"APISAT2014", 2014 Asia-Pacific International Symposium on Aerospace Technology, APISAT2014

Atomization Performance Study of a Fuel Injector in IGC by Experimental and Numerical Investigation

Can Ruan[a], Xiaoyuan Fang[a], Guanxing Huang[b], Hongzhou Ho[b], Fei Xing[a],*

[a]*Department of Aeronautics, Xiamen University, Xiamen, Fujian, 361005 ,China*
[b]*Cleaning Combustion and Energy Utilization Research Center of Fujian Province, Department of mechanical engineering, JiMei University, Xiamen, Fujian, 361021, China*

Abstract

Fuel spray and atomization characteristics play an important role in the performance of internal combustion engines. Experimental and numerical atomization studies of the fuel injector used in IGC (Inner Guide-vane Combustor) under various assembly and operation conditions have been performed in order to determine the configurations of the fuel injector proposed for use in IGC. Droplet Sauter Mean Diameter (SMD) distributions were measured with Phase Doppler Analyzer (PDA) at different operating conditions and used to evaluate the atomization performance. Computational Fluid Dynamic (CFD) study has also been conducted based on Discrete Particle Model (DPM).Current work examines optimizing the atomizer location and assembly condition of these vanes and the cavity within them. Results reveal that the distribution of droplet SMD varies with the assembly and operation condition. The further results showed that fuel injector used in IGC with single slot, forward injection direction and '<' shaped backsplash type can get the best atomization performance compared with other cases tested

1. Introduction

The ITB (inter-stage burner) technology is expected to provide a new way to design greener and more efficiency propulsion for the future. Sirignano et al. [1] find that combustion which is continued purposely inside the turbine

---

 * Corresponding author. Tel.: +86-13063039116.
   E-mail address: f.xing@xmu.edu.cn

can increase the efficiency and specific thrust/power of engine. Sirignano et al. [2] publish the plan that combustor placed between the stage of high-pressure turbine and low-pressure turbine will be easier to realized, which is known as ITB. In NASA's research [3], the Trapped Vortex Combustion (TVC) is assembled between the high and low pressure turbine stage in turbojet and turbofan engine. Jonathan F. Anisko et al. [4] conduct a numerical analysis and design optimization of the Ultra Compact Combustor (UCC) and find that a high acceleration and high turbulence environment can be formed in the circumferential cavity, which shows to reduce flame lengths by ~50% over conventional combustors for similar overall equivalence ratios.

The characteristic of the fuel injector's atomization performance is one key factor we should care about for new turbine engine design. For air-blast atomizer, N. K. Rizk and A. H. Lefebvre [5] tested eight different air-blast atomizers and found that atomization quality was improved by increase in ambient air pressure, decrease in atomizer size, and by minimizing the angle of impact between the fuel jet and the high-velocity air stream.

Fig. 1 shows the overall schematics of Inner Guide-vane Combustor (IGC) and the fuel injector used in IGC. The first step of fuel atomization is conducted by the air-blast atomizer in the fuel injector. Fuel droplets will get heated and evaporated when they strike on the surface of the guide vane just in front of the atomizer. At the same time, the vane will be cooled and protected from the heat generated by the coming flow. The fuel-air mixture will flow out of the vane through the slot and form vortex with the coming flow, the cavity can be used as flame holder.
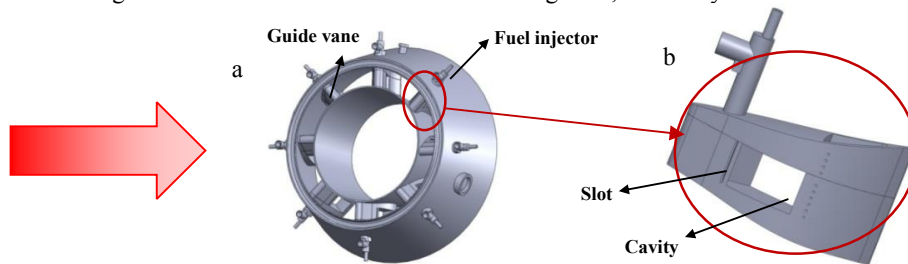

