenuated after several reflections, and finally disappeared near the inlet of the combustion chamber. Table 2 shows the boundary conditions of the quasi-one-dimensional calculation of the combustion chamber obtained by means of the two-dimensional cold-state numerical calculation.

Table 1. Boundary conditions of the two-dimensional cold-state calculation.

Ma	Height (m)	Pressure (Pa)	Temperature (K)
6	24,858	2607	221

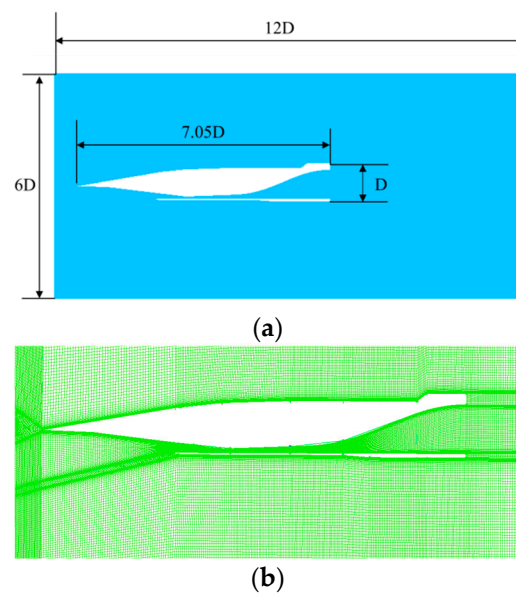


Figure 2. Cont.

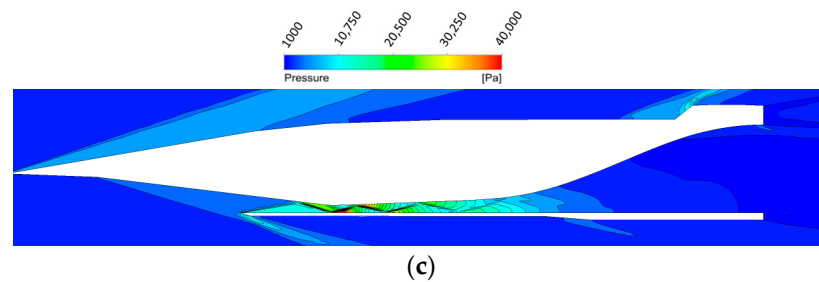


Figure 2. Computational domain, grid, and static-pressure contour of the two-dimensional cold-state calculation. (a) Schematic of the computational domain; (b) Grid; (c) Static-pressure contour of the symmetry plane.

Table 2. Boundary conditions of the quasi-one-dimensional calculation of the combustion chamber.

Inlet Mach Number	Inlet Static Temperature (K)	Inlet Static Pressure (Pa)	Outlet Static Pressure (Pa)
3.35	593	19,689	25,435

4.4. Validation of the Quasi-One-Dimensional Calculation Method in the Scramjet Combustor

To assess the performance of a scramjet within the Mach 7–8 flight regime, the University of Queensland in Australia coordinated two Hyshot flight tests [33]. In the present study, two sets of flight test data were employed to validate the accuracy of the quasi-one-dimensional method in the combustion chamber. Geometric parameters of the combustion chamber and nozzle are shown in Figure 3a. The static-pressure distribution of the combustion chamber obtained through the quasi-one-dimensional calculation method was compared with experimental results from two flight conditions, as shown in Figure 3b. The first condition was the cold-state condition, with the equivalence ratio $\Phi = 0$. The Mach number at the inlet of the combustion chamber was 2, the static temperature was 1571 K, and the static pressure was 65.9 kPa. The equivalence ratio Φ of the second condition was 0.3, the Mach number at the inlet of the combustion chamber was 2, the static temperature was 1528 K, and the static pressure was 73.76 kPa. An observation can be made from Figure 3b,c that the static-pressure distribution of the combustion chamber obtained by the quasi-one-dimensional calculation method was close to the experimental data under the two working conditions. Thus, the accuracy of the quasi-one-dimensional calculation method was verified.

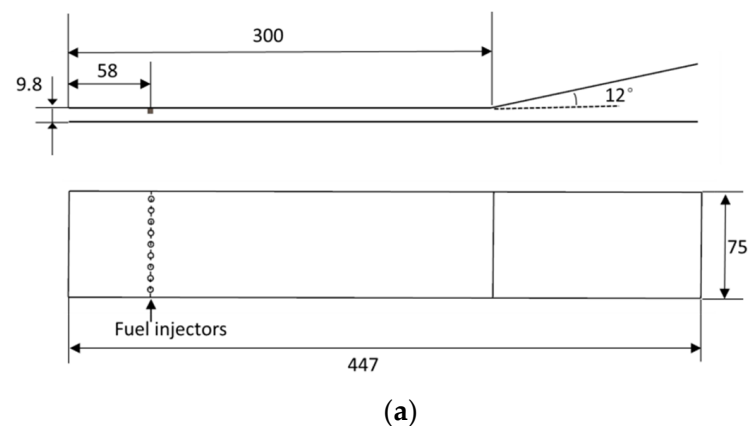


Figure 3. Cont.

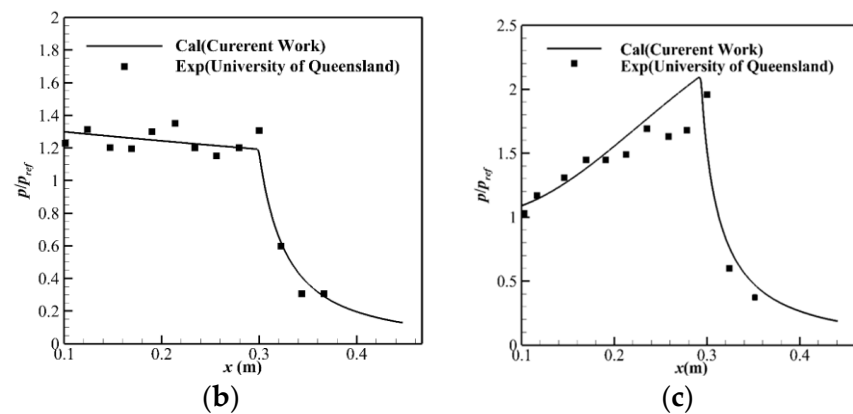


Figure 3. Validation of the quasi-one-dimensional calculation method in the scramjet combustor. (a) Geometric parameters of the combustion chamber and nozzle in the Hyshot-II flight test (mm); (b) $\Phi = 0$; (c) $\Phi = 0.3$.

