Design and simulation of a micro thrust solenoid valve nozzle based on CFD&DOE optimization analysis

Zhiqin Zhuo 1,2, Zongfeng Li* 2,3
1University of Chinese Academy of Sciences, Beijing, China
2Technology and Engineering Center for Space Utilization, Chinese Academy of Sciences, Beijing, China
3Key Laboratory of Space Utilization, Chinese Academy of Science, Beijing, China

Email: lzfeng@csu.ac.cn

Abstract. Supersonic micro nozzles are widely used in aerospace propulsion, supersonic gas assisted processing and other fields. In this paper, combining numerical analysis and computational fluid dynamics (CFD) simulation methods, using a fast parameter optimization method, the design of experiment response surface method (DOE_RSM) to optimize the design of a micro solenoid valve Laval nozzle about single element cold air propulsion. Simulate the flow characteristics of the nozzle under different configurations and working conditions. Using the three-dimensional model to draw the structure grid and analyse the influence of the micro solenoid valve nozzle configuration on the thrust, effective specific impulse, outlet flow rate, mass flow rate, and outlet cross-sectional pressure difference under different environmental conditions. Determine the significant influencing factors and carry out parameter optimization. Different from the traditional rocket or air assisted processing nozzles, the satellite micro-thrust nozzles need to consider frequent switching actions. In this paper, we finally obtained the optimized configuration and size of the solenoid valve nozzle. Based on the calculation results, a cold-air propeller which include replaceable nozzles and can be used under normal pressure was designed, to provide a reference for cold air propulsion in the space station cabin and ground supersonic gas assisted processing.

1. Introduction
Laval nozzle jet propulsion has a wide range of applications in the aerospace field, such as rocket propulsion, satellite attitude, orbit control and space free-floating robots. Based on microelectronics technology, nanotechnology, etc., spacecraft are tending to be miniaturized. Much of them are need to use micro nozzles as propulsion devices [1]. In addition, micro supersonic nozzles are widely used in the fields of supersonic cold spraying process and supersonic gas assisted machining [2]. Under a certain environmental pressure, part of the nozzle thrust will be affected by the outlet section pressure difference. Laval nozzle is a kind of convergent-expanded nozzle and the airflow reaches the critical point at the throat of the nozzle section and expands to supersonic speed in the expansion section [3].

Long distance between the solenoid valve and the nozzle inlet will lead to control hysteresis easily, which will affect the control accuracy and sensitivity. Therefore, the solenoid valve is usually installed as close to the inlet of the nozzle as possible [4]. A small solenoid valve diameter will also affect the nozzle performance, but in order to increase the switching frequency, the solenoid valve diameter cannot be made too large. For the floating robot in the space capsule propulsion by cold air, the air
source mainly comes from the small power compression pump carried by itself, with limited pressure and low pressure difference between the nozzle inlet and outlet. This paper mainly focuses on the analysis and design of the solenoid valve nozzle under a space station atmospheric pressure environment.

Design of Experiment (DOE) is a method to study and process the relationship between multiple factors and multiple response variables. The results can be obtained at a smaller test scale through the optimization analysis of multiple response parameters to the research object [5]. Response surface methodology (RSM) is the main method of DOE experimental design. Traditional CFD forward simulation only one target can be analyzed at a time. With the scientific arrangement of test points by RSM, the design space can be fully explored, reduce the number of experiments, and determined the feasibility design quickly [6][7].

The flow passage diagram of solenoid valve nozzle is shown in the figure 1.

![Figure 1. Schematic diagram of micro solenoid valve nozzle.](image)

2. Mathematical model

For the small Laval nozzle working in the design state, the gas flow in the nozzle can be simplified to a steady flow of one-dimensional isentropic gas [8]. The main parameters of nozzle airflow are shown in table 1.

| Define                  | Symbol | Define                  | Symbol |
|-------------------------|--------|-------------------------|--------|
| Mass Flow               | Qm     | Pressure                | P      |
| Sectional area          | A      | Temperature             | T      |
| Flow velocity           | V      | Density                 | ρ      |
| Gas constant            | R      | Adiabatic index         | k      |