Fig.1. (a)overall schematics of IGC; (b)fuel injector in IGC

2. Experimental procedure

2.1. Test rig system

A schematic view of the Phase Doppler Anemometry (PDA) experimental system setup is shown in Fig.2. The maximum diameter which can be measured with this setup is 250μm. Droplet distributions at different locations were measured by adjusting a three dimensional auto control coordinate system with a size of $200 \times 200 \times 280$ mm.
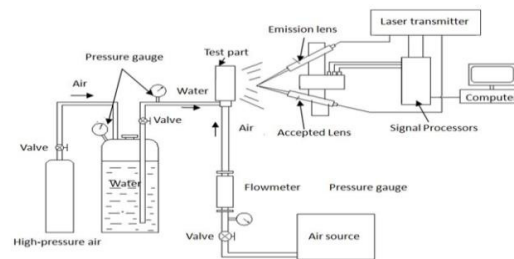

Fig. 2. schematic of experiment system

2.2. Experimental cases

In order to analysis the effect of different assembly schemes on the atomization performance of the fuel injector in IGC, experimental measurements under different parameters, such as air liquid ratio (ALR), injection direction, backsplash and slot type, are needed. Experimental operation conditions and cases are listed in Table 1 and 2.The experimental conditions covered the range of P_a from 0.1 MPa to 0.2MPa and ALR from 0.80 to 0.92 (the pressure of water keeps constant at 0.1MPa). Take the real working geometric environment of the fuel injector into consideration, for each case, droplet SMD was acquired at 3 spatial locations within a range of 8-20 mm in L direction out of the slot (shown in Fig.4 (a)) and 5~16 mm in S direction in the cavity （shown in Fig.4 (c)). At each location, 2000 samples were acquired with PDA.

Table 1.Operation conditions. (pw=0.1MPa, mw=2.88g/s).

$P_a(MPa)$	0.1	0.12	0.14	0.16	0.18	0.2
$m_a(g/s)$	2.32	2.48	2.54	2.57	2.60	2.62
ALR	0.80	0.86	0.88	0.89	0.90	0.91

Table 2.Experimental cases of fuel injector.

No.	Slot type	Inject direction	Backsplash type
1	single	Forward	/
2	single	Backward	/
3	intersect	Forward	flat
4	intersect	Forward	'>' shaped
5	intersect	Forward	'<' shaped

Both three-dimensional (3-D) and two-dimensional (2-D) geometry schematics are shown in Fig. 3 and Fig. 4. Fig. 4 gives 2-D operation schematic for experimental cases, the first step of fuel atomization is conducted by the air-blast atomizer in the fuel injector, and then the fuel-air mixture will flow out of the vane through the slot and form vortex with the coming flow, the cavity works as the flame holder as shown in Fig.4.
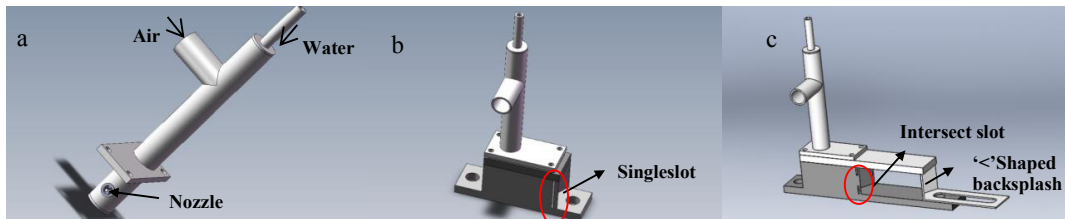

