Numerical Study on the Gas-Particle Two-Phase Jet Flow during Canister Launching Process

Xiao-yang Liang¹, Zhi-tan Zhou¹, Qi-rui Yu² and Gui-gao Le¹*

¹Department of Mechanical Engineering, Nanjing University of Science and Technology, Nanjing, Jiangsu 210094, China
²Sichuan Aerospace Institute of Systematic Engineering, Chengdu, Sichuan 610100, China
*E-mail: leguigao@njust.edu.cn

Abstract: The Euler-Lagrange model of Jet-gas and $AL_2O_3$ is established to study the impact of gas-particle jet flow during canister launching process. The gas phase is the Navier-Stokes equation of the two-component transport of gas and air, and the turbulence model is the realizable two-equation model. The airflow flux is numerically discretized via AUSM format, and the viscous dissipation flux is a second-order central difference. The movement trajectories of particles with different diameters ranging from 1 to 100 μm and the distribution characteristics of jet flow dynamic parameters in the launching box are studied. The jet impingement flow field results of gas-particle two-phase flow are compared with those without particle phase. The results show that the particle phase has a retardation and heat transfer effect on the expansion of jet flow, which affects the distribution of jet flow parameters in the launch canister. Furthermore, compared with the pure jet flow impingement effect, the velocity of the two-phase jet flow along the central axis decreases and the temperature increases. Meanwhile, the trajectory of the particle phase is related to the particle diameter. As the diameter of the particles becomes smaller, the flow ability is better and affected by turbulence, and a random movement area in the downstream area of the jet is formed. As the diameter of the particles becomes larger, a particle aggregation area is formed on the jet axis due to inertial force.

1. Introduction
In order to improve the specific impulse of solid rocket motors and suppress unstable combustion [1,2], a large amount of aluminum powder is often added to the propellant. When this aluminum-containing composite propellant is working, a large number of $AL_2O_3$ particles will be produced in the jet flow. Since the particle phase occupies a considerable proportion in jet flow, the flow field characteristics and flow field structure of jet flow are quite different from those of pure gas-phase jet flow. In the past, when studying the jet flow impingement field in the missile launch canister, the particle phase factor is often ignored, and pure gas flow analysis is performed, which led to certain errors in the impact load and flow parameter distribution results. It can be seen that to carry out further gas-solid jet flow research is of great significance for accurately predicting the impingement field effect and flow characteristics in the launch canister, and for the design and thermal protection of the launching device.

At present, gas-solid gas jet models are mainly classified into particle trajectory model (DPM) and two-fluid model (TFM). TFM uses the Euler-Euler calculation framework, which makes an quasi-fluid assumption for the particle phase and considers the gas phase and particle phase as two kinds of fluid mixture. DPM adopts Euler-Lagrange calculation framework, Euler method is used to describe gas-
phase motion, and Lagrange method is used for particle tracking. Chang et al.[3], simulated the two-phase inviscid flow in the nozzle based on the Mac Cormack format and the TFM model, giving the main effects of particle phase parameters on the flow field; Dupay et al.[4], used the TFM model to study the influence of alumina particles on the stability of the two-phase flow field in the engine; Golafshanid et al.[5], used the DPM method to numerically simulate the two-phase free jet of gas particles in the JPL nozzle and found that small particles follow the flow well and can fill the entire nozzle, while large particles tend to converge on the axis. Similarly, many researchers have achieved certain results in the study of the two-phase flow field in the nozzle by using DPM and TFM methods, but there is few reports on the impingement flow field of the two-phase gas jet in the launch canister.

In order to solve the above problems, this paper uses the Euler-Lagrange two-phase flow analysis method to numerically simulate the flow field characteristics of a jet containing solid particles impinging on the wall of the launch canister. The flow characteristics of gas-solid coupling jet are studied, and the distribution characteristics of the particles and the influence of the solid particle size on the impinging jet are analysed and compared with the impinging flow field of a pure gas-phase gas jet.