2.1. Numerical calculation

According to the energy equation, the basic parameters of the nozzle can be obtained from the following equation. Stagnation parameters: Stagnation temperature $T^*$, Stagnation pressure $P^*$, Stagnation enthalpy $h^*$ [8]. For the reversible adiabatic one-dimensional steady flow of a compressible ideal gas, Stagnation parameters can be calculated by formula 1, 2 and 3.

\[
T^* = T + \frac{V^2}{2c_p}
\]

\[
P^* = P \left(\frac{T^*}{T}\right)^{\frac{k}{k-1}}
\]
\[ h^* = c_p T^* \]  

\[ T \text{ and } v \text{ are the temperature and velocity of air flow on any section. } P \text{ and } T \text{ are the pressure and temperature of air flow on any section. the stagnation enthalpy } h^* \text{ equal to the sum of the enthalpy and the kinetic energy of the airflow on any section [7].} \]

According to the above formula, the calculation formula for the adiabatic flow velocity of the gas on any section of the nozzle can be obtained.

\[ v_2 = \sqrt{2(h^* - h_2)} = \sqrt{\frac{2k}{k-1} R T^* \left[ 1 - \left( \frac{P_2}{P^*} \right)^{\frac{k-1}{k}} \right]} \]  

\[ v_2 \] is outlet section velocity, adiabatic index \( k = 1.4 \), ideal gas constant \( R = 296.8 \text{ J} / (\text{kg} \cdot \text{k}) \), \( P_2 \) is outlet section gas pressure.

\[ Q_m = \frac{k}{\sqrt{R}} \left( \frac{z}{k+1} \right)^{\frac{k+1}{k-1}} \frac{P^*}{\sqrt{T^*}} \pi R_1^2 \]  

\( Q_m \) is nozzle mass flow, \( R_1 \) is throat section radius. Thus the propulsive force can be calculated.

\[ F = Q_m (v_2 - v_0) + (P_2 - P_b) A_2 \]

\( v_0 = 0 \) is the flight speed, \( P_b \) is the environmental pressure, and \( A_2 \) is the outlet section area. Take \( g_0 = 9.8 \text{N/kg} \), effective specific impulse:

\[ I_{sp} = \frac{F}{Q_m g_0} \]  

2.2. Basic size optimization

As it shown in figure 1, Laval nozzle consists of four parts: inlet stable section, subsonic contraction section, throat and supersonic expansion section [9].

Inlet stable section: The inlet stabilization section is used to stabilize the nozzle inlet flow and reduce turbulence caused by the solenoid valve switch. The influence parameters include length and inner diameter. Length is generally take 15-20 mm [9]. Takes the radius of the stable section \( R_0 \) as a variable to evaluate its impact.

Contraction section: The main function of the contraction section is to accelerate the air flow uniformly, until the gas flow reaches the critical sound speed in the throat. Its major configuration has cone-shaped and Vitoshinsky streamline. For the micro Laval nozzle, the line shape of the constricted section has little effect on the nozzle flow field [2]. Thus used the cone-shaped configuration. According to the empirical formula, the length of the constricted section is generally taken as \( L_1 = 0.5 \sim 1D_0[9] \).

Expansion section: Usually the configuration of the expansion section of the Laval nozzle is bell, cone and trumpet. The article [10] uses Monte Carlo method and CFD method to carry out research and show that the thrust performance of the cone-shaped nozzle and trumpet nozzle is basically similar, while the bell-shaped nozzle has the smallest thrust. In consideration of thrust performance and processing difficulty, the expansion section chooses a simple cone. The nozzle outlet velocity has a non-axial component, and it will cause friction loss and eddy current loss in actual flow. Excessive cone angle of the expansion section will aggravate this eddy current loss, while too small cone angle will increase friction loss of the wall boundary layer. The cone angle of the expansion section can be selected from 8° to 12° [9]. In this paper, \( \theta = 8 \text{°} \) is chosen.

Therefore, the throat section radius \( r_1 \), Exit section radius \( r_2 \), Length of expansion section \( L_2 \) and its cone angle \( \theta \) have the following relationship:

\[ r_2 = r_1 + L_2 \tan \frac{\theta}{2} \]  

\[ \theta = 8 \text{°} \]
In order to simplify the difficulty of model parameters, select $R_1$ and $\theta$ as constant, and set $R_0$ and $L_2$ as variables. $R_2$ is set as a variable indirectly. Carry out the next CFD simulation.

