Study on the Effect of Water Injection Momentum on the Cooling Effect of Rocket Engine Exhaust Plume

Kan Yang 1,*, Yanhui Qiang 1, Chenghang Zhong 2, Shaozhen Yu 1

1 Naval Equipment Research Institute, Beijing, China
2 Office of Military Delegates from Navy in 8359 Institute of CASIC, Beijing, China

*Corresponding author e-mail: yk258000381@126.com

Abstract. For the study of water injection momentum factors impact on flow field of the rocket engine tail flame, the numerical computation model of gas-liquid two phase flow in the coupling of high temperature and high speed gas flow and low temperature liquid water is established. The accuracy and reliability of the numerical model are verified by experiments. Based on the numerical model, the relationship between the flow rate and the cooling effect is analyzed by changing the water injection momentum of the water spray pipes. And the effective mathematical expression is obtained. What’s more, by changing the number of the water spray and using small flow water injection, the cooling effect is analyzed to check the application range of the mathematical expressions. The results show that: the impact and erosion of the gas flow field could be reduced greatly by water injection, and there are two parts in the gas flow field, which are the slow cooling area and the fast cooling area. In the fast cooling area, the influence of the water flow momentum and nozzle quantity on the cooling effect can be expressed by mathematical functions without causing bifurcation flow for the mainstream gas. The conclusion provides a theoretical reference for the engineering application.

1. Introduction

In recent years, with the extensive application of large thrust carrier rocket, the use of liquid water vaporization heat absorption principle to reduce the rocket engine tail flame high temperature airflow impact and ablation damage has also been drawn more and more attention, and has achieved the engineering application [1].

To the rocket engine tail flame water injection, most of the current application is to suppress the large thrust carrier rocket in the launch pad ignition off the noise generated in order to avoid the rocket body itself or the load carried being damaged. In this field of research, domestic and foreign relevant scholars and engineering and technical personnel have organized a number of experiments, established the relevant numerical calculation model and in-depth study [2-4], and have acquired a large number of research results that have been applied in space launch. For the rocket ignition process of the impact and ablation, with the large thrust carrier rocket ignition tail after the increase in the radial and length profile, the damage to the ground equipment is more serious, so the basis of water injection noise the cooling effect of tail flame is becoming more and more obvious, and the research on the mechanism of water cooling is becoming more and more further.
The two-phase coupling mechanism between liquid flow and tail flame flow is complex and has many influencing factors. Such as water injection pipe hole shape, gas-liquid two-phase relative position, water velocity and droplet diameter, temperature and water injection angle. In addition, the experimental study on the water injection and cooling effect is carried out by M. J. Miller and the equation between the water injection volume and the cooling effect is obtained according to the collected experimental data. In reference [6], numerical simulation is carried out to simulate the water injection in the spaceflight field. In recent years, domestic research has made important progress in this field. Reference [7] proved that the water injection can effectively achieve the cooling of the gas flow field. In reference [8], the method of experiment and numerical calculation is used to study the two-phase flow field which is cooled by columnar water flow. It is concluded that the effect of water injection on reducing the impact and ablation of the tail rock of the rocket engine is significant, and the accuracy and reliability of the numerical calculation model are verified. In reference [9], the factors such as water injection velocity, liquid water temperature, and water injection pressure and droplet diameter are studied deeply, and the effective droplet diameter can be effectively cooled. The multi-stage water injection pipe used to carry out the gas flow field cooling effect is more effective.

In this paper, the cooling effect on the flow field of the columnar water flow is analyzed. The numerical results are verified by the scaling test. On this basis, the fitting data equation obtained by the experimental method in reference [5] is verified and analyzed influence of water injection location on flow field profile and cooling effect. And it can provide reference for optimal design of water injection cooling in space launch site.

2. Sub-scale model test and numerical calculation model

2.1. Water injection sub-scale model test

The water injection system designed with the launch space of the large launch vehicle should not only achieve the positive cooling effect but also be strictly limited to the noise intensity standard, so the reference gas flow noise reduction test conclusion is very practical significance for the practical application of the water injection cooling system. According to the reference [10] through the analysis of the effect of water spray noise analysis to the relative 60 ° angle of the implementation of water injection will be the best effect of jet noise suppression, drawing on the above conclusions, in the water injection test study, the same 60 ° angle water injection method. Figure 1 (a) shows the use of vertical test bed to eliminate the impact of gravity caused by the error, the height of the test bench to the engine after the exit plane as the origin, from the bottom of the impact platform height of 1.8m. Figure 1 (b) shows the use of double injection pipe water flow has a good symmetry, after the impact of water flow was water mist. It shows that the water injection of the double water pipe can cool the same position of the gas flow field, and will not tilt the tail flame flow field due to the impact of water flow.