5. Methodology for Calculating the Three-Dimensional Flow Field and Heat Transfer in an Air-Breathing Hypersonic Vehicle without a Combustor

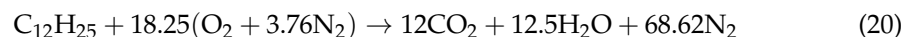
5.1. Boundary Conditions

The boundary conditions for the external flow field utilized a pressure far-field approach, as outlined in Table 1. The outlet of the isolator employed a pressure-outlet boundary condition, with the static pressure derived from the results of the two-dimensional cold-state calculation. The inlet parameters of the nozzle corresponded to the outlet parameters of the combustion chamber obtained from the quasi-one-dimensional calculation. The symmetry plane of the aircraft was designated as a symmetric boundary condition. Detailed internal-flow boundary conditions of the aircraft are provided in Table 3.

Table 3. Boundary conditions of the internal flow field of the aircraft.

Flight Ma	Static Pressure of Isolator Outlet (Pa)	Total Temperature of Nozzle Inlet (K)	Static Pressure of Nozzle Inlet (Pa)	Mach Number of Nozzle Inlet
6	19,689	2818	33,502	1.92

The combustion process in the combustion chamber is simplified by assuming an equivalence ratio of 1 and a combustion efficiency of 50% [34]. A single-step chemical reaction is used to calculate the species mass fractions at the nozzle inlet. The single-step chemical-reaction equation is as follows:



5.2. Grid Division

The ICEM software is used to divide the grid, and the grid is a structured hexahedral grid. The whole aircraft grid is shown in Figure 4a, the forebody/inlet grid of the aircraft is shown in Figure 4b, the afterbody/nozzle grid is shown in Figure 4c, and the number of cells of the used grids is 6.5 million. Moreover, the fluid grid is refined near the wall, thickness of the first boundary layer grid is 0.01 mm, the growth ratio of the boundary layer grid is 1.1, and the number of grid layers is 15. The fluid grid on the symmetry plane is shown in Figure 4d.

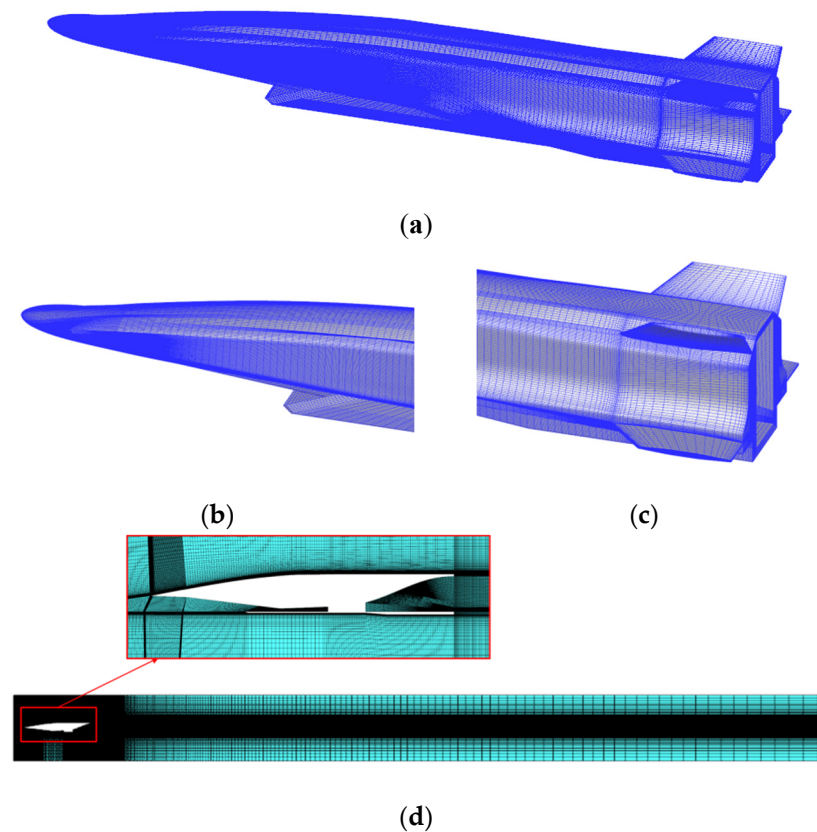


Figure 4. Grid division of the air-breathing hypersonic vehicle. (a) Grid in the solid domain; (b) Afterbody/nozzle grid; (c) Head/inlet grid; (d) The fluid grid on the symmetry plane.

5.3. Validation of the CFD Method for Aerodynamic Heating

The accuracy of the numerical calculation method of the external flow field was verified through an aerodynamic heating experiment of the leading edge of the tube made by Allan R. Wieting [35]. Figure 5a shows a schematic of the aerodynamic heating experiment of the leading edge of the tube. The material of the tube was steel, and the flow parameters were $T = 241.5$ K, $Ma = 6.47$, and $P = 648$ Pa. The inner diameter of the tube was 25.4 mm, the outer diameter was 38.1 mm, and the initial temperature was 294.4 K. The experiment was conducted in an 8-foot high-temperature wind tunnel, with the incoming flow consisting of high-temperature methane gas. The computational domain and boundary conditions for the numerical simulation are illustrated in Figure 5b. Figure 5c presents a comparison of the accuracy of the S-A model and the SST $k-\omega$ turbulence model with the experiment data. The heat flux was normalized to the heat flux at the stagnation point obtained from the experiment. Evidently, the SST $k-\omega$ turbulence model aligns well with the experimental data.