3. CFD&DOE fluid simulation

The CFD&DOE process is divided into two parts. First, designed the initial CFD value and calculated, and then set the DOE optimization conditions and objectives to optimal design. The following aspects should be paid attention to when establishing CFD simulation of solenoid valve nozzle, the basic process of CFD&DOE simulation is shown in figure 2 and figure 3.

### Figure 2. CFD simulation flow chart

### Figure 3. DOE RSM simulation flow chart

3.1. Structure meshing

Perform hexahedral structural meshing on the 3D nozzle model. Theoretically, the denser the grid, the more accurate the calculation result. But it will lead to a relatively large amount of calculation and a long calculation cycle. According to the model grid sensitivity inspection, it is finally determined that the number of micro Laval nozzle grids is between 1 million to 1.5 million. As it shown in figure 4.

### Figure 4. Laval nozzle hexahedral structural meshing

3.2. Nozzle scale effect

There is no unified standard for scale division. Generally, the scale between 1μm and 1mm is called the micro scale, and above 1mm is called the macro scale [11]. The throat diameter of the nozzle is
used as the standard for scale division. For micro-scale nozzles, the performance loss caused by the gas viscosity effect cannot be ignored. The smaller the nozzle size, the more viscous and frictional effects of the gas [12]. When the throat diameter is less than 0.35mm, the flow of gas cannot be predicted by the inviscid compressible flow model [2]. Thereby, compressible flow model was used in this simulation.

3.3. Flow model
There are two kinds of flow models in gas flow simulation: molecular hypothesis of rarefied gas dynamics and continuous assumption of classical gas dynamics. According to the range of Knudsen number $K_n$ to judge whether the fluid is suitable for the continuity hypothesis of gas dynamics [13]. Calculate the Knudsen number of an ideal gas by formula 9.

$$Kn = \frac{K_B T}{\sqrt{2\pi} d_n^2 P^* L_c}$$  \hspace{1cm} (9)

$K_B = 1.3806505 \times 10^{-23} J/K$ is Boltzmann constant. Take $T=296K$, nitrogen molecular diameter $d_n = 3.64 \times 10^{-10} m$ and $P^* = 5 \times 10^5 Pa$. $L_c$ is the characteristic length of the nozzle, the diameter of the throat is taken as the characteristic length $L_c = 4 \times 10^{-4} m$. Incorporating formula 9 to calculate. $K_n = 3.4712 \times 10^{-5}$, the Knudsen number is less than 0.001, which means the flow is continuous. The continuum model can be used, and take the N-S equation without slip boundary conditions to describes the fluid.

The dimensionless Reynolds number can be used to determine what kind of flow the fluid belongs to [8]. Reynolds number calculation formula:

$$Re = \frac{4Q_m}{\pi D_t \mu_0}$$  \hspace{1cm} (10)

Take the mass flow rate of nozzle $Q_m = 1.6 \times 10^{-4} kg/s$, throat diameter $D_t = 4 \times 10^{-4} m$ and gas dynamic viscosity at stagnation state $\mu_0 = 1.7805 \times 10^{-5} kg/m^2 \cdot s$. Substitute into the formula 10 and get $Re = 28604$. For the flow in the pipe, $Re > 2300$ means turbulent, CFD nozzle flow was calculated using a turbulent flow model. In Fluent, the Navier-Stokes equation based on Reynolds time averaging can completely describe the nonlinear partial differential governing equations of turbulent flow [14]. The basic N-S equations are shown in equations 11, 12.

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (11)

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2}$$  \hspace{1cm} (12)

The commonly used available turbulence models based on Reynolds time average in Fluent are k–$\varepsilon$ and k–$\omega$ model. The standard k–$\varepsilon$ model has good stability and is widely used, but it is not suitable for the simulation of severe pressure gradient, separation and strong curvature flow. The Realizable k–$\varepsilon$ model is suitable for complex shear flow, medium vortex flow and local transition flow. SST k–$\omega$ is suitable for boundary layer with high resolution. Refer to the method in Article [15] and considering the calculation cost and model accuracy requirements, the Realizable k–$\varepsilon$ model is used for calculation in this project.

