33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

ICAS 2022 SWEDEN

TWO-PHASE NUMERICAL SIMULATION INVESTIGATION ON THE EFFECTS OF TURBULENCE MODEL AND OPERATING CONDITIONS ON THE CENTRIFUGAL NOZZLE PERFROMANCE

Da Mo[1], Hongjun Lin[1], Caiyun Liang[1], Shoutang Shang[1], Jiaohui Xia[2], Yixiong Liu[1,*]

[1,*]AECC Shenyang Engine Research Institute, Shenyang 110015, China

[2]Aero Engine Academy of China, Beijing 10130, China

Abstract

As one of the critical components of the combustor, fuel nozzle plays a crucial role in deciding the engine performance. One commonly accepted type of fuel nozzle in the aviation industry is the centrifugal nozzle. The air-liquid numerical simulation was performed to explore the atomization mechanism of the centrifugal nozzle. The influences of the turbulence model under different operating conditions were investigated, concerning the renormalization-group (RNG) k-ε model, realizable (RLZ) k-ε model and standard (STD) k-ε model. The air-liquid volume fraction (ALVF), velocity vector and spray cone angle were thoroughly analyzed of the primary and the pilot nozzle. There were no chemical reactions and physical phase transition and the heat exchange between the two phase was not considered. Results show that the RNG, RLZ and STD model were able to capture the air-liquid interface since the primary nozzle deployed a large outlet diameter. Compared to the test, the RLZ model provided the most accurate results at different operating conditions. The RNG model generates a larger cone angle while the STD model was not precise at low fuel supply pressure conditions. A high injection pressure would lead to rapid and improved atomization. The velocity increased from 34m/s to 54m/s when the fuel supply pressure went up from 0.95MPa to 2.6MPa for the pilot nozzle. The two-phase simulation investigation could provide deep insight into the mechanism of air-liquid interaction.

Keywords: turbulence model, centrifugal nozzle, fuel supply pressure, spray cone angle

1. Introduction

Nowadays, the triple-swirler dome has been widely accepted in the high-temperature rise combustor. Compared to the conventional combustor, its wide range of fuel-air ratio (FAR) [1] and fuel flow rate control have made the combustor more stable and efficient. Furthermore, flameout would not occur despite lean burn [2] and better ignition capability could be achieved. Combustion efficiency, soot and emission generation, gas residence duration, and outlet temperature distribution quality [3] are all significantly affected by the fuel nozzle. More importantly, the volume fraction distribution of the fuel is quite influential in combustion efficiency, flame stability, ignition, outlet temperature distribution, and even exhaust pollutants[4]. Pilot fuel injection using a high-pressure fuel nozzle is commonly seen in the gas turbine engine.

Many researchers have explored the design of the pressure-swirl nozzle and the corresponding spray characteristics. Shin [5] carried out a simulation regarding the fuel spray characteristics and investigated the effects of the physical properties. It was found that the increase in density, viscosity and surface tension would generate significant impacts on atomization performance and would slow down the atomization process. A diesel nozzle with variable injection orifice settings was developed by Shatrov [6]. The asymmetrical arrangement of the injector was found to lower the duration and boost the diesel engine efficiency. The Large-Eddy simulation approach was employed by Kaario[7] to simulate the influence of gasoline on high-velocity evaporating fuel sprays. A modest correlation was found between liquid density and liquid penetration.

A numerical approach for modelling the pilot injection procedure was proposed by Gavaises [8]. The flow development surrounding the orifices dwelt time and the subsequent start of the injection cycle were modelled and validated by experiment. The injection strategies have also witnessed remarkable progress with the development of technology. The injection pressure has been risen to 3000bars [9]and regulating the injection-rate shaping and timing the injection [10] has become a common measure. Multiple injection [11][12], modified orifice geometry and orientation [13] are also investigated. On the basis of the inviscid hypothesis, Taylor [14] found that the spray cone angle is exclusively governed by the swirler chamber size. It was Rizk [15][16] who analyzed a vast number of experimental results and proposed the empirical spray cone angle equation and the flow coefficient equation. According to Simmons [17], pressure nozzle droplet dispersion is greatly influenced by two major parameters including the average diameter and the distribution index.

The air-liquid numerical simulation was carried out to have a better understanding of the atomization mechanism of the centrifugal nozzle. The influences of the turbulence model under different operating conditions were studied, concerning the RNG k-ε model, RLZ k-ε model and STD k-ε model. The ALVF, velocity vector and spray cone angle were thoroughly analyzed of the primary and the pilot nozzle. The two-phase simulation investigation could provide deep insight into the mechanism of air-liquid interaction.