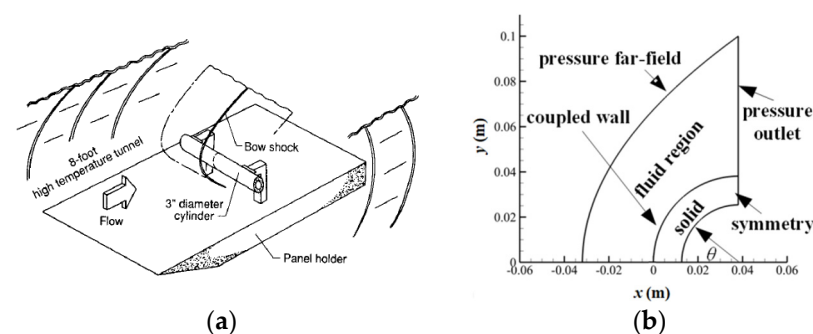


Figure 5. Cont.

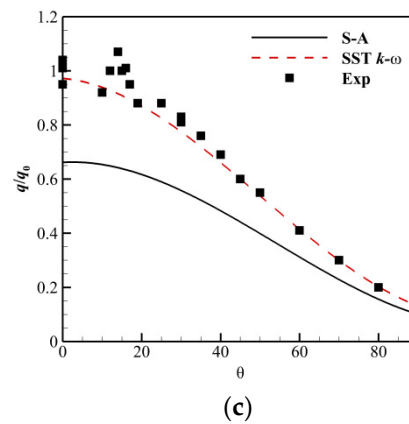


Figure 5. Validation of the CFD method for an external flow field. (a) The schematic of the aerodynamic heating experiment of the leading edge of the tube; (b) Computational domain and boundary conditions of numerical simulation; (c) The variation in the wall heat flux of the circular tube with θ .

6. Numerical Calculation Method for Infrared Radiation

6.1. The Basic Equations for Infrared Radiation Calculation

The basic equations for infrared radiation calculation mainly include the radiation-transfer equation, the radiation-irradiance equation, and the radiation-intensity equation.

(1) Radiation-transfer equation expressed using the radiation radiance.

The spectral radiance-transfer equation characterizes the relationship between the variation in radiation energy during transmission in a participating medium and the absorption, emission, and scattering of the participating medium. Without considering the scattering medium, the spectral radiance-transfer equation is:

$$\frac{dL_{\lambda}(s)}{ds} = -\kappa_{\lambda}L_{\lambda}(s) + \kappa_{\lambda}L_{b\lambda}(s) \quad (21)$$

where κ_{λ} is the spectral absorption coefficient, and $L_{b\lambda}$ is the spectral radiance of blackbody obtained by Planck's law. $\kappa_{\lambda}L_{\lambda}(s)$ represents the attenuation of spectral energy caused by the absorption of the medium, and the second term signifies the increase in spectral energy due to the emission of the medium.

(2) Radiation-irradiance equation on the detector surface.

Radiation irradiance is defined as the radiation flux per unit area of the detector; the unit is W/m^2 , and the spectral irradiance is the ratio of the irradiance in the micro-band to the micro-wavelength range. According to reference [36], the irradiance H_{λ} formed by the target radiation energy on the surface of the detector is as follows:

$$H_{\lambda} = \sum_{k=1}^N L_b(\lambda, T_k) \cos \theta_k \frac{\Omega_{FOV}}{N} \quad (22)$$

(3) Radiation-intensity equation of the target.

The radiation intensity is defined as the radiation flux emitted by the radiation source within the unit solid angle in a given transmission direction, and the spectral radiation intensity is the ratio of the radiation intensity in the micro-band to the micro-wavelength range. When calculating the spectral radiation intensity of the target, the target is considered as a point source. The formula for calculating the spectral radiation intensity is as follows, with R being the distance between the target and the detector.

$$I_{\lambda} = H_{\lambda}R^2 \quad (23)$$

6.2. Solution Method

The reverse Monte Carlo method [37] was used to solve the radiative-transfer equation. In this approach, rays were emitted from the surface of the detector, and a probability model was employed to depict the radiation emission, absorption, and reflection on the surface of the micro-element, as well as the radiation emission, absorption, and scattering within the micro-element. The statistical analysis of random variables was used to solve the radiation-transfer equation. The Monte Carlo method has the advantages of strong adaptability, ease of handling of complex problems, and high calculation accuracy.

6.3. Validation of the IR Calculation Method

A relevant test on the axisymmetrically scaled exhaust model was carried out in Ref. [38], which measured the spectral radiation intensity of the axisymmetric exhaust at different azimuth angles in the band of 3–5 μm . The experimental results of the integrated radiation intensity of the exhaust and the calculation results obtained by the infrared radiation calculation method were compared, as shown in Figure 6. The maximum error between the experimental results and the calculated results of the integrated radiation intensity was 13.6%. This proved that the calculation method of infrared radiation is reliable.

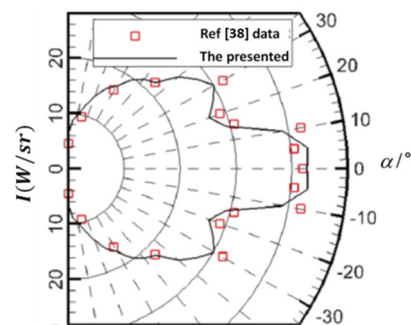


Figure 6. Comparison between the experimental results and the calculated results of the integrated radiation intensity.

