Numerical Prediction and Experimental Verification of Flow Field in Hypersonic Wind Tunnel

Xue-wen SUN¹,³*, Hai-bo YANG¹,²,³ and Tao MI¹,⁴

¹School of Mechanical Engineering, University of Science and Technology Beijing, Beijing, China
²Centre of Excellence for Advanced Materials, Dongguan, China
³Key Laboratory of Fluid Interaction with Material, Ministry of Education, Beijing, China
⁴School of Materials Engineering, North China Institute of Aerospace Engineering, Langfang, China

*Corresponding author

Keywords: Hypersonic, Wind tunnel, Flow field, Numerical simulation.

Abstract. The knowledge of the flow field inside the wind tunnel is an essential element for the identification of test experiment. For the sake of obtaining more insight of the flow propagation throughout a newly built wind tunnel, numerical simulation of the flow field is performed, and the Navier–Stokes equations are solved using the computational fluid dynamics (CFD) code Fluent. Experimental surveys of the wind tunnel flow with pressure sensors and thermocouple sensors are also performed, providing valuable data for the flow characterization. The Mach number, temperature and pressure in the core area of the wind tunnel are obtained, and the behavior of flow propagation is quantitatively described. The simulation results of the CFD model have a good agreement with the experimental data. Through this study, the flow field characteristics of the wind tunnel are obtained, which can provide some reference for future experiments.

Introduction

Sustained flight at hypersonic imposes very high thermal loads on external structure of the vehicle, and the high temperature is a challenge for the thermal protection materials [1,2]. Thermal protection system (TPS) materials are required to protect structural components of space vehicles during the re-entry stage [3-5]. Wind tunnel provides the aerothermodynamic heating environment to test the performance of various types of TPS materials [6,7]. A detailed characterization of wind tunnel flow field is required for both appropriate design of the experimental test and interpretation of results later. However, the flow field in the test section of wind tunnels is very difficult to assess and remains one of the challenging tasks [8]. With the improvement of the physical model and computer technology, numerical simulation plays a more and more important role in wind tunnel flow field prediction.

In recent years, scholars have done some research on wind tunnel flow field. Gokcen established the numerical simulation model of flow field in the NASA Ames 10-MW TP3 arc-jet facility, and analyzed the uniformity of the flow field [9]. Ground numerically simulated the flow field inside the nozzle of 1.6 MW arc-heated wind tunnel by using the Fluent code [10,11]. Carandente established the numerical simulation model of the SPES facility to investigate the possibility to simulate the entry conditions into Titan’s atmosphere [12]. Wu built the wind tunnel in the Institute of Fluid Mechanics, Technical University Braunschweig, and established a numerical model to obtain the flow field inside the wind tunnel [13,14]. Although studies of flow field in the wind tunnel can be back to 1950s, the mechanism is not fully understood yet [15,16].

In this paper, the hypersonic wind tunnel of University of Science and Technology Beijing (USTB) is introduced. The compressible Navier–Stokes equations are solved to numerically simulate the flow field inside the nozzle and test section, and the calculation results are verified by experiment. The results can provide some guidance for the use of wind tunnel.
Facility Description

The hypersonic wind tunnel of USTB (Figure 1) is designed to test the performance TPS materials. It mainly consists air heating system, test chamber, gas ejector system, supply system and control system (Figure 2). The high temperature gas is formed in the air heater, and then accelerated through the nozzle to reach the required Mach number at the nozzle exit. The hypersonic wind tunnel consists of three different nozzles, so that the Mach number can range among 4, 5, 6, with a maximum of 600 s test time. The design parameters of the hypersonic wind tunnel at different conditions are shown in Table 1. In this study, flow field of 6 Ma is analyzed.

![Figure 1. The wind tunnel facility.](image)

![Figure 2. Schematic of the wind tunnel.](image)

### Table 1. Parameters of the wind tunnel.

| Case | Ma | Diameter of nozzle outlet [mm] | Total pressure [MPa] | Total temperature [K] | static pressure [Pa] |
|------|----|--------------------------------|----------------------|-----------------------|----------------------|
| 1    | 4  | Ф120                           | 0.883                | 868.1                 | 5475.2               |
| 2    | 6  | Ф150                           | 5.63                 | 1567.4                | 2511.2               |
| 3    | 5  | Ф200                           | 4.04                 | 2500                  | 3500                 |

Experimental Verification and Computational Model

Experimental Verification

Calibration of flow field is an effective method to test the quality of wind tunnel flow field, and can help us understand the flow field characteristics of the wind tunnel. The total pressure is measured by the pressure sensor. The temperature is measured by the thermocouple sensor. Due to the nozzle diameter is small, the single row calibration frame is accepted (Figure 3). Figure 4 shows the installation position of calibration frame in the wind tunnel.