2. Methodology

2.1 Simulation Model

The simulation model of the pilot and primary nozzle included the swirler chamber, orifice, as well as the extended outlet region. The grid mesh of the pressure fuel nozzle is presented in Fig.1 with the total mesh nodes reaching about 3 million. The grid independence verification was performed, which showed that the 3 million grids was capable of precisely predict the spray cone angle. The mesh of the primary fuel nozzle is similar to the pilot nozzle and would not be illustrated in the paper.

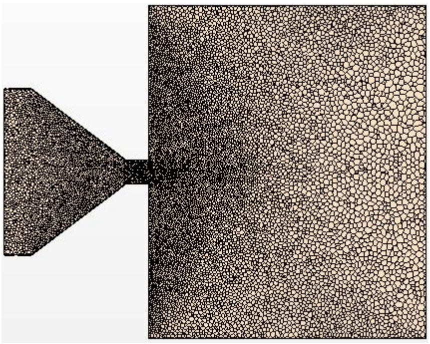


Figure 1 –Mesh of the pilot nozzle.

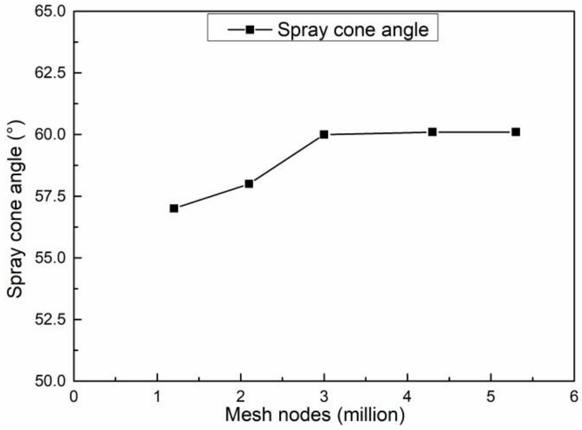


Figure 2 –Grid independence verification.

2.2 Turbulence Model

Three models were investigated in the paper including STD, RNG and RLZ k-ε models[18-19]. The RNG model is similar to STD model but incorporates the influences of vortex and provides the analytical formula for turbulence Prandtl number. It also includes some terms in the k and ε formula. As for RLZ model, it adopts the rotation and curvature terms.

2.3 Boundary Conditions

As for the boundary conditions of the centrifugal nozzle, the outlet was set as the pressure outlet. The fuel supply pressure for the primary nozzle was 0.2 MPa and 0.9 MPa while 0.95 MPa and 2.6 MPa for the pilot nozzle. The first phase was incompressible air while the second phase was Jet-A fuel.

3. Simulation Results

3.1 Primary fuel nozzle

The atomization performance at different fuel supply pressure and using the three turbulence models were investigated. The ALVF and velocity vectors are shown in Fig.3 to Fig.14. It should be noted that 0 and 1 of the ALVF mean the air phase and liquid phase separately.

The fuel is discharged from the nozzle as a thin conical sheet and a hollow cone film would be generated after the fuel passes through the injector. The film thickness at the discharge orifice is a good indicator for assessing the nozzle atomization performance. Specifically, the liquid volume fraction directly indicates the film thickness at the orifice outlet. Compared to the 0.9MPa fuel supply pressure, the 0.2MPa fuel pressure generates a thicker film at the orifice outlet, as shown in Fig.3 and Fig.4. The reason was that the fuel viscous force has dominated the film thickness under low-pressure conditions.

Meanwhile, a lower rotated velocity was observed in Fig.5 compared to Fig.6. The velocity for the 0.2MPa case was no more than 4.0m/s while the velocity for the 0.9MPa case was higher than12.5m/s. Pressure increase has accelerated the liquid speed and lessened the film thickness due to the risen aerodynamic force. Consequently, an apparent interface of the two phases was obtained with the increased air-liquid mixing area. Therefore, an inferior atomization quality was found for the 0.2MPa case while a better atomization performance was achieved for the 0.9MPa case. A consistent conclusion was drawn by comparing the simulation results of the three turbulence models, as presented in Fig.7 to Fig.14.