![Fig.1 Diagram of water injection with two pipes](image_url)

The test engine and charge are shown in Figure 2 (a). Test using the engine compression ratio, according to the reference [11] scaled principles, design and dimensions of the combustion chamber of
the engine Laval nozzle profile, the outlet velocity, temperature and the outlet pressure reaches the design requirements. The theoretical design value of the combustion chamber pressure curve of the model engine is 7 106 Pa and the Mach number of the outlet gas flow rate is 3.3. The measured engine thrust curve is shown in Fig. 2 (b). The engine outlet gas flow is 1.5kg / s.

![The engine model and propellant](image1)

(a) The engine model and propellant

![Pressure curve in the combustor](image2)

(b) Pressure curve in the combustor

**Fig.2** The engine model and pressure curve

### 2.2. Numerical calculation model

The numerical calculation field is shown in Fig. 3, and the end of the combustion chamber (ie, the nozzle inlet) is set as the gas inlet, and the nozzle flow field and the external flow field are integrated. According to the symmetry using the 1/4 model. The boundary condition is set as: the engine nozzle is set as the pressure inlet, the inlet pressure curve input is the discrete curve of the measured curve, and the difference is calculated at each time pressure value. The bottom impact platform is set to solid wall. According to the test procedure, the water injection pipe is reserved in the model and set as the solid wall. The gas flow outside the flow field area is set to quiescent atmospheric conditions. The computational grid adopts structured hexahedral mesh, and the tail flame flow field is processed by the area and the water pipe wall, and the number of grid is 1.1 million.

According to the reference [12] numerical calculation method, the finite volume method and the turbulence calculation method are used to calculate the RNG $k-\varepsilon$ model. The near wall turbulence is calculated using the standard wall function. The Coupled algorithm based on pressure is used in the flow field calculation, and the coupling process of liquid water and gas flow is calculated by unsteady
algorithm. The Mixture multiphase flow model in the Euler-Euler model is used to calculate the component diffusion and transport between the gas-phase gas component and the liquid water vaporization generated by the component transport model. The vaporization model is coupled to the flow field calculation in the process of vaporizing the phase transition process and energy transfer by adding the source term. The source is mainly involved in the vaporization of liquid water and the condensation of water vapor and the interphase energy transfer during vaporization.

3. Model validation

3.1. Free flow field calibration

![Fig.4 The free jet flow](image)

Fig.4 shows the numerical results and the experimental results of the free-flow tail flame flow field. From Fig. 4 (a), the numerical results show that the wave structure and temperature distribution of the tail flame flow field are in good agreement with the experimental results. Figure 4 (b) is the first half of the wave structure of the distribution map, the figure can be clearly seen in the numerical results of the core area of the five high-temperature regions from the distribution position and shape consistent with the test results.

![Fig.5 Contours of Ma and static pressure](image)
Figure 5 shows the flow field Mach number distribution and pressure distribution cloud, from the figure can be seen free jet exit Mach number of 3.4 or so, in line with theoretical design values, and in the mainstream of the gas distribution and pressure distribution in line with high-speed gas jet feature. Therefore, in summary, the numerical calculation of high accuracy, through theoretical analysis of in-depth study.

3.2. Water flow correction of water injection

Figure 6 shows the high-speed photographic images of the tail-flow field after water injection. The water flow parameters are as follows: the flow rate of the outlet water is 16 m/s and the flow rate of the single water pipe is 1.2 kg/s. In the area after the impact of water flow, the shape and temperature distribution of the tail flame flow are consistent with the experimental results. In the second half of the mainstream area, due to a large number of water vapor generated cover the core area, so in the high-speed photography results in the second half of the flow field cannot be clearly displayed in the picture out, but the high-temperature area contrast image can also be indicating that the temperature of the second half of the flow field is low, there is no high temperature area. It is consistent with the simulation results.