7. Results and Discussion

7.1. Flow-Field Distribution in the Combustion Chamber Based on the Quasi-One-Dimensional Calculation Method

The flow field of the combustion chamber based on the quasi-one-dimensional calculation method is shown in Figure 7. An observation can be made from Figure 7a that the Mach number of the gas along the flow direction was greater than 1, and, thus, had no effect on the flow of the inlet. The Mach number of the gas exhibited a rapid decrease along the flow direction followed by a gradual increase, whereas the static-pressure distribution of the gas displayed an opposite trend to the variation in the Mach number. In the middle of the combustion chamber ($x = 0.25$ m), the static pressure of the gas was the highest, and the Mach number was the smallest. Such findings could be attributed to the law of combustion heat release along the flow direction in the combustion chamber. In the initial stage, combustion heat release played a significant role, with higher heat input leading to increased static pressure and a reduced Mach number. Additionally, the expansion of the combustion chamber channel in the second half was a significant factor, leading to a decrease in pressure. Figure 7b shows that the temperature of the coolant gradually increased along the flow direction. The wall temperature of the combustion chamber increased rapidly in the first half and remained basically unchanged in the second half. This was because the heat release of the fuel in the second half of the combustion chamber was not obvious, and the heating effect on the wall was weak. Figure 7c shows that the gas temperature and wall heat flux of the combustion chamber increased first and then decreased. At the midpoint of the combustion chamber ($x = 0.25$ m), the gas experienced the highest wall heat flux. This observation can likely be attributed to the gradual expansion of the

combustion chamber area in the initial half, as illustrated in Figure 1d, where combustion heat release predominantly influenced the process. In the second half of the combustion, the heat-release effect of the fuel was weakened, the flow area of the combustion chamber increased rapidly, and the expansion of the combustion chamber channel was a significant factor, leading to a decrease in the gas temperature.

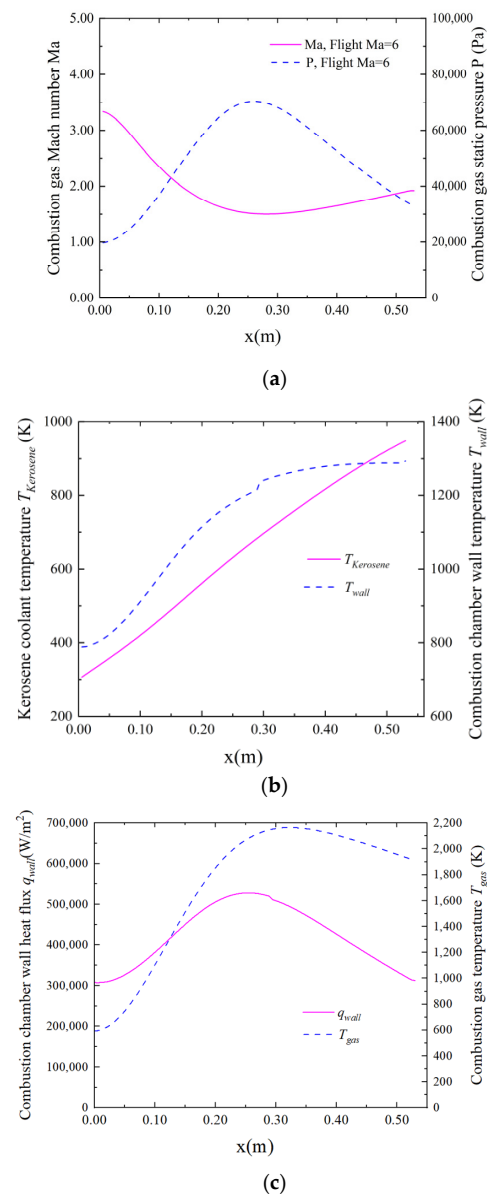


Figure 7. The flow field of the combustion chamber along the flow direction. (a) Gas static pressure and gas Mach number variations along the flow direction x ; (b) Coolant temperature and wall temperature variation along the flow direction x ; (c) Combustion-chamber gas temperature and wall heat flux along the flow direction x .

7.2. Three-Dimensional Flow-Field Distribution in the Air-Breathing Hypersonic Vehicle

The Mach number and velocity distribution on the symmetry plane are shown in Figure 8a. An observation can be made that the Mach number of the plume was smaller than the Mach number of the free flow. Such findings arose from the elevated temperature of the plume and the consequent increase in local sound speed. As such, while the plume Mach number was lower than that of the free flow, the plume velocity was marginally higher than the free-flow velocity. Because there was a certain distance between the nozzle and the fuselage, a recirculation zone was formed behind the fuselage. The low-speed gas

in the recirculation zone gradually mixed with the high-speed gas along the flow direction, which reduced the velocity of the plume. Therefore, an observation can be made from Figure 8a that the velocity of certain gas particles within the shear layer of both the free flow and the plume was lower than that of the surrounding free flow. With the increase in x , the velocity of the plume was gradually the same as that of the free flow.