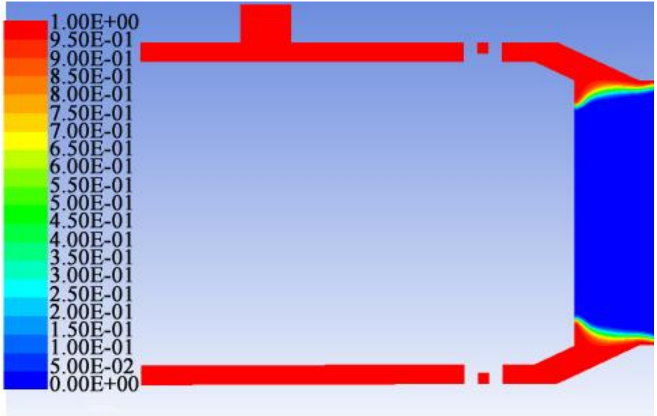


Figure 3 ALVF for RNG model at 0.2MPa condition

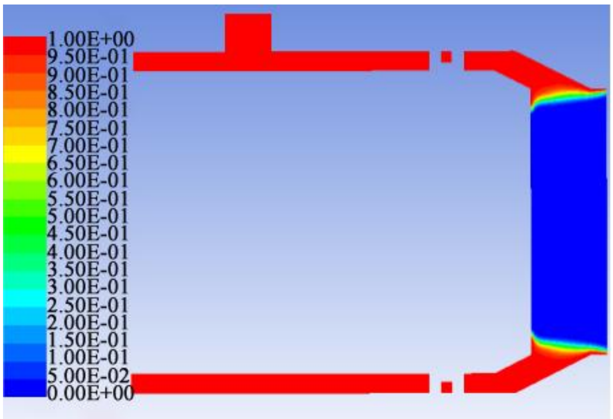


Figure 4 ALVF for RNG model at 0.9MPa condition

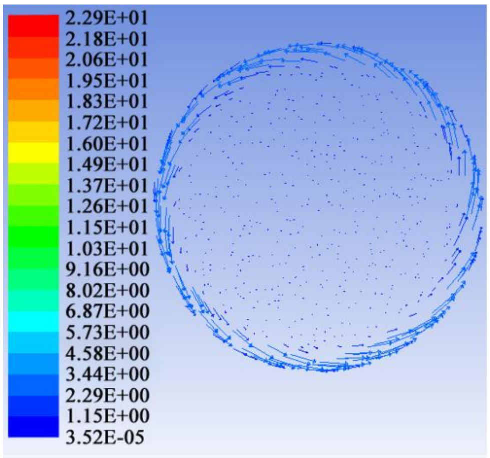


Figure 5 Velocity for RNG model at 0.2MPa condition

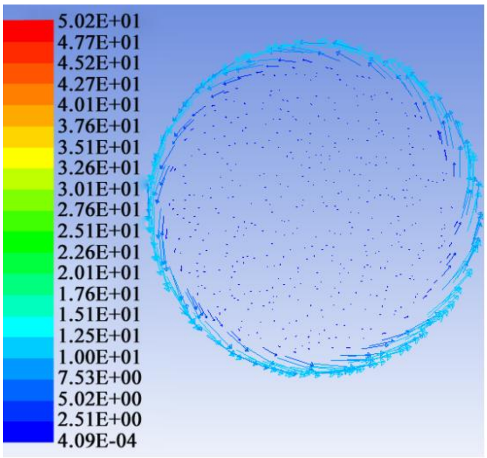


Figure 6 Velocity for RNG model at 0.9MPa condition

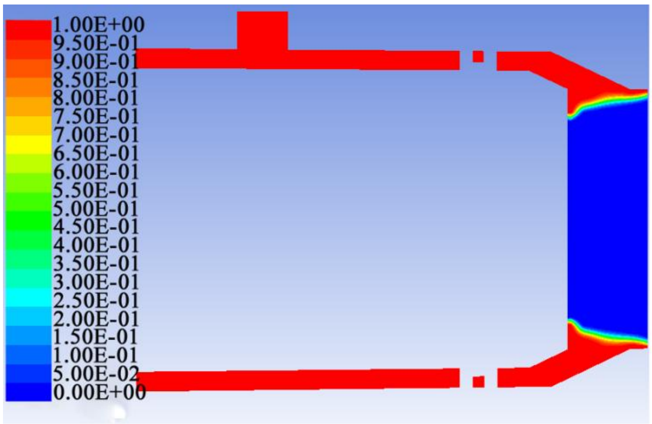


Figure 7 ALVF for RLZ model at 0.2MPa condition

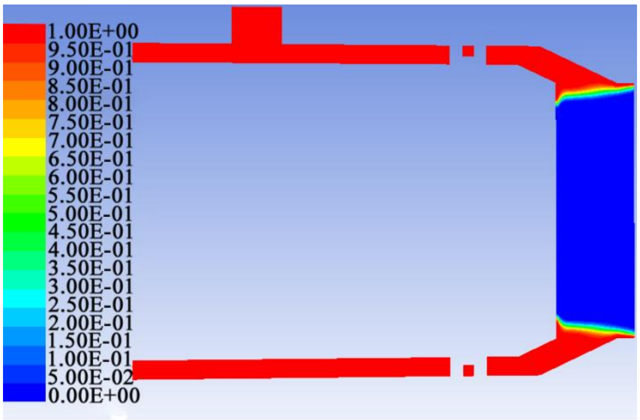


Figure 8 ALVF for RLZ model at 0.9MPa condition

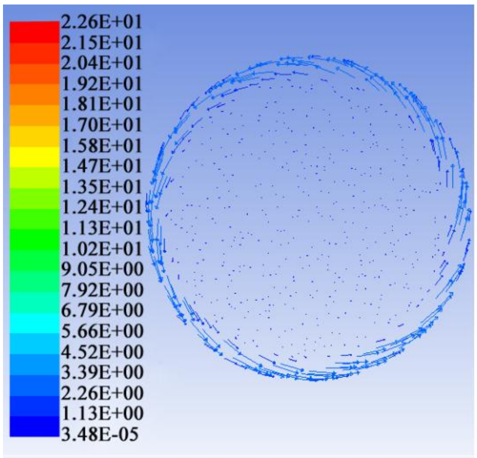


Figure 9 Velocity for RLZ model at 0.2MPa condition

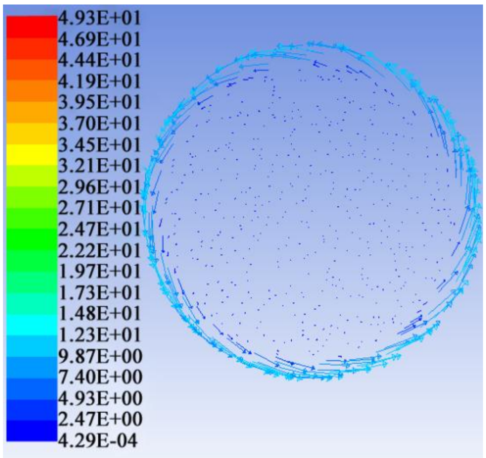


Figure 10 Velocity for RLZ model at 0.9MPa condition

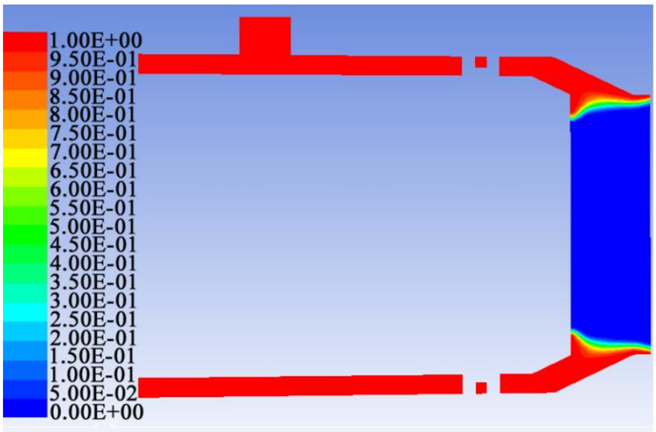


Figure 11 ALVF for STD model at 0.2MPa condition

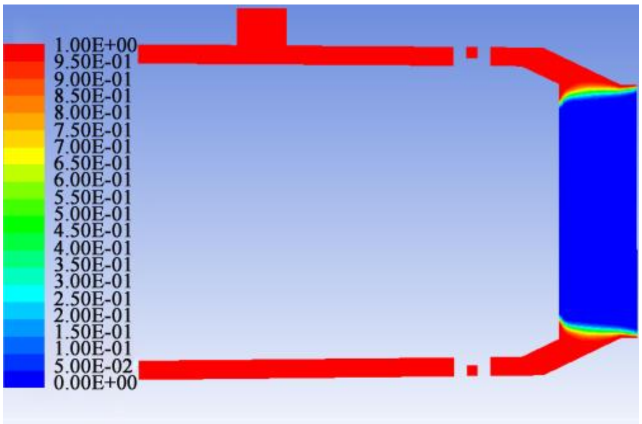


Figure 12 ALVF for STD model at 0.9MPa condition

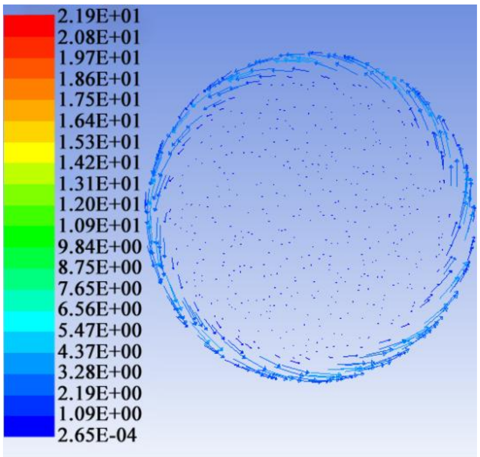


Figure 13 Velocity for STD model at 0.2MPa condition

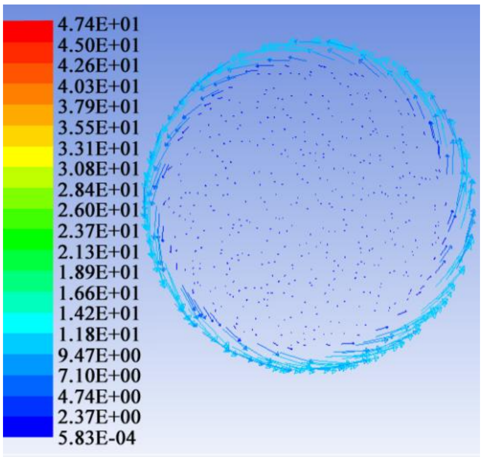