The boundary conditions are shown in tables 2 and 3.

| Type          | Parameter   |
|---------------|-------------|
| Flow model    | Continuity model |
| Solver type   | Pressure-Based |
| Models        | Realizable k-epsilon |
| Materials     | N$_2$       |

Table 2. Fluent simulation parameter table.
Total Temperature 296K  
Total pressure 500000Pa  
operating pressure 100000Pa

| Domain | Boundaries | Type        |
|-------|------------|-------------|
| solid | inlet      | Pressure-inlet |
|       | outlet     | Pressure-outlet |
| Wall  |            | Wall        |

The normal turbulence intensity is 1 ~ 5%. For circular inlet internal flow, select the default value of 5%. The turbulent viscosity ratio is 10 ~ 100, and 10 is selected.

3.4. DOE_RSM optimization
Based on the initial CFD simulation, the basic process of DOE_RSM optimization is shown in Figure 3: In Geometry modelling, set the expansion section length \( L_2 \) as parameter 1, select \( R_1 = 0.2 \text{mm} \) and \( \theta = 8^\circ \). According to formula 8, the direct relationship between the outlet section radius \( R_2 \) and parameter 1 can be calculated. Set the radius \( R_0 \) of the inlet stable section to parameter 2. Use ICEM CFD to divide the grid, and in the Fluent setup, set the environmental pressure as parameter 3. Enter other basic parameters to solve. After calculation completed, set the outlet cross-section thrust \( F \), flow velocity \( v \), mass flow \( Q_m \) and the pressure difference \( P \) between outlet cross-section and the environmental as response parameters in the post-processing Results. Then, set each parameter range, RSM type and number of design points in DOE_RSM. The optimization objective is set in direct optimization to maximize the thrust \( F \) of nozzle outlet section and set its importance as "higher"; minimize the mass flow rate of throat section \( Q_m \) and set its importance as "lower". According to the preliminary calculation of Matlab, the specific parameters can be set as shown in table 4.

| Input Parameters | Value range | Output Parameters | Value range |
|------------------|-------------|-------------------|-------------|
| \( R_0 / \text{mm} \) | 0.32~0.42   | Outlet F          | N           |
| \( L_2 / \text{mm} \) | 0.2~1.2     | Outlet V          | m/s         |
| \( R_2 / \text{mm} \) | 0.214~0.284 | Outlet P          | Pa          |
| \( P_b / \text{KPa} \) | 80~105      | \( Q_m \)         | kg/s        |
| \( R_1 / \text{mm} \) | 0.2         |                   |             |
| \( P^* / \text{Pa} \) | 500000      |                   |             |
| \( T^* / \text{K} \) | 296         |                   |             |
| \( R_1 / \text{mm} \) | 0.2         |                   |             |

4. Calculation result
In order to ensure that the optimal conditions are included in the DOE response surface and find the best advantages quickly, we first use Matlab numerical analysis to determine some optimized parameters, reasonable experimental ranges, simplified experimental conditions. The simulation results include numerical simulation and CFD simulation.

4.1. Numerical analysis and calculation results of MATLAB
Set the nozzle \( R_1 = 0.2 \text{mm} \), the inlet pressure \( P_0 \): 0.3~ 0.8MPa, and the environmental pressure \( P_b \) is 0.1MPa. Calculated the effective specific impulse change of the nozzle and shown the result in figure 5.
In figure 5, under the same environmental pressure, the effective specific impulse $I_{sp}$ can reach more than 50s at an inlet pressure of 0.8MPa, while at an inlet pressure of 0.3MPa is less than 45s. The calculation result shows that increasing the inlet pressure appropriately and increasing the pressure difference between the inlet and outlet can increase $I_{sp}$. However, in view of the limited pressurizing capacity of the compression pump carried by the robot in the space cabin. It can usually only be pressurized to about 0.7~1 MPa, the larger inlet pressure cannot be provided after the pressure regulator valve, so we choose inlet pressure $P_0 = 0.5MPa$ to further analysis.