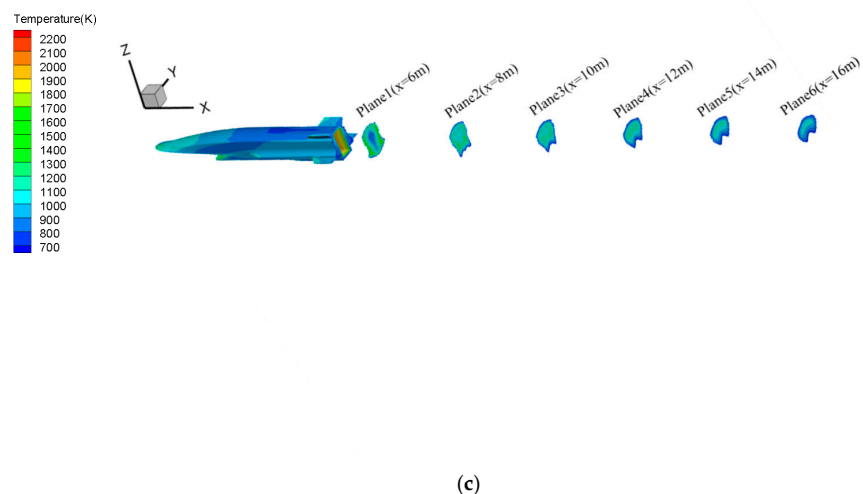
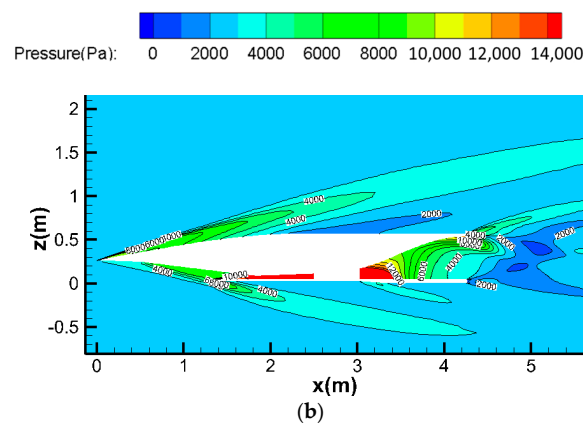
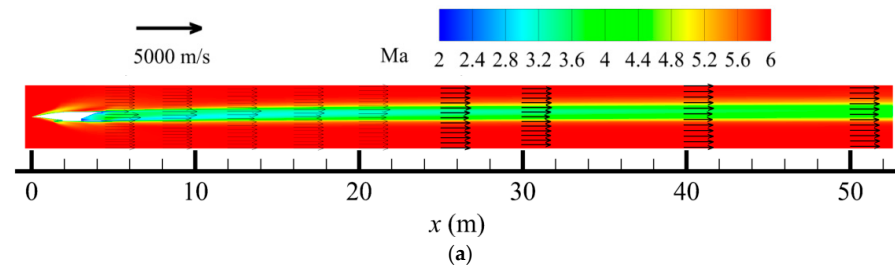


Figure 8. Cont.

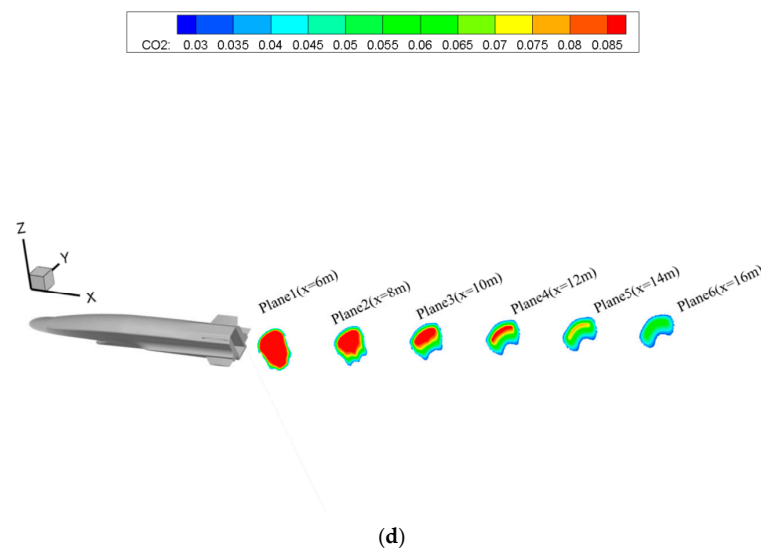


Figure 8. Three-dimensional flow-field distribution of the air-breathing hypersonic vehicle. (a) The Mach number and velocity distribution on the symmetry plane; (b) The static pressure distribution on the symmetry plane; (c) The temperature distribution of the aircraft wall and the plume; (d) The mole-fraction contours of CO_2 distribution for the aircraft.

Figure 8b shows the static-pressure distribution on the symmetry plane. Evidently, there was an obvious oblique shock wave at the forebody. Positioned beneath the forebody, the oblique shock wave was situated near the leading edge of the inlet. This waverider configuration leveraged the shock wave to augment the static pressure at the inlet. Upon airflow entry into the inlet, the static pressure of the air underwent further augmentation. Inside the nozzle, due to the expansion of the channel, the supersonic airflow gradually expanded, and the static pressure of the airflow gradually decreased. Near the nozzle outlet, due to the influence of the tail and the rear of the fuselage of the aircraft, a recirculation zone was formed, which was a low-pressure zone.

Figure 8c shows the temperature distribution for the aircraft wall and the plume. When the flight Mach number was 6, the average temperature of the skin wall was 850 K, and the average wall temperature of the nozzle was 2102 K. The leading edge of the forebody, the inlet, and the tail were high-temperature regions, because the aerodynamic heating was most obvious there. With the development of the plume, the gas temperature gradually decreased, and the shape of the high-temperature zone gradually changed from a rectangle to a circle. In a section not far from the nozzle outlet ($x = 6$ m), there was a low-temperature zone in the center of the plume, which could potentially be ascribed to the formation of a recirculation zone at the rear of the tail and the fuselage of the aircraft and the low-temperature air being sucked into the interior of the high-temperature gas.

Figure 8d shows the mole-fraction contours of CO_2 distribution for the aircraft. An observation can be made that there were species distributions of CO_2 characterized by a core region that interacted with the plume. As the plume evolved, the core region of CO_2 gradually contracted and shifted upward. This phenomenon could be attributed to the nozzle's shape, which deflected the jet direction upward. Moreover, the distribution of CO_2 closely resembles the distribution of the gas temperature.

Table 4 shows the drag of each component and the net thrust of the aircraft along the axial axis. The drag of each component was composed of pressure drag and viscous drag. As shown in Table 4, the total thrust slightly exceeded the total drag, suggesting that the aircraft can generate positive thrust during the flight condition at Mach 6 and maintain a nearly constant speed. This observation was further supported by Figure 8a, where it is evident that the velocity of the plume was slightly higher than that of the free flow. In addition, the drag from skin constituted 82.6% of the total drag. At the same time, it was observed that the pressure drag exerted by the nozzle and the combustion chamber was

negative, owing to the divergent channel configuration, which contributed positively to the thrust.

Table 4. The drag of each component and the net thrust of the aircraft.