2. Computational model

The structure of the computational model in this paper is shown in Fig. 1. The model mainly includes launch canister, guide rail (missile orientation device), missile, and nozzle. The center of the bottom of the launching canister is defined as the origin of coordinates. The X axis coincides points to the missile launching direction. The Y axis and Z axis are normal to the walls along both sides of the launch canister. The nozzle exit diameter is D. The length and width of the launch canister are 5.5D, the diameter of the missile is 2D, and the distance between the bottom of the missile and the bottom of the launch box is 29D.

A high-quality grid can not only improve the stability of numerical calculations, but also help improve the accuracy of calculations. In this paper, multi-block grid is used to partition computational domain. Denser meshes are generated in places with large flow gradients, such as the boundary layer region, near the nozzle walls, and the core region of the jet. In order to meet the requirements of the wall function in the turbulence calculation for the grid near the wall, 12 prismatic layer grids are provided in the boundary layer near the wall. The height of the first layer near the wall is about 30 mm to ensure that the typical Y + value near the wall is less than 3. Fig. 2 shows the computational model grid system, and the total number of grids is 8.16 Million hexahedral elements.
In the calculation, the gas-phase and particle-phase boundary conditions are: (1) gas-phase boundary conditions: nozzle inlet cross-section is defined as the pressure inlet conditions, given the total pressure and total temperature; the outer boundary is defined as the local ambient pressure and temperature using the pressure outlet. All walls such as the launch canister, guide rail, etc. are defined as the boundary conditions of the non-slip adiabatic wall; (2) Particle phase boundary conditions: In the nozzle inlet cross-section, the midpoint of each grid edge at the inlet is taken as the inflow point of the particles. The initial velocity, temperature, and incidence angle of the particles are the same as those of the gas phase, and the collision between the particles and the wall is a fully elastic collision.

3. Model validation
In order to verify the accuracy of the numerical method in this paper for predicting the flow field structure in the jet, this paper carries out a comparative verification between numerical and experimental validation with the experiments conducted by Lamont and Hunt et al[6]. In the experiments, the nozzle exit pressure ratios PR are 1.2 and 2.0, where PR is the ratio of the static pressure at the nozzle exit to the external ambient pressure. The plate diameter used in the experiment is 300mm (10 times of the nozzle outlet diameter), and the nozzle outlet diameter $D_n$ is 15mm. Fig. 3 (a) shows that the distance from the center of the nozzle exit plane to the plate is $2D_n$ and the angle between the nozzle axis and the plate is $30^\circ$. Fig. 3 (a) shows that the distance from the center of the nozzle exit plane to the plate is $2D_n$ and the angle between the nozzle axis and the plate is $30^\circ$. When the static pressure ratio is $PR = 2.0$, the calculated density gradient and Mach number distribution graphs are compared with the experimental schlieren graph. Fig. 3 (b) shows that the distance from the center of the nozzle exit plane to the plate is $3D_n$ and the angle between the nozzle axis and the plate is $45^\circ$. It can be seen from the figure that the flow field structure agrees well with the experiment.
4. Results and discussion

4.1 Results of flow field

Fig. 4(a) and 4(b) show the comparison of Mach number and temperature distribution for two types of flows in the launch box, pure gas-phase and particle-containing phase, respectively, in the upper half of the central axial diagram is the calculated result of pure gas-phase flow, and the lower half is the calculated result of gas-particle two-phase flow. A series of Mach disks formed by the underexpanded jet in the box can be clearly seen in the figure. Compared with the pure gas-phase jet, the solid particles have certain inertia, which hinders the gas jet and heat transfer, so the expansion and compression performance of the gas flow are reduced, and the flow field structure in the exhaust plume of the two-phase flow field is changed. It can also be seen in Fig. 4(b) that the temperature of the two-phase flow field is higher than the temperature of the flow field under pure gas-phase conditions at the same position.