4. Result analysis

4.1. Analysis of Flow Field Parameters after Water Injection
Figure 7 shows the distribution of the temperature and velocity of the flame at both the water injection and the non-water injection. Figure 7 (a) shows forming the spindle structure in the high temperature region of the flow field after injection. From the water injection position, the wave structure of the core region of the tail flame is destroyed, and as the development temperature of the flow gradually decreases, it can be seen that the length of the high temperature region is about 1/2 of the flow field of the non-water injection tail, and there is no obvious high temperature region. Cooling effect and influencing flow field profile. Figure 7 (b) and Figure 7 (c) for the two flow field temperature and velocity comparison, it can be drawn from the figure, after the high temperature flow field and the free jet flow field at the same location temperature and velocity distribution consistent. It is shown that the energy transfer between the gas and liquid phases is not transmitted in the radial direction in the cross section of the flow field, that is, the heat transfer rate in the gas flow field is much lower than the phase heat transfer rate.

Figure 8 shows the temperature and velocity curve on the flow field axis. From the figure, the temperature change effect of the flow field can be obtained intuitively. The temperature change on the
axis of Fig. 8 (a) is slightly decreased from the intersection point, which can be called the slow cooling section. After 0.8m, the mainstream temperature drastically decreases. From 1.4m, the temperature of the gas stream is basically gentle, reaching a minimum value of about 550K, which can be regarded as a rapid decline in temperature. The same trend also appears in Figure 8 (b) for velocity changes. The rapid cooling and cooling area in the flow field is between 0.8m and 1.4m, which indicates that the water injection has obvious effect on the cooling and deceleration of the second half of the main stream.

4.2. Analysis of Factors Affecting Water Cooling

4.2.1. Analysis of Flow Field Calculation Results.

Table 1. Operating conditions of water injection

| Parameter                  | Case 1 | Case 2 | Case 3 | Case 4 | Case 5 |
|----------------------------|--------|--------|--------|--------|--------|
| Gas momentum/(kg·(m/s²))   | 3966   | 3966   | 3966   | 3966   | 3966   |
| Flow momentum/(kg·(m/s²))  | 180    | 102.3  | 90     | 72     | 45     |
| Momentum ratio M           | 22     | 39     | 44     | 55     | 88     |

Reference [5] through the test data collection analysis, obtained after the water flow cooling effect with the water flow associated with the amount. Therefore, the numerical calculation of the establishment of Table 1 in the five calculation conditions, and the temperature on the axis of the analysis. The gas momentum in Table 1 is based on the fuel flow parameters at the exit of the engine nozzle. The use of double injection pipe by changing the flow rate to change the water injection momentum. Momentum ratio M is the ratio of gas flow to water injection.
Figure 9 shows the flow chart of the flow field in different calculation conditions. The graph shows that the plane is perpendicular to the plane of the water injection pipe. It can be concluded that the main gas flow is affected by the flow in the working conditions 1 and 2, and there has been the phenomenon of bifurcation flow. In case 2 the mainstream appears bifurcation flow and then polymerized into a share. In case 3, case 4 and case 5 the flow rate of the water flow is further reduced, the impact on the mainstream is weakened, the main flow is not in the bifurcation flow phenomenon, the post-flow field temperature distribution structure is basically the same, and the temperature distribution is also similar.
Figure 10 (a) shows the temperature profile of the axis 1-3, and Figure 10 (b) shows the temperature profile of the case 3-5. From the curve of the temperature curve, it can be seen that the overall trend of temperature change is similar in several water injection conditions on the axis. After the water cooling effect on the axis of the two changes in the stage. That is, the rapid decline in temperature and temperature stability section. In the next stage of rapid temperature, from the point of view of the temperature change point of each calculation condition, the position of temperature mutation gradually shifted with the decrease of water injection momentum, and the temperature of the axis gradually increased. As the working conditions of the mainstream by the impact of separation into two shares, the temperature on the axis relative to the other four conditions, the difference is obvious. The condition 2 ~ 5 remained stable at the position of 1.1 m, the temperature changed diametrically opposite, the temperature dropped to around 450, and the change was slow, indicating that the heat transfer and mass transfer between the two phases did not work after this position. Obvious. The temperature of the flow field is increasing after 1.1 m, and the temperature of the curve is in the range of 0.3m ~ 0.4m. In the case of 1 ~ 3, the temperature jumping point appears. Intersecting intersection, the impact between the two phases in this position to form a stagnation, the speed caused by lower temperature rise. The peak values in the working conditions 4 and 5 are not obvious, indicating that the effect of water flow on the mainstream of the gas is not obvious for the core area.
Fig. 11 The velocity curves on the axis

Figure 11 shows that the trend of the axial velocity is valid for the temperature change of the above diagram. From the analysis of the velocity curve, the effect of the water spray on the gas flow field is quite obvious, and the purpose of reducing the temperature of the gas flow can be achieved. It can effectively reduce the impact of gas flow on the chute and ablation.

