Numerical Simulation on Combustor of Impinging Stream Vortex Engine (ISVE)

Xiaojing Yu 1,*, Yang Liu 2 and Jingqiu Pei 3

1 School of Power and Energy, Northwestern Polytechnical University, Xi’an, China
2 Science and Technology on Combustion, Internal Flow and Thermal-structure Laboratory, Xi’an, Shaanxi, China
3 Zunyi Xinzhou Airport, Zunyi, Guizhou, China

*Corresponding author e-mail: xj_yu@nwpu.edu.cn

Abstract. The impinging stream vortex engine (ISVE), with the advantages of strong thermal protection performance, low cost and simple structure, emerges as the times require. In this paper, the propellant mixing performance, atomization performance, combustion performance and thermal protection performance under different working conditions are calculated and analyzed by numerical simulation for the designed impinging swirl cooling thrust chamber, and the influence of thrust chamber structure on its flow and combustion performance is discussed. The results show that the wall temperature of the thrust chamber is lower than 400K, and the combustion efficiency reaches 93%. The double eddy current structure will strengthen the combustion reaction and make the top temperature of the thrust chamber too high, which will provide theoretical guidance for the next structural design of the thrust chamber.

1. Introduction

Impacting Swirl Thrust Chamber is a two-component thrust chamber structure of liquid rocket engine proposed by AMRDEC[1] in 1990s. The difference between the turbo-cooled engine designed by this technology and the traditional liquid rocket engine is that it does not inject propellant at the head, but injects propellant tangentially into the side wall of the combustion chamber. Under the action of centrifugal force, the unburned propellant will form a protective film on the wall, isolating the gas from the wall of the thrust chamber, thus achieving cooling effect. In order to verify the injection mode of propellant in impingement swirl thruster, Huu Trinh et al.[2] of the United States carried out an experimental study on the injection mode of fuel in impingement swirl thruster in 2003, and proposed two fuel injection schemes. Unlike impinging injector module and chasing injector drawing. Using liquid oxygen kerosene as fuel, the experiment shows that the performance of the injector is excellent, and the wall temperature of the thrust chamber of the engine decreases obviously. The thermal protection effect of the injector is obvious. Although the engine using the chase injector has good performance and its specific impulse efficiency is over 92%, the chase injector has almost no cooling wall effect.

In order to optimize the structure of the impinging eddy current thruster (ISVE), Michael J. Nusca et al. [3] carried out two-dimensional numerical calculation of the impinging eddy current thruster in 2004. The unsteady, multi-component, non-equilibrium and two-phase flow programs were used to simulate the flow field and chemical reactions in the thrust chamber. The pressure distribution and combustion
product distribution of the two-dimensional model of the thrust chamber were obtained. Around the small L/D gel propellant swirl combustion chamber (short and coarse combustion chamber), Michaels and others [4] carried out a variety of injector structure ignition tests, including the three blow and reciprocal tangential jet swirl combustor. The test results show that the thermal protection performance of the impinging vortex cooling structure thrust chamber is good, and the specific impulse efficiency can reach 92%.

From the above research status abroad, we can see that a lot of research has been carried out on the technology of impinging eddy current cooling thrust chamber for liquid dual-component liquid rocket engine. The technology in this field is becoming more and more mature, and has entered the stage of optimization and improvement and field expansion. At present, there is no theoretical and experimental research on this aspect in China, so the technology of impinging eddy current cooling thrust chamber has been developed. The academic research is of great significance.

In this paper, aiming at the basic structure of the designed thrust chamber with impinging eddy current cooling, the mixing, combustion and thermal protection performance of the flow field structure in the thrust chamber are analyzed in detail through numerical simulation and numerical calculation of the flow field structure in the thrust chamber, which provides a basis for improving the design method of the thrust chamber.

2. Computational Configuration and Model

2.1. Thrust Chamber Configuration and Mesh

Based on AMRDEC’s [1] two-component thrust chamber structure (Figure 1), a two-component thrust chamber is designed for liquid oxygen/kerosene propellant. The main geometric parameters are shown in Figure 2 and the basic dimensions of the thrust chamber are given in Table 1.