Figure 14 Velocity for STD model at 0.9MPa condition

Another important factor influencing the atomization performance is the spray cone angle since it reflects the dispersion degree of the droplets. It is well-established that the increase of the cone angle would enforce the interaction between the droplets and the air, resulting in improved atomization performance, heat and mass transfer. The velocity magnitude and direction were employed to estimate the cone angle for the three different turbulence models. The airfield, liquid field and mixture field would be obtained by the velocity while the average cone angle would be achieved by Eq. (1).

$$\beta = \arctan(\frac{\sqrt{v_x^2 + v_z^2}}{v_y^2})/\pi/180 \qquad (1)$$

Where,

x, y represent radial direction, z is the axial direction.
β is the average spray cone angle.
v_x is the x component velocity of the fuel.
v_y is the y component velocity of the fuel.
v_z is the z component velocity of the fuel.

Fig.15 presents the estimated spray cone angle along the radial direction at z=10mm section for 0.9MPa case. It was found that when the radius was lower than 9mm, the cone angle was zero degree, revealing a pure air field. When the radius slightly enlarged, the cone angle became negative, which indicates a air-liquid mixing field. The air was injected oppositely to the liquid and rotated with the swirling fuel. Then, the cone angle dropped with the increase in radial distance. Since the fuel was discharged in the form of conical sheet, the outer boundary away from the axis happens a violent interaction between the liquid and air, leading to torn and broken up droplets. A closer inspection on the influences of the three turbulence model was demonstrated in Fig.16.

Results show that the RNG, RLZ and STD model were able to capture the air-liquid interface since the primary nozzle deployed a large outlet diameter. Compared to the test, the RLZ model provided the most accurate results at different operating conditions. The RNG model generates a larger cone angle relative to the test while the STD model was not precise at low fuel supply pressure conditions.