Set the $P_0 = 0.5MPa$, the $P_b = 0.1MPa$ and respectively calculate $F$ and $I_{sp}$ when the Laval nozzle throat radius $R_1 = 0.1, 0.15, 0.2, 0.25mm$. The calculation results are shown in figure 6.

From figure 6-a, when $R_1 = 0.2mm, 0.15mm$, we can see the nozzle thrust cannot reach more than 50mN, and the nozzle processing is difficult. When the throat section $R_1 = 0.25mm$, the thrust can reach more than 100mN. Between 50-100mN thrust value, choose $R_1 = 0.2mm$.

According to the conservation of mass, when the total pressure and total energy of the air flow remain unchanged, the mass flow of the nozzle depends on the throat section area \[8\]. The greater the mass flow, the greater thrust, but the corresponding $I_{sp}$ may not increase. As it shown in figure 6-b, in the same inlet and outlet pressure values, and the maximum effective specific impulse of different throat sections is between 45 and 50s, the change trend is basically the same.

For design a solenoid valve nozzle with a thrust value of 50-100mN, use CFD further analyse the throat section radius $R_1 = 0.2mm$. 

---

**Figure 5. Effective specific impulse curve of solenoid valve nozzle**

In figure 5, under the same environmental pressure, the effective specific impulse $I_{sp}$ can reach more than 50s at an inlet pressure of 0.8MPa, while at an inlet pressure of 0.3MPa is less than 45s. The calculation result shows that increasing the inlet pressure appropriately and increasing the pressure difference between the inlet and outlet can increase $I_{sp}$. However, in view of the limited pressurizing capacity of the compression pump carried by the robot in the space cabin. It can usually only be pressurized to about 0.7~1 MPa, the larger inlet pressure cannot be provided after the pressure regulator valve, so we choose inlet pressure $P_0 = 0.5MPa$ to further analysis.

Set the $P_0 = 0.5MPa$, the $P_b = 0.1MPa$ and respectively calculate $F$ and $I_{sp}$ when the Laval nozzle throat radius $R_1 = 0.1, 0.15, 0.2, 0.25mm$. The calculation results are shown in figure 6.

**Figure 6. THRUST F and $I_{sp}$ curve of solenoid valve nozzle**

From figure 6-a, when $R_1 = 0.2mm, 0.15mm$, we can see the nozzle thrust cannot reach more than 50mN, and the nozzle processing is difficult. When the throat section $R_1 = 0.25mm$, the thrust can reach more than 100mN. Between 50-100mN thrust value, choose $R_1 = 0.2mm$.

According to the conservation of mass, when the total pressure and total energy of the air flow remain unchanged, the mass flow of the nozzle depends on the throat section area \[8\]. The greater the mass flow, the greater thrust, but the corresponding $I_{sp}$ may not increase. As it shown in figure 6-b, in the same inlet and outlet pressure values, and the maximum effective specific impulse of different throat sections is between 45 and 50s, the change trend is basically the same.

For design a solenoid valve nozzle with a thrust value of 50-100mN, use CFD further analyse the throat section radius $R_1 = 0.2mm$. 

---

**Figure 5. Effective specific impulse curve of solenoid valve nozzle**

In figure 5, under the same environmental pressure, the effective specific impulse $I_{sp}$ can reach more than 50s at an inlet pressure of 0.8MPa, while at an inlet pressure of 0.3MPa is less than 45s. The calculation result shows that increasing the inlet pressure appropriately and increasing the pressure difference between the inlet and outlet can increase $I_{sp}$. However, in view of the limited pressurizing capacity of the compression pump carried by the robot in the space cabin. It can usually only be pressurized to about 0.7~1 MPa, the larger inlet pressure cannot be provided after the pressure regulator valve, so we choose inlet pressure $P_0 = 0.5MPa$ to further analysis.

Set the $P_0 = 0.5MPa$, the $P_b = 0.1MPa$ and respectively calculate $F$ and $I_{sp}$ when the Laval nozzle throat radius $R_1 = 0.1, 0.15, 0.2, 0.25mm$. The calculation results are shown in figure 6.