Figure 4. Mach number and Temperature contours of one phase and two phase at the oxy plane

Fig. 5 (a) and 5 (b) are the Mach number and temperature distribution along the nozzle axis under pure gas and gas-solid two-phase flow conditions, respectively. It can be seen from Fig. 5(a) that at the same position, the Mach number along the axis of the nozzle under the condition of pure gas phase is greater than the Mach number distribution when $\text{Al}_2\text{O}_3$ particles are included. The gas temperature in the two-phase flow field is 240K higher than the maximum value of the single-phase flow, which shows that solid particles have a great influence on the flow field structure in the launch canister.

Figure 5. Mach number and Temperature distributions along the nozzle axis
4.2 Particle trajectory
The trajectories of the particles in the nozzle for particle diameters of 5 μm, 10 μm, 25 μm, 50 μm and 90 μm are given in Fig. 6. It can be seen that there are significant differences in the trajectories of particles with different diameters in the nozzle. When the particle mass is small (5 μm), the particles are easily affected by the continuous phase flow field, and the particles can be filled with the expansion of the airflow after entering the expansion section. As the particle diameter increases, the direction of motion changes slower than the gas expansion.

Therefore, the larger the particle diameter, downstream of the nozzle throat, near the nozzle wall will form a particle trajectory cannot reach the region, namely the particle-free zone, and the larger the particle diameter, the more obvious the particle-free zone downstream of the throat.

![Figure 6. Particles trajectory for the five sizes of particles](image)

![Figure 7. Particle trajectories of five particle sizes in the launch canister](image)
The trajectories of particles of different diameters entering the launch box from the nozzle exit section are given in Fig. 7. It can be seen from the figure that small-sized particles are easily affected by turbulence. After leaving the nozzle exit plane, the trajectory will be in a random motion state, so the probability of small-sized particles colliding with the wall of the launching canister is higher. It can also be seen from the diagram that with the increase of particle diameter (the particle diameter exceeds 50 μm), the inertia increases and the particles move closer to the axis under the action of inertial force.

5. Conclusion
In this paper, the Euler-Lagrangian method is used to model and numerically simulate the flow of a gas-solid two-phase jet inside a missile launch box, taking into account the two-way coupling effect between the two phases. The Mach number, temperature and trajectories of particles of different sizes in the launch box under two-phase flow conditions are obtained. The results show that: (1) \( AL_2O_3 \) particle size has a great influence on the compressibility of the gas jet. (2) In a two-phase flow, the Mach number along the center axis of the nozzle is smaller than that of a pure gas flow. The temperature is higher than the temperature under pure gas phase conditions. (3) Through the analysis and comparison of the \( AL_2O_3 \) particle trajectory, it is concluded that the small size particles follow the jet flow well. Due to the influence of drag and turbulence, the trajectory of small particles in the launch canister is closer to the gas trajectory, so it will have a significant impact on the wall of the launch canister. Large size particles will converge in the nozzle center axis and form a particle aggregation zone.

References
[1] De Luca L. (2007) Burning of aluminized solid rocket propellants: from micrometric to nanometric fuel size. International Autumn Seminar on Propellants, Explosives and Pyrotechnics. Science Press. pp. 277-289.
[2] Liu Z, Li S, Liu M. (2017) Experimental investigation of the combustion products in an aluminised solid propellant. Acta Astronautica,133: 136-144.
[3] Grossi M, Bianchi D, Favini B. (2020) Modeling Multiphase Effects on Pressure Oscillations in Solid Propulsion. AIAA Propulsion and Energy 2020 Forum. NO. 3927.
[4] Dupays J. (2002) Two-phase unsteady flow in solid rocket motors. Aerospace Science and Technology, 6(6): 413-422.
[5] Golafshani M, Loh H T. (1989) Computation of two-phase viscous flow in solid rocket motors using aflux-split eulerian-lagrangian technique. 25th Joint Propulsion Conference. NO. 2785
[6] Lamont P J, Hunt B L. (1980) The impingement of underexpanded, axisymmetric jets on perpendicular and inclined flat plates. Journal of Fluid Mechanics. 100(3): 471-511