4.2.2. Influences of Water Injection on Cooling Effect. For the relationship between the flow rate and the cooling effect of the flow field, reference is made to Miller et al. [5], which concludes that the dimensionless relationship between the heat flux density and the two-phase momentum ratio $M$ in the test conditions is:

$$ Q_r = ae^{bM} $$

(1)

Where: $a = 0.850$, $b = -0.00116$ is the percentage decrease in the measured heat flux density.

According to the heat flow meter measurement principle and the Fourier law of heat conduction, the heat flux density expression is:

$$ q^* = \frac{Q}{F} = -\lambda \frac{\Delta T}{\Delta X} $$

(2)

Where: $Q$ is the unit time through the area $F$ of heat; $\Delta T$ and $\Delta X$ were measured on both sides of the sensor temperature difference and the sensor length. In the test $\Delta X$ is considered as a constant, because the sensor itself using thermal insulation material to maintain the accuracy of heat flux.
measurement, so $\Delta T$ actually reflect the main point of the measured temperature changes, that is, heat flux density $q^*$ is proportional to the measured temperature $T$.

Because the engine parameters and water spray parameters used in the experiment are different from those of the Miller test, this paper analyzes the calculation conditions by reverse verification method for the above dimensionless formula (1), if there is a relationship with the equation function, it is necessary that the temperature value of a certain point on the axis conforms to the above relation. If the point exists, it proves that the above equation has certain applicability, and also proves that the calculation result has certain reliability.

Analysis of the calculation of working conditions 1 to 5, because the value of the impact of $M$ is the numerical value of the size of the index $e^{bM}$, so the case of 1 ~ 5 axis temperature according to the value of appropriate amplification or reduction of a certain proportion, there must be a data intersection. The specific conversion equation is as follows:

Substitute formula (1) into the axis temperature to obtain the corresponding conditions $Q_{ne}$, and take the condition 5 as the reference parameter, then the following equation is used to calculate the ratio between the temperature values in different working conditions, that is:

$$R_i = (1 - Q_{ne}) / (1 - Q_{ref}) \quad (i = 1, 2, 3, 4)$$

(3)

The corresponding conditions on the axis of the temperature amplification of $R_i$ times the curve shown in Figure 12.

![Adjusted temperature curves of the axis](image)

**Fig.12** Adjusted temperature curves of the axis

It can be seen from Fig. 12 that there are common points in the vicinity of the 1.3m position except for the working condition 2, and the value of the working condition 2 is close to the common value, so it can be explained that in the analysis of the water flow rate (1), and then can be applied to (1) on the axis of the other location points to establish the corresponding parameter equation.

According to the temperature change of the axis in the working condition shown in Fig. 10, the temperature of the axis begins to decrease obviously after the 0.9m, and then the temperature point data of 0.9m are verified by the above equation. It is assumed that the temperature change after the position is in accordance with the equation (1), which is expressed as follows:

$$\ln Q_r = \ln ae^{bM} = \ln a + bM$$

(4)
For a certain point, the gas flow and water flow ratio $M$ as an independent variable, the temperature change value for the dependent variable, the data connection between the different conditions should be a straight line, the slope is $b$.

![Graph showing temperature change](image)

**Fig.13 Relations of temperature on different points**

Figure 13 is the logarithmic relationship between the equation 4 after the establishment of the curve, it should be noted that in the case of 1 gas flow was shocked into two shares, so the axis temperature changes compared to the latter four kinds of calculation conditions larger, So the curve in all the curves in the working conditions 1 to 2, the slope of the line with the main line slope of the difference between the larger, from the other four conditions, in addition to 1.6m position point of work-5 abnormalities, the other locations (1) The equation of function (1) can be used to determine the relationship between the temperature change and the relative momentum ratio on the axis. As can be seen from Fig. 12, the temperature change parameter $a$ and $b$ at different positions of the axis is different because of the difference in the slope, and the relevant numerical solution needs to be determined. In addition, the application condition of the formula (1) is limited to the value of the momentum ratio $M$ greater than the condition 2 in the double water pipe water injection condition.