Component	Pressure Drag (N)	Viscous Drag (N)	Net Drag (N)
skin	1030	565	1595
fin	276	59	335
isolator	−114	138	24
forebody	154	84	238
combustor	−123	59	−64
inlet	133	114	247
nozzle	−670	225	−445
total drag		1930	
total trust		2003	

7.3. The Integrated Radiation Intensity of the Air-Breathing Hypersonic Vehicle

The layout of detecting points is shown in Figure 9a, which is divided into a horizontal detection plane, a top detection plane, and a lower detection plane, with intervals of 15° between each detecting point. The azimuth angle in the detection plane is represented by α .

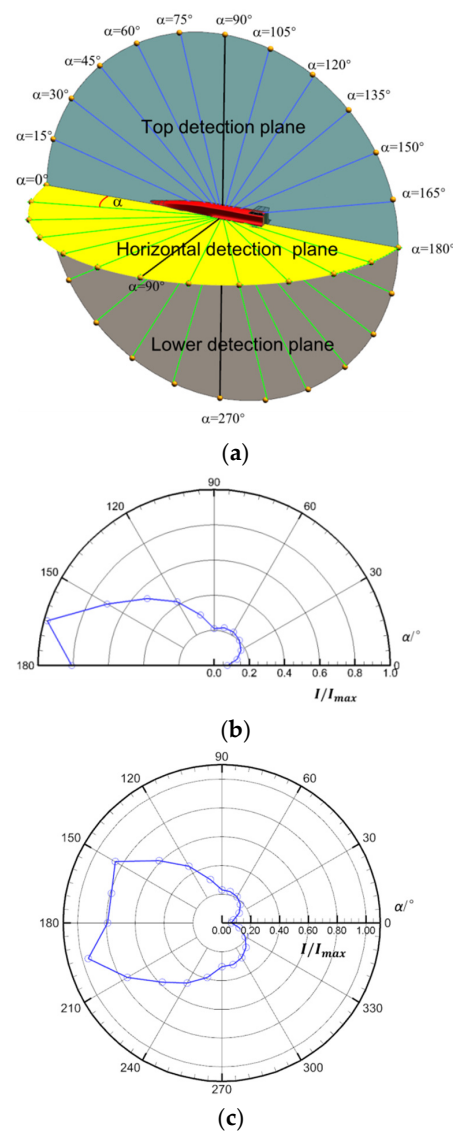


Figure 9. Cont.

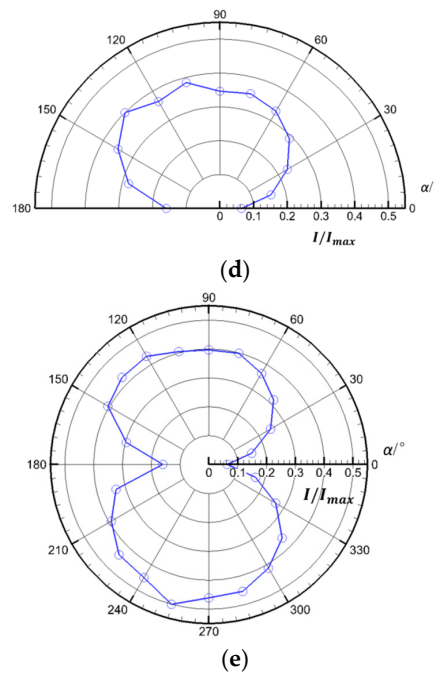


Figure 9. Integrated radiation-intensity distribution of the vehicle at different azimuth α values. (a) Distribution of detection points; (b) Horizontal detection plane (1–3 μm); (c) Top and lower detection plane (1–3 μm); (d) Horizontal detection plane (3–5 μm); (e) Top and lower detection plane (3–5 μm).

Figure 9b,c present the variations in the integrated radiation intensity with the azimuth angle in the 1–3 μm band. The integrated radiation intensity of the vehicle was normalized by dividing it by the maximum integrated radiation intensity in the 1–3 μm band, rendering the data dimensionless. For the horizontal plane, with the increase in the azimuth angle α , the integrated radiation intensity increased slowly and then increased rapidly. When the azimuth angle was equal to 165°, the integrated radiation intensity reached its maximum value, then gradually decreased with the increase in the azimuth angle α . Because the temperature of the skin was lower than the temperature of the nozzle's solid wall, the main contributor to the integrated radiation intensity in the 1–3 μm band was the nozzle wall. When the azimuth angle α was greater than 90°, with the decrease in α , the projection area of the visible nozzle first increased and then decreased. Therefore, the integrated radiation intensity first increased and then decreased. When the azimuth angle α was less than 90°, the nozzle wall could not be seen, and therefore the integrated radiation intensity gradually decreased with the decrease in the visible skin projection area. For the top detection plane, the trend of the integrated radiation intensity with the azimuth angle α resembled that of the horizontal plane. However, the key difference lies in the azimuth angle at which the maximum radiation intensity occurred, which was 150°. This discrepancy could be linked to the variation in the visible nozzle's projection area with the azimuth angle. Regarding the lower detection plane, the variation in the integrated radiation intensity with the azimuth angle mirrored that of the horizontal plane. Additionally, at identical azimuth angles, the integrated radiation intensity of the lower detection plane surpassed that of both the horizontal and top detection planes. Such findings could be attributed to the fact that the visible nozzle projection area of the lower detection plane was larger, while the temperature of skin was relatively higher.