Fig. 3. Schematics of experimental articles (example). (a) air-blast atomizer; (b) No.1 fuel injector; (c) No.5 fuel injector
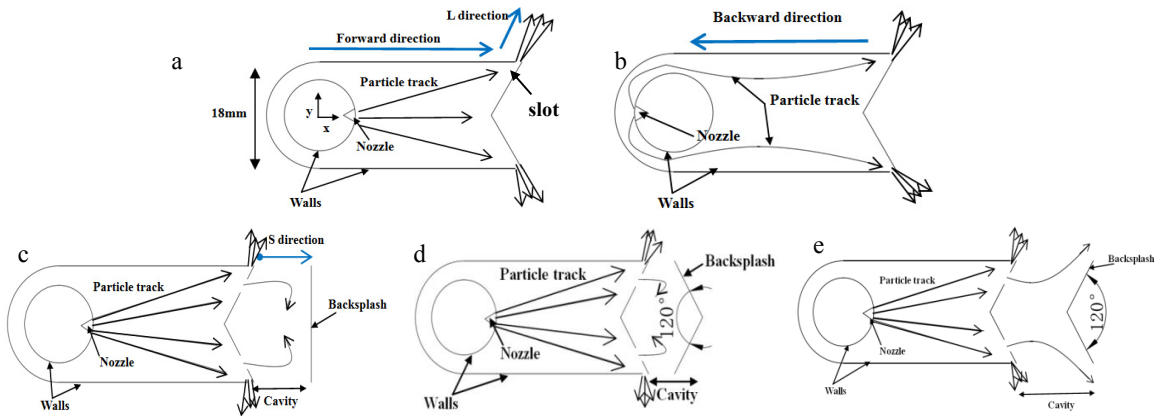
Fig. 4. Operation sketch for fuel injector at plane Z=0. (a) No.1; (b) No.2; (c) No.3; (d) No.4; (e) No.5

3. Numerical model procedure

In order to have more details of the whole spray field of the fuel injector, a Computational Fluid Dynamic (CFD) study has also been conducted based on the Discrete Particle Model (DPM) [6] with commercial CFD software FLUENT 14.0. Fig.5 shows the three-dimensional (3-D) computational domain and boundary conditions for No.1 and No.5 experimental cases. The flow field of continuous phase was firstly resolved with standard k-ε turbulence model [7], and then the solution results of the continuous phase were coupled with the discrete particles with the DPM model, FLUENT computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included. The coupling procedure of continuous and discrete phase is shown in Fig.5 (c).
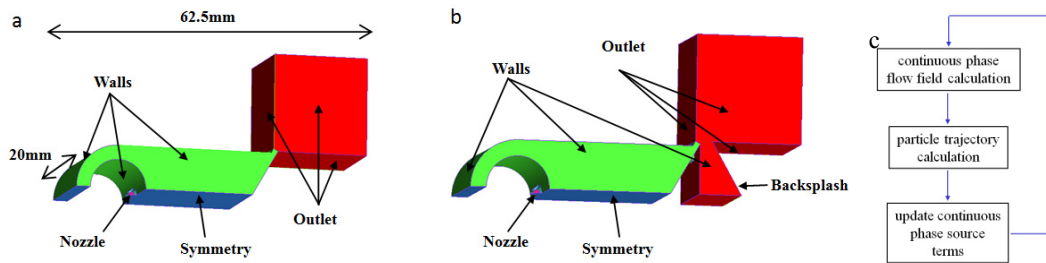

Fig .5. Geometry and boundary conditions, three-dimensional.(a) No.1 fuel injector; (b) No.5 fuel injector; (c) two-phase coupling procedure

4. Results and discussion

4.1 .Fuel atomization procedure and atomization performance evaluation index

As illustrated in Fig.3 and Fig.4, after releasing from the air-blast atomizer, the fuel droplets will flow into the vane and come out from the slot combined with the coming flow. The effect on atomization of injection direction, slot and backsplash type is discussed in this study. The distribution of droplet Sauter Mean Diameter (SMD) is used to evaluate the atomization performance of the fuel injector in IGC under different working conditions at different given locations, Rosin-Rammler (R-R) [8] distribution is used to get the uniformity index *n* and evaluate the uniformity of the droplet size.