**Figure 6. THRUST F and $I_{sp}$ curve of solenoid valve nozzle**

From figure 6-a, when $R_1 = 0.2mm, 0.15mm$, we can see the nozzle thrust cannot reach more than 50mN, and the nozzle processing is difficult. When the throat section $R_1 = 0.25mm$, the thrust can reach more than 100mN. Between 50-100mN thrust value, choose $R_1 = 0.2mm$.

According to the conservation of mass, when the total pressure and total energy of the air flow remain unchanged, the mass flow of the nozzle depends on the throat section area \[8\]. The greater the mass flow, the greater thrust, but the corresponding $I_{sp}$ may not increase. As it shown in figure 6-b, in the same inlet and outlet pressure values, and the maximum effective specific impulse of different throat sections is between 45 and 50s, the change trend is basically the same.

For design a solenoid valve nozzle with a thrust value of 50-100mN, use CFD further analyse the throat section radius $R_1 = 0.2mm$.
4.2. Fluent simulation and DOE optimization calculation results
Take CFD simulation for R1=0.2mm nozzle, and use DOE response surface method to optimize the calculation. First calculated the response surface between each response parameter and the input variable.

Figure 7. the response surface between response parameter and the input variable
In figure 7-a, b, c, the x and y axes are the length of the expansion section $L_2(R_2)$ and the radius of the inlet stabilization section $R_0$. The z axis is the outlet velocity $V$, the mass flow $Q_m$ and the thrust $F$.
In figure 7-a, it can be seen that with the increase of $L_2(R_2)$, the outlet $V$ presents a trend of first increasing to decreasing, while the influence of $R_0$ on $V$ is smaller. The results are basically consistent with the numerical analysis. When the outlet area reaches a certain value, the airflow produces shock waves at the outlet and moves towards the expansion section.
In figure 7-b, the nozzle $Q_m$ increases with the enlarge of $L_2(R_2)$ and $R_0$, the change range is about 0.159–0.163g/s, the change rate is about ±1.25%. The basic change law is that with the increase of $R_0$ and $L_2(R_2)$, it shows a trend of first increase and then decrease, which has a limited impact on the thrust.
In figure 7-c, the thrust $F$ has a certain relationship with $R_0$ and $L_2(R_2)$. Thrust $F$ increases with $R_0$. While $L_2(R_2)$ increases, thrust $F$ shows a trend of first increasing and then decreasing.

Further analyse the two-dimensional response curve of each parameter.

Figure 8. The relation curve between $F$, $Q_m$ and $L_2(R_2)$ under different environmental pressures
In figure 8-a, the nozzle thrust $F$ first increases and then decreases rapidly as the $L_2(R_2)$ increases. Under different environmental pressure conditions, the peak thrust range is different. It can be seen from the figure that, when $L_2$ is set at 0.3–0.4mm, the nozzle thrust varies less under different environmental pressures, but at the same time, the nozzle thrust is also lower relatively. $L_2$ takes the value of 0.5–0.6mm, the nozzle thrust within the ambient pressure range is greater than 79mN, and the variation range is 79–79.5mN. The overall thrust of the solenoid valve nozzle is higher.
Figure 8-b shows the relationship curve between nozzle $Q_m$ and $L_2(R_2)$ under different ambient pressures. $Q_m$ increases slightly with the increase of the outlet cross-section $L_2(R_2)$. $L_2$ takes a value of 0.5–0.6mm, and the mass flow rate under different environmental pressures varies between 0.159–0.1655g/s. $F$ is above 79mN, and according to formula 6, $I_{sp}$ is above 48.8s.

![Figure 8-b](image)

**Figure 8-b. Relationship curve between nozzle $Q_m$ and $L_2(R_2)$ under different ambient pressures.**

$Q_m$ increases slightly with the increase of the outlet cross-section $L_2(R_2)$. $L_2$ takes a value of 0.5–0.6mm, and the mass flow rate under different environmental pressures varies between 0.159–0.1655g/s. $F$ is above 79mN, and according to formula 6, $I_{sp}$ is above 48.8s.