Figure 9d,e show the variations in the integrated radiation intensity with the azimuth angle in the 3–5 μm band. For the horizontal plane, with the increase in the azimuth angle, the integrated radiation intensity first increased and then remained basically unchanged, and then it gradually decreased, showing the characteristics of a pear-shaped distribution. According to Wien's displacement law, as the temperature increases, the wavelength

corresponding to the maximum radiation intensity becomes smaller. The wavelength corresponding to the peak radiation intensity of the skin at a lower temperature was larger than that of the nozzle at a higher temperature, and the main contribution band of the integrated radiation intensity of the skin was 3–5 μm . Hence, the integrated radiation intensity in the 3–5 μm band adhered to the aforementioned law owing to variations in the skin projection area. Further, when the azimuth angle exceeded 90° , the integrated radiation intensity in the 3–5 μm band at identical azimuth angles was lower than that in the 1–3 μm band. This is because the high temperature wall of the nozzle was visible when the azimuth angle was greater than 90° . The peak of the spectral radiation intensity was located in the 1–3 μm band, and the main contribution of the radiation intensity was the high temperature wall of the nozzle. For the top and lower detection planes, the variation in the integrated radiation intensity with the azimuth angle was similar to that of the horizontal plane. Moreover, at identical azimuth angles, the integrated radiation intensity is greatest at the lower detection plane, followed by the horizontal plane, with the top detection plane exhibiting the smallest intensity.

8. Conclusions

In the present study, a quasi-one-dimensional calculation method including source terms such as area change, friction, and chemical-reaction heat release was established for solving the flow and heat-transfer processes in the combustion chamber. Through the application of zooming technology, the internal and external flow-field characteristics were meticulously investigated, as were the infrared radiation properties of the air-breathing hypersonic vehicle at various azimuth angles. This methodology integrated quasi-one-dimensional simulation within the combustion chamber with three-dimensional numerical simulation outside the combustion chamber. The main conclusions are as follows:

- (1) Under the flight condition of $\text{Ma} = 6$, the gas temperature and wall heat flux of the scramjet combustion chamber first increased and then decreased along the flow direction. In the middle of the combustion chamber ($x = 0.25 \text{ m}$), the wall heat flux of the gas was the largest. The Mach number of the gas decreased rapidly along the flow direction and then increased slowly, while the static-pressure distribution of the gas was opposite to the variation trend of the Mach number.
- (2) The Mach number of the plume of the scramjet-powered hypersonic vehicle was smaller than that of the free flow, which could be ascribed to the higher temperature of the plume. However, the velocity of the plume was marginally higher than that of the free flow. This observation suggests that the scramjet engine is capable of producing positive thrust under the flight condition of Mach 6. Under the flight condition of $\text{Ma} = 6$, the average wall temperature of the skin was 850 K, and the average wall temperature of the nozzle was 2102 K.
- (3) Under the flight condition of $\text{Ma} = 6$, with the increase in the azimuth angle, the integrated radiation intensity of the scramjet-powered hypersonic vehicle increased slowly at first before increasing rapidly and then decreasing gradually in the 1–3 μm band. In the 3–5 μm band, with the increase in azimuth angle, the integrated radiation intensity showed the characteristics of a pear-shaped distribution.

Author Contributions: Conceptualization, X.W., H.J. and X.S.; methodology, X.W., X.S. and J.H.; validation, X.W., H.J. and J.H.; formal analysis, X.S.; investigation, X.W.; data curation, X.W.; writing—original draft preparation, X.W. and X.S.; writing—review and editing, X.S. and X.W. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Science and Technology Major Project (J2019-III-0009-0053).