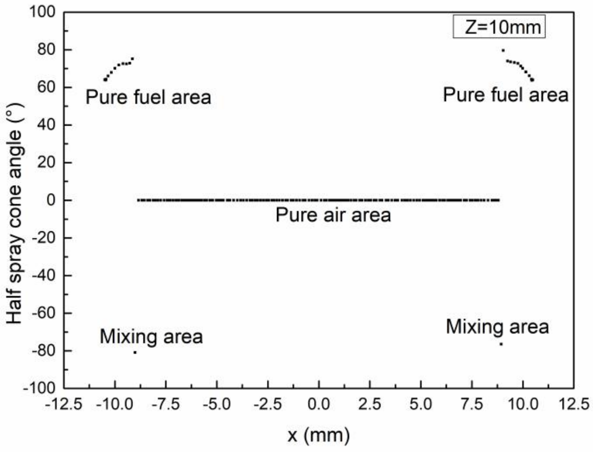


Figure 15 The estimated cone angle for the primary nozzle for 0.9MPa case

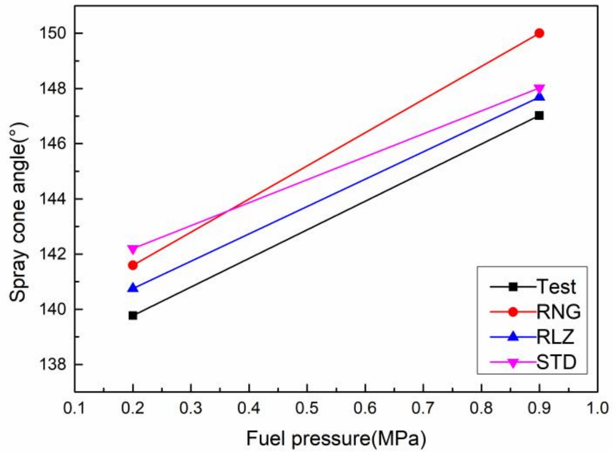


Figure 16 Cone angel comparison for simulation and test

3.2 Pilot Fuel Nozzle

The same analysis was performed to explore the influences of the turbulence model and operating conditions on the pilot nozzle. Keeping the orifice diameter constant, the L_C/d_C is enlarged from 0.7 to 1.2 by increasing the orifice length. The spray cone angle and fuel volume fraction distribution variation are compared in Fig.17 and Fig.18. One interesting observation is that the orifice length is an influential factor affecting the spray cone angle. The longer orifice length provided an intense fuel distribution with onion-shaped spray. It results in inferior mixing and degraded combustion efficiency. Besides, the velocity magnitude fell from 50m/s to 45m/s at the discharge outlet. Nevertheless, a shorter orifice length would generate an overly opened spray cone and splash the fuel to the flame tube wall. This finding has offered a better understanding of the orifice critical parameters design for engineering practice.

It was also found that the elevated fuel supply pressure would boost droplet velocity at the orifice discharge and shorten the film thickness. Specially, the velocity increased from 34m/s to 54m/s when the fuel supply pressure went up from 0.95MPa to 2.6MPa. It could be concluded that a high injection pressure would lead to rapid and improved atomization.

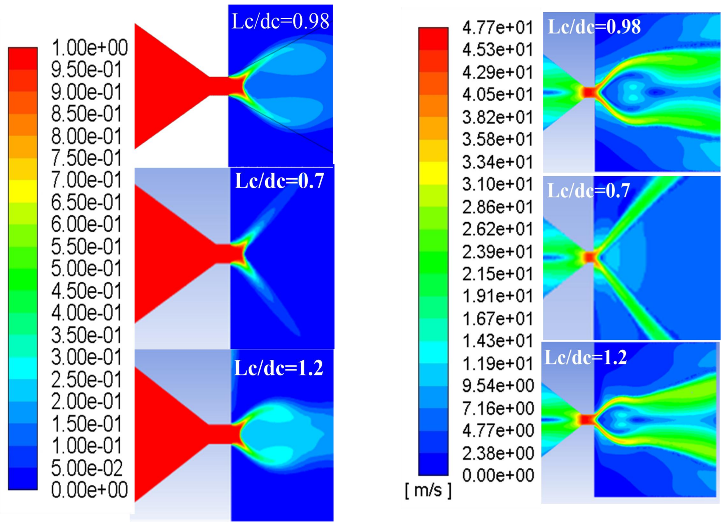


Figure 17 ALVF and velocity distribution for different Lc/dc cases under 0.95MPa

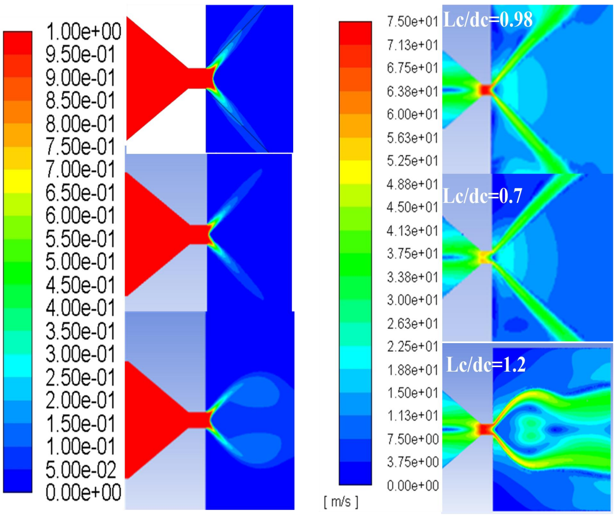


Figure 18 ALVF and velocity distribution for different Lc/dc cases under 2.6MPa

4. Test Results

Fig.18 depicts the experimental data on the effects of the aspect ratio Lc/dc of the discharge orifice on the fuel flow rate. The curves in Fig.19 show that there has been a gradual decline in the fuel flow rate under a specific injection pressure as the aspect ratio increases. The fundamental reason is that a longer discharge orifice would bring in greater pressure-flow loss, implying less pressure energy available for driving the fuel outward. Nevertheless, the increased injection pressure would compensate for the deteriorated fuel flow. Therefore, a proper injection pressure strategy should be employed to accomplish the desired atomization performance.