4.2.3. *The effect of the number of nozzles on the cooling effect.*

| Parameter                              | Case6 | Case7 | Case8 |
|----------------------------------------|-------|-------|-------|
| Gas momentum/(kg*(m/s)²)               | 3966  | 3966  | 3966  |
| Single nozzle flow mass flow rate/(kg/s)| 3.6   | 3.6   | 3.6   |
| Mean flow velocity /(m/s)              | 16    | 16    | 16    |
| Pipe quantity                          | 2     | 3     | 4     |
| Injection momentum /(kg*m/s²)          | 115.2 | 172.8 | 230.4 |
| Momentum ratio $M$                     | 34    | 23    | 17    |

In order to verify the relationship between the cooling effect on the axis and the water injection momentum under the condition of the number of different water pipes and the same single nozzle water injection parameters, the calculation condition is established as shown in Table 2.
Figure 14 is the temperature and velocity curve of the flow field after the injection of water from 6 to 8, and the temperature of the axis increases with the increase of the number of water pipes in figure 14 (a), which is inversely proportional to the water injection momentum. With the increase in the number of water pipes, the gas flow is affected by the impact of water flow, by the side of the double water pressure side of the expansion side of the expansion of the deformation of a 3 nozzle or 4 nozzle under the state of uniform compression deformation, water flow into the mainstream gas. The depth of the reduction, resulting in the above results, which also shows that the number of water pipes is also one of the factors affecting the cooling of the gas flow. Figure 14 (b) for the flow field axis velocity curve, from the 0.8m position began to rapid decline in the axis speed, the flow of the block effect is significant, the figure 4 due to the large amount of water flow, resulting in the axial flow rate slightly lower than the case 3.

Fig.15 Relations of temperature on different points

Formula (4) transfers the temperature change of the axis to verify whether the calculated condition conforms to the function relation obtained by the formula (1), as shown in Fig. Through the curve slope change can be seen in the axial temperature changes 0.9m after the three conditions of
temperature changes in accordance with the formula (1), from Figure 13 axis temperature curve shows 0.9m after the rapid decline in the temperature of the axis, the main heat loss main occurrence area. After 0.9 m for the relevant position, the value a and b can be obtained through the curve relationship.

4.2.4. Effect of small flow water injection on cooling effect.

Table 3 Operating conditions of water injection

| Flow parameter                          | Case 9 | Case 10 | Case 11 |
|----------------------------------------|--------|---------|---------|
| Gas momentum/(kg•(m/s2))               | 3966   | 3966    | 3966    |
| Single nozzle flow mass flow rate/(kg/s)| 0.9    | 0.6     | 0.45    |
| Mean flow velocity /(m/s)              | 4      | 2.67    | 2       |
| Pipe quantity                          | 4      | 6       | 8       |
| Injection momentum /(kg•m/s2)          | 14.4   | 8       | 7.2     |
| Momentum ratio M                       | 275    | 496     | 550     |

Water consumption is one of the main optimization design factors of water injection system. Reducing the amount of water will inevitably weaken the effect of water cooling. However, reducing the velocity of water injection can extend the action time after two-phase contact and increase the energy transfer. In order to test the effect of small flow water injection cooling, the total water injection rate is 3.6 kg/s, which is 2.4 times of the engine gas flow rate, increases the number of nozzles and changes the water flow and water velocity of each water spray pipe accordingly. The calculation conditions are shown in Table 3.

![Fig.16](image1)

**Fig.16** The curves of flow parameters on the axis

Figure 16 shows the temperature and velocity variations on the flow field axis. Figure 16 (a) The temperature of the axis decreases significantly after 0.3 m at the intersection, which is about 100 K
lower than the temperature at the same position as in Figure 14 (a), and after the rapid descent of the temperature starts at 0.9 m, it is delayed by 0.1 m. In the temperature drop trend, the impact surface temperature above 600K. The results show that the flow of water flow and the decrease of flow rate lead to the relative growth of the two-phase flow at the same position, and the heat transfer between the two phase's increases and the cooling effect is enhanced in the first half of the flow field. The same analysis of the speed curve Figure 16 (b), with the increase in the number of water pipes, the main effect of the block for the more obvious gas. However, compared with Figure 14 (b) using a small flow of water, the flow field of the block at the same location greatly weakened. And the dominant position of the dominant velocity moves more forward.