4.2. SMD distribution for different injection directions

Fig. 6 shows the SMD distribution of No.1 and No.2 fuel injectors at the different downstream locations of the slot (L is the distance between the measuring point and the slot, D is the diameter of the nozzle). Fig.6 shows that SMD is much smaller (about 50% smaller than backward case) and has a better distribution when water is injected forward.
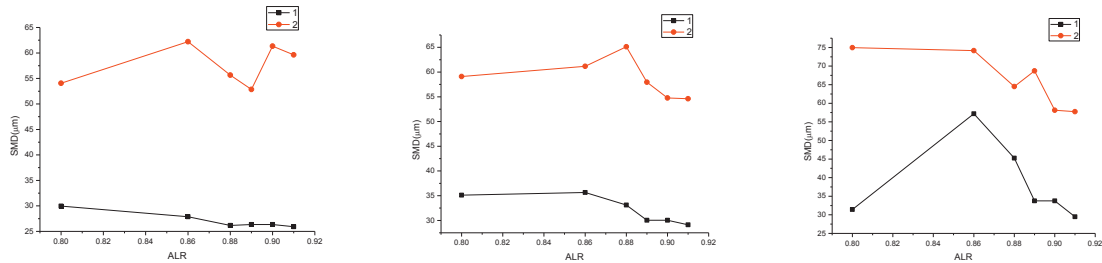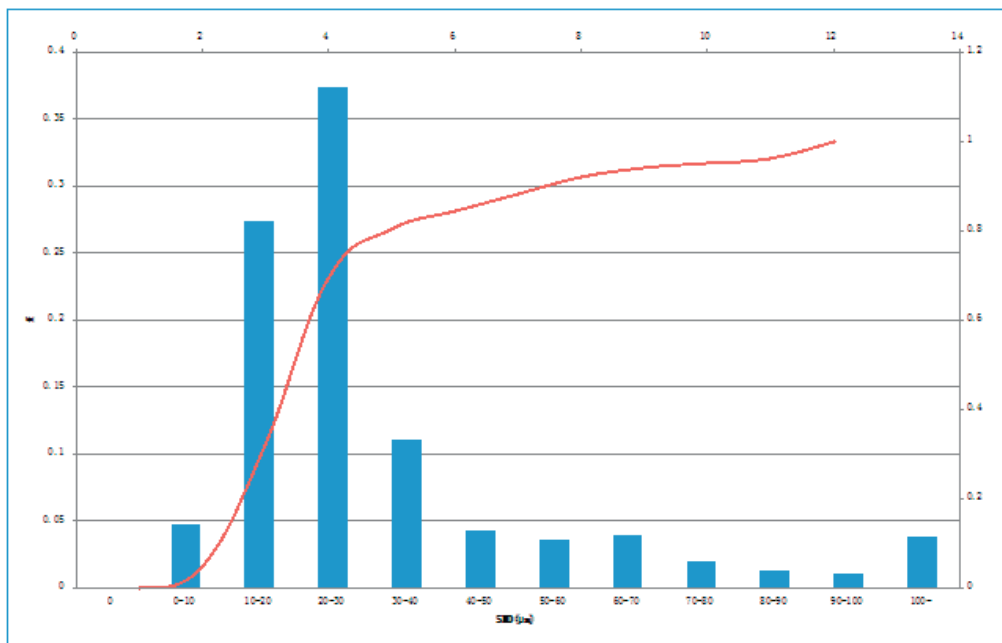

Fig. 6. SMD distribution for the No.1 and 2 fuel injector (a) L/D=13; (b) L/D=21; (c) L/D=33