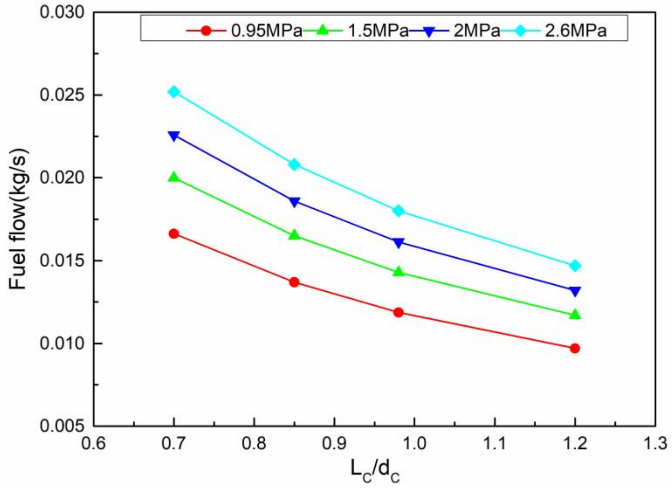


Figure 19 Effects of Lc/dc on fuel flow

Fig.20 demonstrates the variation characteristic of spray cone angle with the fuel supply pressure and the orifice aspect ratio. This figure is quite revealing in several ways. On one hand, the change of the cone angle and the rate of fuel flow show the same pattern as well. In particular, the cone angle falls continuously as the aspect ratio grows, owing to the higher pressure loss along with the extended-release orifice, which results in a lower amount of energy to open the conical spray. Consequently, the generated cone angle was dramatically shrunk, leading to an inferior mixture with the air and combustion efficiency. On the other hand, the increase in fuel supply pressure has increased the cone angle to a greater degree while maintaining the same aspect ratio. A suitable injection pressure would have the potential benefits of raising the evaporation rate and mixing intensity while not posing a risk of liner corrosion in the process of combustion.

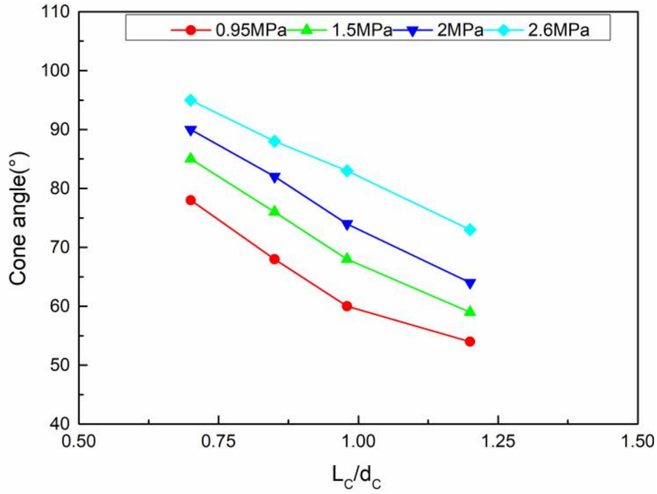
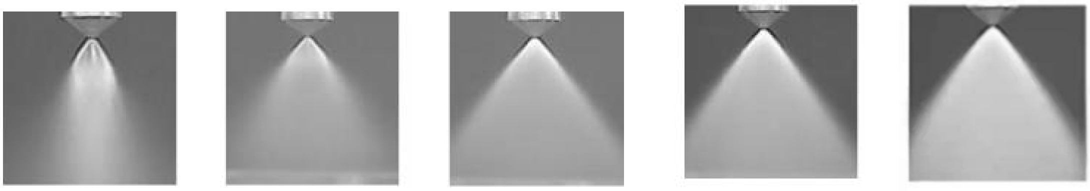


Figure 20 Effects of Lc/dc on spray cone angle

The fuel spray of pilot nozzle under various fuel supply pressure is represented in Fig.21 When the fuel supply pressure was 0.07MPa, the spray demonstrated a form of trumpet-type shape with a particular length of the film. It suggests that the dominance of surface tension and inertia force under low injection pressure settings results in a weakening of the primary atomization process. As a result, it is challenging to disintegrate the film into shreds and ligaments. Obviously, increasing pressure has resulted in a shorter film length, as faster fuel injection would boost the atomization process and more large drops would be fragmented into little droplets. Despite this, when the fuel supply pressure was elevated to 2.6MPa, an evident enhancement in the fuel distribution density and cone angle could be noticed. This would result in a greater amount of heat and mass transfer occurring since the fuel drops were fully exposed to the atmospheric air.