The above shows that the use of small flow of water can achieve the cooling of the end of the flow field, but the water injection should be optimized to meet the thermal protection needs. In addition, as shown in Fig. 16 (a), the water injection of the Nozzle and the water injection in the 6 nozzles are crossed at the rapid descending section of the axial temperature and do not conform to the equation (1). It is shown that the formula (1) does not apply to the calculation of low flow water injection cooling.

5. Conclusion
The effective cooling and deceleration of the rocket engine wake can be realized by means of water injection, and the profile of the tail flame flow field can be reduced to realize the thermal protection of the ground diversion tank and the auxiliary launching device.

Liquid water in the rocket engine tail flame flow field, in the two-phase intersection after the existence of the main flame temperature and flow rate of slow decline and rapid decline in the area, the use of large water injection and water flow under the premise of the mainstream temperature eventually reached stable state, the use of small flow of water state, the main gas temperature drop, but cannot reach the final stable state.

The ratio of the mainstream temperature of the tail flame to the ratio of the gas flow and the flow rate of the injected water can be established. The calculated results showed that by injecting the water in the double water pipe and the two-phase flow field does not destroy the main flow, the expression is valid, and the number of nozzles can be increased. Similarly, the constant value in the expression should be determined based on the calculated result or the test result.

In the case of slow flow rate, the temperature of the axis decreases from the two-phase contact position, but in the rapid cooling section, the temperature is lower than the high-flow water injection cooling effect. It can be optimized design in the actual engineering applications.

References
[1] Liu Han. The LM-7 passed water injection test[J]. Aerospace China. 2015, 04: 13.
[2] Ignatius J K, Sathiayaveeswaran S, Chakravarthy S R. Hot-flow simulation of aeroacoustics and suppression by water Injection during rocket liftoff [J]. AIAA Journal, 2014, 53(1): 235-245.
[3] Vu B T, Bachchany N, Peromianz O, et al. Multiphase modeling of water injection on flame deflector[C] 21st AIAA Computational Fluid Dynamics Conference. San Diego, USA, Jun 24-27, 2013.
[4] Chen Yu, Zhou Xu, Tong Li, et al. Estimation of rocket noise reduction by water injection[J]. Environmental Engineering, 2014, 30:172-175.
[5] M. J. Miller, J. H. Koo, F. M. Sickler ,et al. Effect of water to ablative performance under solid rocket exhaust environment [C]. 29th Joint Propulsion Conference and Exhibit. American Institute of Aeronautics and Astronautics. 1993.
[6] G. Pierre, F. Philippe, G. Laurent. Simulation of water injection into a rocket motor plume [M]. 35th Joint Propulsion Conference and Exhibit. American Institute of Aeronautics and Astronautics. 1999.
[7] Jiang Yi, Zhou Fan, Zhang Xue-wen. Experimental study on flow field of high temperature supersonic impinging jet injected by water[J]. Journal of Experiments in Fluid Mechanics,
2011, 25(4):32-36.

[8] J Li, Y Jiang, et al. Cooling effect of water injection on a high-temperature supersonic jet[J]. 2015, 8 (11):13194-13210.

[9] M Rajagopal, D Rajamanohar. Modeling of an exhaust gas cooler in a high-altitude test facility of large-area ratio rocket engines[J]. Journal of Aerospace Engineering. 2015, 28(1):04014049.

[10] V Bakulev. Features of experiment research of supersonic jet noise reduction[C]. International conference on mechanics-seventh polyakhovs reading. RUS. Feb 2-6, 2015:1-3.

[11] Chen Jing-song, Ma Hong-ya, Lin Yu. Rocket-launched gas jet shrinkage test similarity parameter[J]. Acta Aerodynamica Sinica, 2005, (03):307 - 311.

[12] S Jai, A Vineet, H Ashvin, et al. Analysis of Flame Deflector Spray Nozzles in Rocket Engine Test Stands [M]. 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. American Institute of Aeronautics and Astronautics. 2010.