It can also be inferred from Fig.7 and Fig.8 that No.1 and No.2 fuel injectors share a similar uniformity index (*n*), however, over 90% particles of No.1 injector are under 50μm while 60% of No.2 injector at the given condition, which means that the forward injecting case have a better atomization performance over the backward case. This situation occurs mainly because the continuous phase of the backward is weaker in the vane and behind the slot for backward injection case compared with forward injection case, which reduces the process of droplet breakup, thus getting bigger SMD distribution.
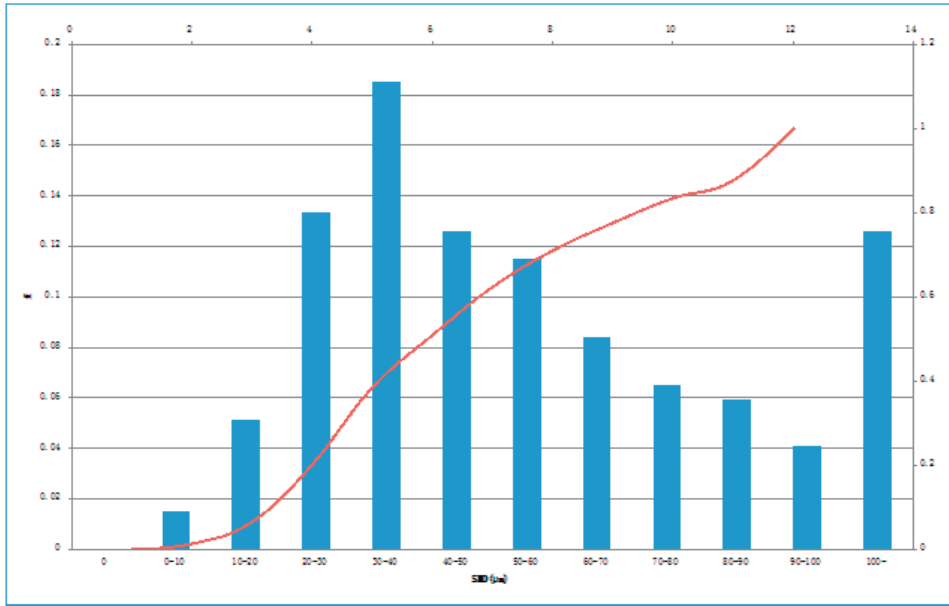
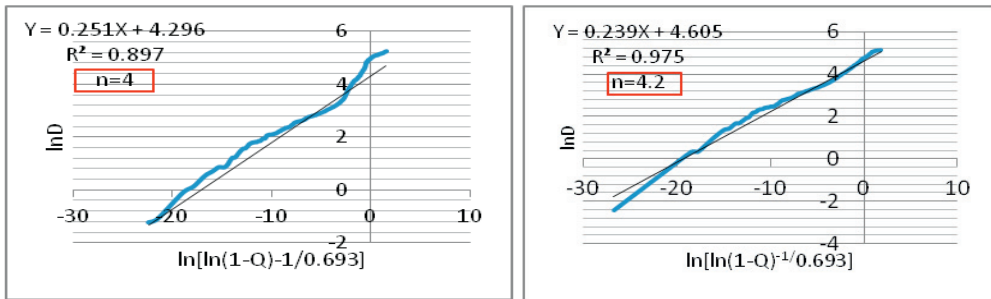Fig. 7. SMD frequency distribution (a) No.1; (b) No.2 (ALR=0.9，L/D=21)

Fig. 8. SMD R-R treatment (a) No.1; (b) No.2(ALR=0.9，L/D=21)

4.3. SMD distribution for different slot types

There are experimental results of No.1 and No.3 fuel injector. After the first atomization conducted by the air-blast atomizer in fuel injector, the fuel-air mixture will flow out of the vane through the slot and form vortexin the cavity, where fuel and air mixture can be ignited and form steady combustion. The SMD distribution of No.1 and No.3 fuel injector at different downstream locations in L direction are shown in Fig.9. The SMD of No.1 fuel injector shows a more obvious pulsation in the field far from the slot, however, smaller than that of No.3 fuel injector by a percentage of 20%.
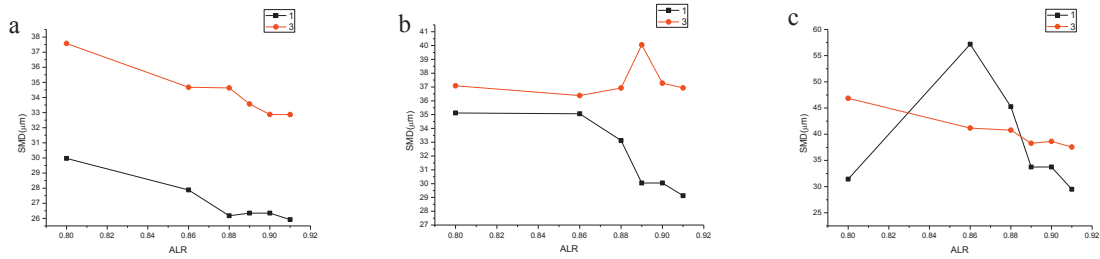
Fig. 9. SMD for the No.1 and 3 fuel injector (a) L/D=13; (b) L/D=21; (c) L/D=33