![Figure 1. Sketch map of specific structure of ISVE](image1)

![Figure 2. Geometric dimensions of ISVE combustor](image2)

| Table 1. Basic Dimension Parameters of Thrust Chamber |
|-----------------------------------------------|
| Name                                      | Symbol | Unit       | parameter values |
|-----------------------------------------------|
| Combustion chamber Diameter                  | $D_c$  | mm         | 56               |
| Dome height                                  | $L_1$  | mm         | 4                |
| Length of straight line Segment              | $L_3$  | mm         | 22               |
| Throat diameter                              | $D_t$  | mm         | 14.5             |
| Shrinkage angle                              | $\theta_c$ | Degree (°) | 100             |
| Impact Point Height                          | $L_2$  | mm         | 14               |
| Oxidizer angle                               | $\theta_o$ | Degree (°) | 22               |
| Fuel angle                                   | $\theta_f$ | Degree (°) | 15               |
In order to ensure the atomization performance of the thruster chamber and obtain the optimum design scheme of the thruster chamber, four groups of nozzle velocities are compared and analyzed from 20m/s to 5m/s interval in order to obtain the appropriate design parameters of the propellant injector. Under different velocity conditions, the average circumferential velocity of synthetic jet can be obtained from the momentum conservation according to the corresponding nozzle aperture and fuel aperture and velocity as shown in the table below.

| Operation and geometry parameters of four different injectors |
|---------------------------------------------------------------|
| Oxidizer speed(m/s)                                           | 20 | 25 | 30 | 35 |
| Oxidant pore size(mm)                                         | 1.34 | 1.20 | 1.09 | 1.01 |
| Fuel speed(m/s)                                               | 20 | 25 | 30 | 35 |
| Fuel aperture(mm)                                             | 0.99 | 0.89 | 0.81 | 0.75 |
| Circumferential synthesis velocity(m/s)                       | 18.7 | 23.5 | 28.15 | 32.84 |

In the process of meshing, due to the limitation of computer performance and the convenience of improving mesh quality, the geometric model is simplified as follows: the structure of injection section of oxidizer and fuel is omitted, and only the interface between injection section and thrust chamber is reserved; the main content of this paper is focused on the interior of thrust chamber, so only the nozzle with shorter expansion section is reserved. Due to the existence of forced vortices with large velocity gradient and pressure gradient in the central region of the thrust chamber, a sufficiently small grid scale is set up in the central region. At the same time, in the collision area of the impinging nozzle, the flow is more complex, so the grid of the area near the nozzle is refined. In order to meet the requirements of wall function, boundary layer grids are used near the wall. Structured grids are used in the computational area, with a number of about 700,000.

2.2. Computing Model
All the numerical simulations in this paper are performed by solving the steady compressible Navier-Stokes equation using the ANSYS Fluent software package. The governing equations of continuity, momentum and energy expressed in differential form are shown as follows. And for a flow in the compressor path, the ideal gas state equation is used. Where $S_M$, $S_E$, $h^*$ are source term of momentum, energy and specific total enthalpy respectively. And the RNG k-ε model is adopted to predict the turbulence of the flow.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0
\]

\[
\frac{\partial (\rho \vec{U})}{\partial t} + \nabla \cdot (\rho \vec{U} \times \vec{U}) = \nabla \left( \rho \vec{U} \cdot \left( \nabla \vec{U} + (\nabla \vec{U})^T \right) \right) + S_M
\]
\[ \frac{\partial (\rho h^*)}{\partial t} - \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U} h^*) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mu \nabla \mathbf{U} + \mu (\nabla \mathbf{U} + \nabla \mathbf{U}^T) - \frac{2}{3} \nabla \cdot \nabla \mathbf{U}) + S \]  

The solution of fully implicit algorithm is employed to calculate continuity, momentum and energy equation simultaneously. And a first-order backward difference scheme in time and second-order central difference scheme in space are employed in discretization.