0.07MPa 0.16MPa 0.95MPa 1.5MPa 2.6MPa

Figure 21 Atomization for the pilot fuel nozzle under different injection pressure

5. Conclusion

This paper performs a systematic investigation on the effects of turbulence model and operating conditions on the spray performance of fuel nozzle. The main outcomes are summarized as follows:

1) The investigated three turbulence models all identified the air-liquid interface of the primary nozzle due to the large orifice diameter. However, the RLZ is more suitable for primary fuel nozzle simulation by comparing the numerical results with the experimental results.

2) The RNG model is more accurate in predicting the atomization performance of the pilot nozzle. The calculated spray cone angle was almost the same with the tested value, reaching 60° under 0.95 MPa condition.

3) With the increase in operating pressure, the spray cone angle and the velocity at the orifice outlet were rising concurrently. It indicates that increasing pressure has boosted the rotational intensity in the swirler chamber and generated a large cone angle.

6. Contact Author Email Address

Yixiong Liu: yixiong.liu5021@gmail.com

7. Copyright Statement

Acknowledgement

The authors would like to thank AECC Shenyang Engine Research Institute for the support and fund.

References

[1] Bahr D W, 1985. “Technology for the design of high temperature rise combustors,” AIAA-1985-1292.

[2] Kress E J,1990. “Multiple swirler dome combustor for high temperature rise applications,” AIAA-1990-2159.

[3] PENG S Y, 1978. “Aeroengine combustion chamber structure,” Beijing: National Defense Industry Press, 1978: 47-49. (in Chinese）

[4] JIN R S, 1988. Gas turbine combustion. Beijing:Aerospace Press, 1988:180-200. (in Chinese)

[5] Shin J., et al, 2020. “Effects of the physical properties of fuel on spray characteristics from a gas turbine nozzle,” Energy. Doi: https://doi.org/10.1016/j.energy.2020.118090.

[6] Shatrov M. G., et al, 2019. “Simulation of fuel injection through a nozzle having different position of the spray holes,” Periodicals of Engineering and Natural Sciences, Vol. 7, No. 1, June 2019, pp.458-464.

[7] Kaario T. O., 2020. “The effect of fuel on high velocity evaporating fuel sprays: Large-Eddy simulation of Spray A with various fuels,” International J of Engine Research, 2020, Vol. 21(1) pp. 26-42.

[8] Gavaises M., 2021. “Numerical simulation of fuel dribbling and nozzle wall wetting,” International Journal of Engine Research, pp.1-18.

[9] Pierpont DA and Reitz RD, 1995. “Effects of injection pressure and nozzle geometry on D.I. diesel emissions and performance,” SAE paper 1995-950604. 1995.

[10] Sayin C, et all, 2008. “Influence of injection timing on the exhaust emissions of a dual-fuel CI engine,” Renewable Energy 2008; 33: 1314–1323.

[11]Pierpont DA, et al, 1995. “Reducing particulate and NOx using multiple injections and EGR in a D.I. diesel,” SAE technical paper 950217, 1995.

[12] Durnholz M, et al, 1994. “Preinjection a measure to optimize the emission behavior of DI-diesel engine,” SAE paper 940674, 1994.

[13] Molina S, et al, 2014. “A computational investigation on the influence of the use of elliptical orifices on the inner nozzle flow and cavitation development in diesel injector nozzles,” Energy Convers Manage 2014; 79: 114–127.

[14]Taylor G I, 1948. “The mechanics of swirl atomizers,” Seventh International Congress of Applied Mechanics. London: Imperial College, 1948: 280-285.

[15] Rizk N K and Lefebvre A H, 1987. “Prediction of velocity coefficient and spray cone angle for simplex swirl atomizers,” International Journal of Turbo and Fuel Nozzles. 1987, 4: 65-73.

[16] Rizk N. K. and Lefebvre A. H, 1985. “Internal flow characteristics of simplex swirl atomizers,” Journal of Propulsion and Power, 1985, 1(3):193-199.

[17] Simmons H. C., 1977. “The correlation of dropsize dist ribution in fuel nozzle sprays,” Journal of Engeering for Gas Turbine and Power, 1977, 99 (3): 309-319.

[18]Darmawan S , Budiarso, Siswantara A I . “CFD Investigation Of Standard k- and RNG k- Turbulence Model In Compressor Discharge Of Proto X-2 Bioenergy Micro Gas Turbine”. Karya Ilmiah Dosen Teknik Mesin, 2014.

[19] Xu J S, Li DK, Hu J R, et al. “Realizable k-ε Model Simulation on the Structure of Flame Induced by an Obstacle”, Industrial Safety and Environmental Protection, 2014, 40(4):40-42.(in Chinese)