It can also inferred from Fig.10 that SMD of No.1 fuel injector are mainly concentrated in the range of 10~50µm which adds up to over 90% of the whole fuel droplets group, while 70% for No.3 fuel injector. These results are due to the continuous phase weakened by intersect slot so that the disturbance of the air force also declines, which reduce the break of the droplet, therefore increasing the size of SMD. This phenomenon can be proved by the numerical study results later.
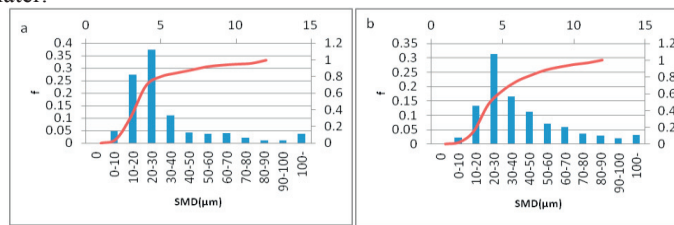

Fig. 10. SMD frequency distribution (a) No.1; (b) No.3 (ALR=0.9，L/D=21)

Fig.11 shows the SMD fitting results for No.1 and No.3 fuel injector by R-R treatment at given condition, a higher uniformity index (*n*) shows that No.1 fuel injector can get a more uniform droplet SMD distribution compared with No.3 case. Droplet's breakup was weak due to a low air momentum behind the slot.
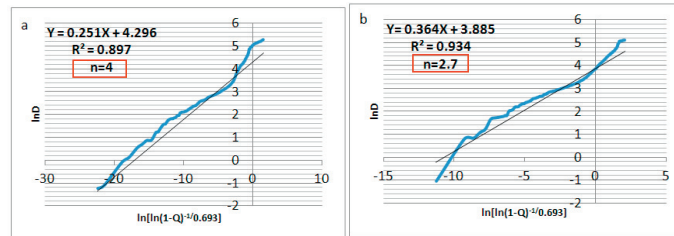

Fig. 11. SMD R-R treatment (a)No.1; (b) No.3 (ALR=0.9，L/D=21)

4.4. SMD distribution in the cavity for different backsplash types

In order to evaluate the effect of backsplash type on SMD distribution in the cavity, comparative atomization experiment continues with No.3, 4 and 5 fuel injector. For each case, the results of SMD vs ALR at given location are shown in Fig.12. We find that the SMD of No.5 fuel injector is smaller than that of No.3 fuel injector by the percentage of about 40% and 20% for No.4, which means No.5 fuel injector can get the best atomization performance among them. We speculate that this is because the air can get a relatively high velocity in the cavity

area with '<' shaped backsplash, which results in a bigger *We* and the breakup process of droplet in the cavity is strengthened.
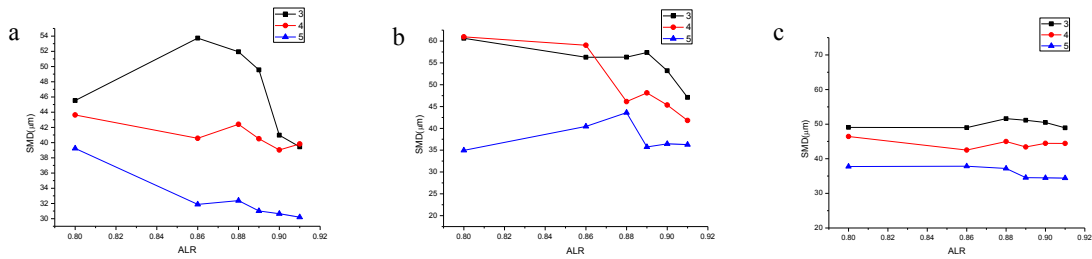