![Figure 3. Location of calibration points.](image)

![Figure 4. Calibration of the wind tunnel.](image)

The Mach number of the flow field is calculated according to the total pressure and the wall static pressure as Eq. 1.

\[
Ma = \frac{1}{\sqrt{\frac{P_{\text{total}}}{P_{\text{wall static}}}}} - 1
\]
\[
\frac{P_{02}}{P_1} = \left[ \frac{2\gamma}{\gamma+1} Ma^2 - \frac{\gamma - 1}{\gamma + 1} \right]^{\frac{1}{\gamma-1}} \left[ \frac{\left(\frac{\gamma+1}{2}\right) Ma^2}{1 + \frac{\gamma - 1}{2} Ma^2} \right]^\frac{\gamma}{\gamma-1}
\]

Where \( P_{02} \) is the total pressure, \( P_1 \) is wall static pressure, \( \gamma \) is the specific heat ratio of gas.

**Computational Model**

The flow field is obtained by using the Fluent code. The compressible Navier–Stokes equations are solved to simulate the flow field inside the nozzle and test section. The Navier–Stokes equations are written as follows:

\[
\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0
\]

\[
U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ E \end{pmatrix}, \quad F = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ (E + p)u \end{pmatrix}, \quad G = \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + p \\ (E + p)v \end{pmatrix}
\]

Energy equation:

\[
E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho (u^2 + v^2)
\]

Ideal gas law:

\[
p = \rho RT
\]

Where \( \rho \) is the gas density, \( u, v \) are velocities along the \( x \) and \( y \) directions, respectively. \( E \) is the gas energy, \( p \) is the gas pressure, \( \gamma \) is the specific heat ratio of gas. \( R \) is the gas constant.

Figure 5 is the geometry of the hypersonic wind tunnel. In order to simplify the calculation, a symmetric model is adopted. Figure 6 is the computational grids of the wind tunnel, and the wall boundary mesh is refined. The total number of the grid of the wind tunnel are 28012. The inlet at the nozzle is set as total pressure and total temperature, the outlet at the diffuser exit is set as static pressure conditions, and the wall is a non-slip isothermal wall.

![Figure 5. Model of the wind tunnel geometry.](image1)

![Figure 6. Computational grids of the wind tunnel.](image2)
Results and Discussion

Analysis of the Mach Number

Figure 7 is the Mach number contours of the flow field in wind tunnel. As show in the figure, the compression wave reaches the axis and reflects at the x=0.5m in the nozzle, but the compression wave after reflection does not enter the uniform area. Therefore, the flow field in the core region of the test section is relatively uniform and will not be affected by compression wave. Mach number distribution can also be seen from the Figure 8. The compression wave formed in at x=0.5 m in the nozzle. The flow field of test section is uniform and the Mach number is 6.1.

Figure 9 shows the Mach number distribution along the radius of at the different section. There is a large area of uniformity in the wind tunnel test section. At the section of +0.1 m, +0.15 m from the nozzle exit, the Mach number increased at both sides. This is mainly due to the fact that these points are located in the expansion region of the jet, and also confirm the Mach distribution diagram in Figure 7. The test result of the flow Mach number at the nozzle exit in Figure 10 is slightly smaller than that of numerical simulation, the distribution of flow field uniformity agrees well with the numerical simulation results.

Analysis of the Temperature

The temperature of test section is an important parameter in wind tunnel test. Figure 11 is the static temperature contour of the wind tunnel nozzle and test section. Uniform core area is formed by the gas expansion through the nozzle. The temperature of the test section core area is 183K. As shown in Figure 12, the temperature at the throat of the nozzle drops sharply and is stable at 183 K in the test section.
Figure 11. Temperature distribution.  

Figure 12. Evolution of temperature along the centreline.  

Figure 13 is the static temperature distribution at different cross sections of the wind tunnel. As can be seen from the diagram, in the section of -0.05m, nozzle exit, +0.05m and +0.1 m, the temperature uniformity is greater than $\phi 0.1m$, due to the wall boundary layer effect, the temperature of both sides is increased. At the +0.15 m section, the uniform region decreases slightly, and the temperature at both sides is reduced.