Figure 9. The relation curve between $F$, $Q_m$ and $R_0$ under different environmental pressures

Take the value $L_2 = 0.59$mm, fit the curve of the relationship between $R_0$ and $F$. As shown in figure 9-a, the thrust $F$ first increases and then decreases slightly with the increase of $R_0$. The maximum value is around 0.4–0.41mm. Thrust increased from 77mN to over 79mN, an increase of about 2.6%. Figure 11-b is the relationship curve between $Q_m$ and $R_0$. The nozzle $Q_m$ increases slightly with $R_0$ increase, and the maximum value is between 0.39–0.4mm. The corresponding effective specific impulse increases from 49.1 to above 50. It can be seen that appropriately increasing the cross-sectional area of the inlet stabilization section is beneficial to improve the performance of the solenoid valve nozzle.

![Figure 9.](image)

**Figure 9. The relation curve between $F$, $Q_m$ and $R_0$ under different environmental pressures.**

Take the value $L_2 = 0.59$mm, fit the curve of the relationship between $R_0$ and $F$. As shown in figure 9-a, the thrust $F$ first increases and then decreases slightly with the increase of $R_0$. The maximum value is around 0.4–0.41mm. Thrust increased from 77mN to over 79mN, an increase of about 2.6%. Figure 11-b is the relationship curve between $Q_m$ and $R_0$. The nozzle $Q_m$ increases slightly with $R_0$ increase, and the maximum value is between 0.39–0.4mm. The corresponding effective specific impulse increases from 49.1 to above 50. It can be seen that appropriately increasing the cross-sectional area of the inlet stabilization section is beneficial to improve the performance of the solenoid valve nozzle.

Figure 10. Local Sensitivity histogram

![Figure 10.](image)

**Figure 10. Local Sensitivity histogram.**

Figure 10 shows a sensitivity histogram of each input parameter of the response surface. It can be seen from the figure that at a certain throat $R_1$, the nozzle thrust $F$ has the highest sensitivity to the inlet stable section $R_0$, while the outlet velocity $V$ has the highest sensitivity to the expansion section $L_2(R_2)$. And the sensitivity is lower for changes in the environmental pressure $P_b$. Therefore, it can be seen that the solenoid valve nozzle has a certain adaptability to the range of environmental pressure changes in the space station.

The direct optimization calculation results give three candidate points, as it shown in table 5.

| Name | 1   | 2   | 3   |
|------|-----|-----|-----|
| $R_0$/mm | 0.391 | 0.400 | 0.403 |
| $R_2$/mm | 0.236 | 0.241 | 0.241 |
The optimization results show that the range of $R_0$ is about 0.391~0.403mm, $R_2$ is 0.236-0.241mm, the thrust value can reach more than 79.3mN, and the effective specific impulse can reach 50.8s. According to formula 6, the thrust value generated by the gas momentum conversion is about 75mN, accounting for 95%. And the other thrust generated by the pressure difference accounting for about 5%. It can be seen that the existence of environmental pressure will have a certain impact on the nozzle performance.

Figure 11 shows the cloud diagram of velocity streamline diagram and pressure cloud diagram of the Laval nozzle. It can be seen that the air flow velocity increases near the solenoid valve. The existence of the steady flow section slows down the gas flow velocity and reduces the turbulence of the gas flow at the entrance of the Laval nozzle contraction section. Stabilizes the airflow, and make the airflow finally achieves supersonic speed in the expansion section. Under the influence of ambient pressure, it can be seen that the pressure value decreases as the gas flow rate increases, and at the exit decreases to near negative.

Figure 11. Velocity and pressure cloud diagram of nozzle airflow

Finally, the configuration parameters of the solenoid valve nozzle can be determined as: $R_0 = 0.4mm; R_1 = 0.2mm; R_2 = 0.24mm; L_2 = 0.59mm$. The configuration of the solenoid valve nozzle can be shown in figure 12.

5. Conclusion
The study found that adding a steady flow section between the cut-off surface of the solenoid valve and the entrance section of the Laval nozzle helps to increasing the nozzle thrust and the effective specific impulse of the gas. While the excessive sectional area of the stable section may cause shock wave dissipation at the exit of the solenoid valve's cut-off surface, and reducing the nozzle thrust performance.