The chemical kinetic model adopts the commonly used finite rate model, and the combustion process of liquid oxygen kerosene adopts the three-step reaction mechanism [5]. In order to properly simulate the movement and trajectory of liquid oxygen kerosene atomized particles in the flow field, according to the actual calculation structure, the plain-orifice atomizer model is adopted, which can effectively simulate three different working areas of the flat nozzle: single-phase area, cavity area and reflux area. The particle trajectory model is used to track the movement and transport of these discrete atomized particles in the whole flow field by Lagrangian method.

3. Numerical simulation results and analysis

3.1. Analysis of propellant mixing performance

The mixing performance of propellant will have an important impact on the combustion performance of the thrust chamber. Firstly, the mixing performance of propellant in the thrust chamber is analyzed by using the component transport model. The mixing performance of propellant is judged by comparing the oxygen-fuel ratio distribution of propellant at four different speeds. The results of mixing performance calculation under four working conditions are shown in Figure 4.

![Figure 4. The distribution of oxygen-fuel ratio in combustor](image)

Compared with 20m/s, 25m/s and 35m/s, the oxygen-fuel ratio gradient near the wall of the thrust chamber decreases significantly, and it can be concluded that the mixing degree of the thrust chamber propellant is better with the increase of velocity. Under the condition of 20m/s, because of the poor mixing condition of propellant with too low velocity, the oxygen-fuel ratio in the impact zone has a distinct demarcation line; under the condition of 25m/s, the mixing effect is obviously improved, but the overall oxygen-fuel ratio is low; under the condition of 35m/s, the mixing degree is the best, the oxygen-fuel ratio distribution is uniform, but the oxygen-fuel ratio is relatively low, which is not suitable for combustion reaction, and the marginal oxygen-fuel ratio is basically the same as the central region. At the same gradient, it is not conducive to engine cooling. At 30m/s, the oxygen-fuel ratio of the central axis is high and the gradient of the axial distribution is small. At the same time, the oxygen-fuel ratio of the edge is obviously higher than that of the central region, which makes it easier to control the combustion reaction in the central region of the thrust chamber. Therefore, under the same impact angle and flow rate, the propellant mixing performance under 30m/s condition is the best.

In order to further verify the mixing performance of the thrust chamber, the velocity distribution of the longitudinal section of the thrust chamber was analyzed. Because of the complex structure of the swirl flow field in the thrust chamber, in order to simplify the analysis, the axial velocity distribution of the thrust chamber section was analyzed. At the same time, in order to make the analysis more accurate, the fuel mixing and the residence time of propellant in the combustion chamber are deduced mainly through the axial velocity gradient distribution.
Figure 5. Axial velocity gradient distribution in combustor

From the above figure, it can be seen that the equipotential surface of the axial velocity gradient increases gradually from 20 m/s to 30 m/s, that is to say, the range and number of the axial vortices increase. However, under 35 m/s condition, the axial velocity of synthetic jet varies little in the thrust chamber because of its high velocity. This shows that the interior of the thrust chamber is a turbulent flow field with high intensity and strong rotation, which is very suitable for propellant mixing. At the same time, the increase of axial vortices will help to prolong the residence time of propellant in the thrust chamber. Comparing the overall performance of the four working conditions, it is considered that the performance of the 30 m/s working condition is the best.

3.2. Analysis of Atomization Performance

After calculating the mixing performance and liquid film coverage, it is considered that the mixing performance is the best under 30 m/s condition. The discrete term model is used to further calculate and simulate the mixing performance. The physical processes of propellant collision, fragmentation and evaporation are considered in the calculation using a flat-mouth atomization model. Figure 6 shows the particle size distribution of propellant atomization in the thrust chamber calculated by the discrete term model. According to the calculation, the average Sauter mean diameter (SMD) of the propellant droplets in the thrust chamber after impact atomization can be obtained:

\[ D_{32} = 2.283614 \times 10^{-5} m \]

Figure 6. Droplet diameter distribution of propellant in combustor

As can be seen from Figure 6, the concentration area of propellant atomization particles is mainly divided into two parts: one is the center area of the thrust chamber, the other is the annular area close to the wall of the thrust chamber.