Analysis of the Pressure

Figure 14 is pressure contour of the wind tunnel nozzle and test section. The pressure is rapidly reduced at the nozzle throat, and the core area of pressure formed in the test section, the pressure of the core area is 3183.9 Pa. The static pressure of the test chamber is about 272Pa. As shown in Figure 15, the axial distribution of the static pressure in the wind tunnel shows that the pressure at the throat of the nozzle drops sharply and is stable at 3183.9Pa in the test section.

Figure 16 is the pressure distribution at different sections of the wind tunnel. As can be seen from the diagram, at the section of -0.05 m, the nozzle exit and +0.05 m, the static pressure field uniformity is greater than $\phi 0.1m$, and the pressure at both sides decreases. At the +0.15 m section, the uniform region decreases slightly.
Summary

Numerical prediction and experimental verification of flow field in the wind tunnel of USTB are presented. The conclusions are as follows.

(1) The numerical simulation results are in agreement with the experimental results, and the calculation method can be used to calculate the hypersonic flow field.

(2) The Mach number, temperature and pressure distribution in the nozzle and test section are obtained by combining numerical calculation with test calibration. The diameter of flow field uniform zone is above φ0.1 m. The Mach number in uniform zone is 6.15, the temperature is 183 K and the pressure is 3183.9 Pa.

(3) The flow field in the wind tunnel is uniform, which can be used for service evaluation and failure mechanism test of thermal protective materials.

Acknowledgement

This work was supported by the National Key Scientific Instrument and Equipment Development Project of China [No. 2011YQ140145] and Hebei Science and Technology Department Research Fund [No. 17211117].

References

[1] Sziroczak, David, and Howard Smith. A review of design issues specific to hypersonic flight vehicles. Progress in Aerospace Sciences. 84 (2016): 1-28.

[2] Pichon, Thierry, and Renaud Barreteau. Thermal Protection Systems: Heritage-Development Status-Perspectives. 18th AIAA/3AF International Space Planes and Hypersonic Systems and Technologies Conference. 2012.

[3] Schmisseur, John D. Hypersonics into the 21st century: A perspective on AFOSR-sponsored research in aero thermodynamics. Progress in Aerospace Sciences. 72 (2015): 3-16.

[4] Koo, Joseph, Wai Ho, and Ofodi KeEzekoye. A review of ablation modeling for thermal protection systems. 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. 2006.

[5] Natali, Maurizio, Jose Maria Kenny, and Luigi Torre. Science and technology of polymeric ablative materials for thermal protection systems and propulsion devices: A review. Progress in Materials Science. 84 (2016): 192-275.

[6] Gokcen, Tahir, and Antonella Alunni. Flow Characterization Studies of the 10-MW TP3 Arc-Jet Facility: Probe Sweeps. 46th AIAA Thermophysics Conference. 2016.
[7] Leslie, John, and Dan Marren. Hypersonic test capabilities overview. US Air Force T&E Days 2009. 2009. 1702.

[8] Bensassi, K., A. Lani, and P. Rambaud. Unsteady simulation of hypersonic flow around a heat flux probe in ground testing conditions. International Journal of Heat and Mass Transfer. 113 (2017): 889-897.

[9] Gokcen, Tahir, John Balboni, and Antonella I. Alunni. Computational simulations of the 10-MW TP3 arc-jet facility flow. 45th AIAA Thermophysics Conference. 2015.

[10] Ground, Cody, Luca Maddalena, and Valerio Viti. Computational analysis and characterization of the UTA 1.6 MW arc-heated wind tunnel facility. 51st AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition. 2013.

[11] Ground, Cody, Luca Maddalena, and Valerio Viti. Numerical analysis and characterization of the dissociated flowfield inside a 1.6 MW arc-heated wind tunnel facility. Aerospace Science and Technology. 30.1 (2013): 306-314.

[12] Carandente, Valerio, et al. Experimental and numerical simulation, by an arc-jet facility, of hypersonic flow in Titan’s atmosphere. Experimental Thermal and Fluid Science. 48 (2013): 97-101.

[13] Wu, Jie, and R. Radespiel. Experimental investigation of a newly designed supersonic wind tunnel. Progress in Flight Physics–Volume. 7 7 (2015): 123-144.

[14] Wu, Jie, Xijin Liu, and Rolf Radespiel. RANS simulations of a tandem nozzle supersonic wind tunnel. Aerospace Science and Technology. 49 (2016): 215-224.

[15] Wu, Jie, and Tze How New. An investigation on supersonic bevelled nozzle jets. Aerospace Science and Technology. 63 (2017): 278-293.

[16] Wu, Jie, Pradip Zamre, and Rolf Radespiel. Flow quality experiment in a tandem nozzle wind tunnel at Mach 3. Experiments in Fluids. 56.1 (2015): 20.