Under an atmospheric pressure, the thrust of the nozzle is not all obtained by the momentum exchange of the gas ejected from the nozzle outlet, some of them are obtained by the pressure difference at the outlet. Different from vacuum environment, the outlet gas velocity is not the higher the better. And the effective specific impulse increases with the enlarge of nozzle inlet pressure. If conditions permit, the inlet pressure of nozzle should be properly increased.

The CFD_DOE method is used to optimize the design of a solenoid valve nozzle, and get the optimal structure of the nozzle. As shown in figure 12. Among them, the nozzle and the solenoid valve
outlet are threaded connection. This part is designed as a structure design of replaceable nozzles. After trial production, the solenoid valve nozzle of this configuration is feasible.

Considering the difficulty and error of the actual machining, the calculated result will be slightly larger than the actual value, but it does not affect the design of the optimal configuration of the nozzle. Our next work is to manufacture the solenoid valve nozzle according to the simulation results, and set up the experimental environment to further verify the thrust and specific impulse of the nozzle.

References

[1] Mueller J 1997 Thruster options for microspacecraft - A review and evaluation of existing hardware and emerging technologies. J. American Institute of Aeronautics and Astronautics Inc. AIAA 3058.
[2] Cai Yukui 2016,24(11) Design, Manufacture process and thrust performance of micro nozzle. D. Shandong University.
[3] Yu Weiling, Feng Yudong, Zhou Hui, Cao Shengzhu and Zhang Xiaoyu 2018,24(04) Fluid simulating analysis and optimization of micro Laval nozzle. J. Vacuum and Cryogenics 246-250.
[4] Wang Dingjun, Song Huiling and Liu Wei 2010,4 Simulation and research of the nitrogen thruster flow field. J. Journal of Rocket Propulsion 36-39.
[5] Ray D. Rhew and Peter A. Parker 2007 A parametric geometry computational fluid dynamics (CFD) study utilizing design of experiments (DOE). Collection of Technical Papers - U.S. Air Force T and E Days 94-106.
[6] Uk-Hee Jung, Joon-Hyung Kim and Jin-Hyuk Kim. 2016,30(3) Optimum design of diffuser in a small high-speed centrifugal fan using CFD & DOE J. Journal of Mechanical Science and Technology 30 1171~1184.
[7] Mun Chol Nam, Ba Dechun, Yue Xiangji and Jin Mingri 2018 Design optimization of hydraulic turbine draft tube based on CFD and DOE method C. International Conference on Power Science and Engineering 136.
[8] Wang Xinyue, etc 2006,5 Fundamentals of aerodynamics M. Xian: Northwest University of Technology Press.
[9] Wang Keyin, Hang Xingxing, Zhang Xiaotao, Liu Yaopeng and Chen Jichao. 2011,9(03) Design and simulation on scaling supersonic nozzles J. Chinese Journal of Construction Machinery, 304-308.
[10] Yang Haiwei, Zhao Yang. 2007(01) Effects of geometry design on micro nozzle’s performance J. Journal of Propulsion Technology 68-72.
[11] Tao Ran, Quan Xiaobo and Xu Jianzhong. 2001(05) Several Questions in Research of Micro Scale Flow J. Journal of Engineering Thermophysics 575-577.
[12] Peng-Fei Hao, Ying-Tao Ding, Zhao-Hui Yao, Feng He and Ke-Qin Zhu. 2005,15(11) Size effect on gas flow in micro nozzles J. Journal of Micromechanics and Microengineering 2069-2073.
[13] Xu Yanzhi, Zhu Huiren and Zheng Jie. 2013,28(08) Effect of Knudsen number on microscale similar flow characteristics J. Journal of Aerospace Power 1752-1758.
[14] Wang Ping, Li Changping and Chen Baisong. 2012,42(04) Laval Nozzle Flow Analysis Based on CFD Computation J. Aeronautical Computing Technique 60-62.
[15] Kamali R, Mousavi SM and Binesh AR. 2015,24(06) Three dimensional CFD investigation of shock train structure in a supersonic nozzle J. ACTA ASTRONAUTICA 116 56-67.