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

References

- Blankson, I.M. Air-Breathing Hypersonic Cruise: Prospects for Mach 4–7 Waverider Aircraft. *J. Eng. Gas Turbines Power* **1994**, *116*, 104–115. [\[CrossRef\]](#)
- Tang, C.S. Research on integrated optimization design of hypersonic cruise vehicle. *Aerosp. Sci. Technol.* **2008**, *12*, 567–572.
- Jiang, Z.L.; Zhang, Z.J.; Liu, Y.F. Criteria for hypersonic airbreathing propulsion and its experimental verification. *Chin. J. Aeronaut.* **2021**, *34*, 94–104. [\[CrossRef\]](#)
- Wang, Y.H.; Wang, Q. Experiment of the thermo-radiation characteristic of infrared window of hypersonic vehicles. *Infrared Laser Eng.* **2015**, *44*, 1716–1720.
- Horvath, T.J.; Cagle, M.F. Remote observations of reentering spacecraft including the space shuttle orbiter. In Proceedings of the 2013 IEEE Aerospace Conference, Big Sky, MT, USA, 2–9 March 2013; IEEE: Piscataway, NJ, USA, 2013; pp. 1–15.
- Schwartz, R.; Horvath, T.J.; Propp, T. Scientifically Calibrated In-Flight Imagery (SCIFLI) Mission Operations for the SpaceX CRS-4 Airborne Infrared Imaging. In Proceedings of the AIAA SPACE Astronautics Forum and Exposition, Orlando, FL, USA, 12–14 September 2017; p. 5209.
- Kennedy, K.; Walker, B. Tactical missile exhaust plume at angle of attack. In Proceedings of the 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Fort Lauderdale, FL, USA, 11–14 July 2004.
- Mu, L.; Ma, Y.; He, Z.H.; Dong, S.K. Radiation characteristics simulation of non-equilibrium flow field around the hypersonic blunted cone. *J. Eng. Thermophys.* **2012**, *33*, 1958–1962.
- Levin, D.A.; Candler, G.V.; Limbaugh, C.C. Multispectral shock-layer radiance from a hypersonic slender body. *J. Thermophys. Heat Transf.* **2000**, *14*, 237–243. [\[CrossRef\]](#)
- Niu, Q.L.; Yuan, Z.C.; Chen, B.; Dong, S.K. Infrared radiation characteristics of a hypersonic vehicle under time-varying angles of attack. *Chin. J. Aeronaut.* **2019**, *32*, 861–874. [\[CrossRef\]](#)
- Niu, Q.L.; Yang, S.; He, Z.H.; Dong, S.K. Numerical study of infrared radiation characteristics of a boost-gliding aircraft with reaction control systems. *Infrared Phys. Technol.* **2018**, *92*, 417–428. [\[CrossRef\]](#)
- Niu, Q.L.; Yang, X.; Chen, B.; He, Z.H.; Dong, S.K. Infrared radiation characteristics and detectability analysis of point source based on high-speed sliding. *Infrared Laser Eng.* **2018**, *47*, 160–167.
- Yang, X.; Niu, Q.L.; He, Z.H.; Dong, S.K. Analysis of infrared radiation characteristics and detectability of HTV-2 like hypersonic gliding aircrafts. *Acta Opt. Sin.* **2017**, *37*, 45–54.
- Shen, W.B.; Huang, Y. Mixing and transient combustion processes of scramjet combustor with transverse injector and hypermixer. *Case Stud. Therm. Eng.* **2021**, *26*, 101104. [\[CrossRef\]](#)
- Gugulothu, S.K.; Nutakki, P.K. Dynamic fluid flow characteristics in the hydrogen-fuelled scramjet combustor with transverse fuel injection. *Case Stud. Therm. Eng.* **2019**, *14*, 100448. [\[CrossRef\]](#)
- Huang, S.Z.; Chen, Q. Numerical evaluation of shock wave effects on turbulent mixing layers in a scramjet combustor. *Case Stud. Therm. Eng.* **2021**, *25*, 100893. [\[CrossRef\]](#)
- Kummitha, O.R.; Pandey, K.M.; Gupta, R. Cfd analysis of a scramjet combustor with cavity based flame holders. *Acta Astronaut.* **2018**, *144*, 244–253. [\[CrossRef\]](#)
- O'Brien, T.F.; Lewis, M.J.; Starkey, R.P. Quasi-one-dimensional high-speed engine model with finite-rate chemistry. *J. Propuls. Power* **2001**, *17*, 1366–1374. [\[CrossRef\]](#)
- Zhang, Y.F.; Yang, Q.C.; Xu, X. One dimensional method for calculation of dual mode scramjet combustor core flow area. *J. Propuls. Technol.* **2018**, *8*, 1803–1809.
- Wang, L.; Xing, J.W.; Zheng, Z.H. One dimensional evaluation of the scramjet flow path performance. *J. Propuls. Technol.* **2008**, *29*, 641–645.
- Doolan, C.J.; Boyce, R.R. A Quasi-one-dimensional mixing and combustion code for trajectory optimisation and design studies. In Proceedings of the 15th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, Dayton, OH, USA, 28 April–1 May 2008.
- Jiang, J.; Chu, M.; Xu, X. A Quasi-one-dimensional method for prediction of dual mode scramjet combustor performance. *J. Propuls. Technol.* **2013**, *7*, 803–808.
- Follen, G.; auBuchon, M. *Numerical Zooming between a NPSS Engine System Simulation and a One-Dimensional High Compressor Analysis Code*; NASA/TM-2000-209913; NASA: Washington, DC, USA, 2000.
- Lytle, J.K. *The Numerical Propulsion System Simulation: An Overview*; NASA TM-2000-209915; NASA: Washington, DC, USA, 2000.
- Micka, D.; Torrez, S.; Driscoll, J. Heat release distribution in a dual-mode scramjet combustor -measurements and modeling. In Proceedings of the 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference, Bremen, Germany, 19–22 October 2009; pp. 1–19.
- Yuan, H.C.; Liang, D.W.; Guo, R.W. Predicting method for the performance of constant area isolator under back pressure. *J. Aerosp. Power* **2009**, *11*, 2421–2428.
- Zheng, L.M.; Sun, B. 2D thermal environment numerical simulation of scramjets. *J. Aerosp. Power* **2007**, *22*, 823–828.
- Rohsenow, W.M.; Hartnett, J.P.; Cho, Y.I. *Handbook of Heat Transfer*; Osborne McGraw-Hill: Sebastopol, CA, USA, 1973.
- Chilton, T.H.; Colburn, A.P. Mass transfer (absorption) coefficients: Prediction from data on heat transfer and fluid friction. *Ind. Eng. Chem.* **1934**, *26*, 1183–1187. [\[CrossRef\]](#)

30. Lefebvre, A.H.; Herbert, M.V. Heat transfer processes in gas-turbine combustion chambers. *Proc. Inst. Mech. Eng.* **1961**, *175*, 641–668. [[CrossRef](#)]
31. Zhang, F.; Fan, X.; Yu, G. Heat transfer of aviation kerosene supercritical conditions. *J. Thermophys. Heat Transf.* **2009**, *23*, 543–550. [[CrossRef](#)]
32. Nie, Y.; Le, G.G.; Ma, D.W. Numerical investigation on properties of angle of attack (AOA) for the X-51A Vehicle. In Proceedings of the 4th International Conference on Mechanical Engineering and Mechanics, Suzhou, China, 10–12 August 2011; pp. 540–544.
33. Smart, M.K.; Hass, N.E.; Paull, A. Flight data analysis of the HyShot 2 scramjet flight experiment. *AIAA J.* **2006**, *44*, 2366–2375. [[CrossRef](#)]
34. Wang, X.D.; Song, W.Y. Effects of fuel injection scheme on the mixing and combustion efficiency of a scramjet combustor. *Chin. J. Aeronaut.* **2004**, *25*, 556–559.
35. Allan, R.W. Experiment study of shock wave interference heating on cylindrical leading edge at Mach 6 and 8. In Proceedings of the AIAA 22nd Thermophysics Conference, Honolulu, HI, USA, 8–10 June 1987.
36. Yong, S.; Zhang, J.Z. Numerical investigation of flow mixture enhancement and infrared radiation shield by lobed forced mixer. *Appl. Therm. Eng.* **2009**, *29*, 3687–3695.
37. Collins, D.G.; Blattner, W.G.; Wells, M.B. Backward Monte Carlo calculations of the polarization characteristics of the radiation emerging from spherical-shell atmospheres. *Applied Opt.* **1972**, *11*, 2684–2696. [[CrossRef](#)]
38. Liu, J.; Ji, H.H. Investigation on infrared signature of axisymmetric vectoring exhaust system with infrared suppressions. *J. Thermophys. Heat Transf.* **2018**, *32*, 627–636. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.