From the point of view of the working process of the thrust chamber, propellant impact injects propellant near the wall, and propellant also impacts atomization near the wall, so the atomized particles are concentrated near the wall. At the same time, the particle size distribution near the wall of the distribution chart is obviously larger, which indicates that the larger mass atomized particles are concentrated to the edge under the action of centrifugal force due to the swirling motion of the propellant
atomized particles in the thrust chamber. There are two main reasons for the centralized particles. Firstly, the impact has a larger atomization angle, and some propellants move towards the central region; secondly, the propellant atomized particles move upward and downward after reaching the top, which will form a weak double eddy current flow field structure in the thrust chamber, as shown in Figure 7.

![Figure 7. The streamline of droplets in combustor](image)

The flow field in the thrust chamber is very complex, especially in the vicinity of the impact and the top of the thrust chamber. From the streamline distribution of the whole flow field, it can be seen that the movement of particles is mainly concentrated in the near wall area and the central area, and the movement of particles around the wall area can form a thermal protective layer. It can be seen from the observation of particle streamlines in the central region that the propellant particles have undergone impact and rotation from the injector inlet to the central region of the thrust chamber. The movement direction of the propellant particles has changed many times, which is very beneficial to the mixing performance of the propellant. At the same time, the loss of kinetic energy of the particles is large, the velocity of the central region is small, the residence time of the fuel is increased, and the combustion reaction is facilitated. Conduct. Therefore, the motion of the propellant particles in the central region is helpful to the mixing and combustion of the propellant.

### 3.3. Combustion Performance Analysis

In order to further analyze the performance of the thrust chamber, the combustion efficiency was analyzed by comparing the theoretical calculated characteristic velocity with the numerical simulation value. The theoretical characteristic velocity of propellant is 1375.39m/s under the calculation condition obtained by the thermodynamic software CEA. The characteristic velocity in numerical simulation is obtained by thrust chamber pressure, throat area and propellant flow rate.

\[
c^* = \frac{P_0 A_t}{m} = 1286.05 \text{m/s}
\]

It can be seen that the combustion efficiency of the thrust chamber is about 93.5%, which proves that the combustion performance of the thrust chamber is good.
The temperature distribution of the thrust chamber is a key parameter reflecting the thermal protection performance of the impinging swirl thrust chamber. The temperature distribution of the thrust chamber is shown above after the combustion reaction of the thrust chamber is stable. From the temperature distribution of the thrust chamber, it can be seen that the combustion temperature in the center of the thrust chamber reaches 3400 K, but the wall temperature is only 400K, and the compatibility of the chamber wall is good. Because of its special structure design, the high temperature reaction area of the impinging eddy current cooling thrust chamber is enclosed by the liquid film, unburned propellant and its vapor, thus ensuring that the wall remains in a low temperature state. It can be seen that the high temperature gas in this design scheme is covered by unburned propellant except for a small part of the dome of the thrust chamber. The wall temperature of the thrust chamber is below 400K, and the cooling effect is excellent. The core area of high temperature gas presents olive shape, which occupies the largest area at the junction of straight cylinder section and convergence section, and gradually reduces along the axis, but at the top of the thrust chamber, the high temperature area increases gradually.

It can be concluded that the structure of the high temperature region is mainly caused by two reasons: (1) the height of the central part of the high temperature reaction region is basically the same as the position of the injector; the droplet particles of the impinging nozzle colliding with the wall enter the central part of the thrust chamber mainly from the region, and mix with each other under the action of swirling flow, so that the high temperature region occupies the largest volume ratio in the central part of the thrust chamber. (2) A small amount of propellant moves upward to form a double eddy current flow field structure. At the same time, the interaction between the reverse rotating propellant and the forward rotating propellant produces viscous shear force and local pressure fluctuation, which further strengthens the mixing of propellant, and makes the mixing degree in this region higher, the relative velocity lower, and suitable for combustion reaction. Therefore, the high temperature area at the top of the propellant is enlarged.

**Figure 8.** The temperature distribution and streamline in combustor

**Figure 9.** The temperature and density distribution in the injection impinging cross-section
The temperature distribution of the cross section of the impact nozzle is analyzed separately (Figure 9). It can be seen that the high temperature zone is completely covered, and the 400 K cryogenic cooling layer occupies 50% of the space of the thrust chamber. The cooling effect is also good in the convergence zone and the straight cylinder section, and the high temperature zone over 3000K is enlarged. Comparing the two temperature distributions, it can be concluded that the wall-attached propellant gradually participates in the reaction from top to bottom, and the liquid film gradually evaporates and breaks up with the increase of cockroach moving distance, so the high temperature region gradually expands from top to bottom. From the density distribution of the impact surface, it can be seen that the density of the box position is much higher than that of the surrounding area, which is the impact position of the impinging nozzle. Due to the effect of impact atomization and high temperature evaporation, some propellants changed from liquid phase to gas phase after impact, and their density changed sharply, but remained stable in the subsequent flow. Because the impact synthesis angle is downward, there is no liquid film attached to the wall along the flow direction in the cross-section distribution; and from the density distribution of the transition surface, it can be seen that there is almost no liquid film on the wall of the region due to the evaporation of the liquid film attached to the wall, but the high temperature region is wrapped by the propellant film to achieve cooling effect.

In this paper, the performance and temperature distribution of traditional liquid rocket engine [6, 7] under the same working conditions are compared and analyzed. The combustion efficiency is 94.2%, which is very close to 93.4% of the combustion efficiency of the impinging swirl thrust chamber in this paper. However, the wall temperature of the thrust chamber is more than 2000K, which will bear a large thermal load, and the shell material and cooling measures will be put forward higher requirements. The wall temperature of the thrust chamber cooled by the junction impingement eddy current is basically lower than 400K, and the heat flux density of the shell is much lower than that of the traditional liquid rocket engine. The shell has the ability to work for a long time with cheap materials, which can greatly reduce the material and the cost of manufacture.

4. Conclusion
(1) The propellant mixing and atomization performances under four kinds of injection speed conditions were analyzed and calculated. Comparing the calculated results, it is considered that the comprehensive performance of mixing and flow in thrust chamber is the best when the oxidant injection speed is 30 m/s. The average Sauter mean diameter value is $2.283614 \times 10^{-5} m$, and the atomization effect is good, which is suitable for combustion reaction.

(2) The distribution of the main parameters of the thrust chamber is obtained by numerical analysis. The wall temperature of the thrust chamber is below 400K, and the combustion efficiency of the thrust chamber reaches 93%. Compared with the traditional liquid film cooling thrust chamber of the same level, the combustion efficiency is basically the same, and the thermal protection performance has obvious advantages.

References
[1] Liang, R, and Ungewriter, R, Multi-phase simulations of coaxial injector combustion[R]. AIAA 1992-345.
[2] Huu Trinh, Btad Bullard, Scott Michaels, Evaluation of Impinging Stream Vortex Chamber Concepts for Liquid Rocket Engine[R]. AIAA 2003-4476.
[3] Michael J. Nuscal, R. Scott Michaels, Program in the Development of a Computational Model for the Army’s Impinging Stream Vortex Engine [R]. AIAA 2004-3851.
[4] Michael J. Nuscal, and Michael J. McQuaid, Combustion Chamber Fluid Dynamics and Hypergolic Gel Propellant Chemistry Simulations for Selectable Thrust Rocket Engine[R]. IEEE0-7695-2259-9/04.
[5] Huang Y. H., Theoretical, Numerical Simulation and Experimental Investigations of Combustion Instability in Liquid Rockrt Engine[D]. National University of Defense Technology, 2001.
[6] Lin Q. G., Zhou J., A Numerical Study of Combustion and Heat Transfer in Small Thrust Liquid Rocket Engine[J]. Journal of National University of Defense Technology, 2012, 34(4):13-17.

[7] Lin S., Zhou J., Liu C. G., Simulation of Operation Process of Small Thrust Chamber in High performance Liquid Engine[J]. Shanghai Institute of Space Propulsion, 2007, 6.