Fig.12 influence of backsplash type on SMD in the cavity (a) S/D=8; (b) S/D=13; (c) S/D=26

4.5. Numerical simulation results

Computations are performed with commercial CFD software FLUENT 14.0, both three-dimensional and two-dimensional geometry and boundary conditions are shown in Fig.4 and Fig.5, the breakup, evaporation and collision processes of the droplets were taken into consideration in this numerical simulation. A whole three-dimensional computational domain was also established. The DPM model was used to get the whole atomization field of the fuel injector ranges from No.1 to No.5, particle diameter simulation results are illustrated in Fig.13. As shown in Fig.13 (a) and (b), with forward injection direction, the droplets behind the slot shows a more uniform distribution without the coming flow. Furthermore, it can be inferred from Fig.13 (c), (d) and (e) that No.5 fuel injector can get the best atomization field, these conclusions are in good agreement with the experimental results.
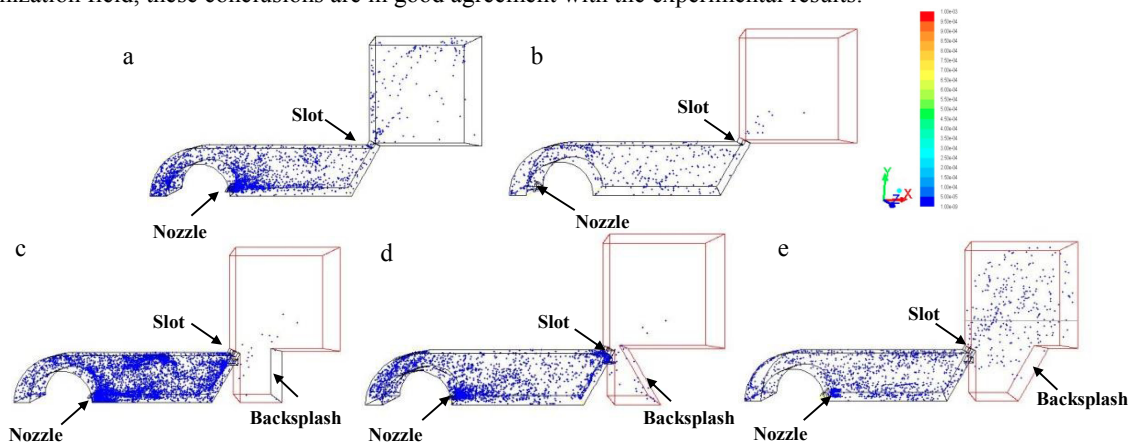

Fig. 13. atomization field simulation results (a) No.1; (b) No.2; (c) No.3; (d) No.4; (e) No.5

Conclusion

In present work, experimental studies of the fuel injector used in IGC under various assembly conditions have been performed without the coming flow, results show that the test cases can all get good atomization performance, while test case with single slot, forward injection direction and '<'shaped backsplash type can get the best atomization performance than other cases; numerical simulation with DPM model in FLUENT shows a good agreement with experimental results, furthermore, the hot coming flow and high operation temperature will be taken

into consideration. These results can provide necessary references for turbine engine design in the future.

Reference

[1] Sirignano W A, Liu F. Performance increases for gas-turbine engines through combustion inside the turbine[J]. Journal of Propulsion and Power, 1999, 15(1): 111-118.
[2] Delplanquef W A S J J R, Liu F. Selected challenges in jet and rocket engine combustion research[J]. 1997.
[3] Liu F, Sirignano W A. Turbojet and turbofan engine performance increases through turbine burners[J]. Journal of Propulsion and Power, 2001, 17(3): 695-705.
[4] Anisko J, Anthenien R A, Zelina J. Numerical investigation of cavity-vane interactions within the ultra compact combustor[M]. Defense Technical Information Center, 2006.
[5] Rizk N K, Lefebvre A H. Influence of atomizer design features on mean drop size[J]. AIAA Journal, 1983, 21(8): 1139-1142.
[6] Dukowicz J K. A Particle-Fluid Numerical Model for Liquid Sprays [J]. J Comp Physics, 1980, 35(2): 229-253.
[7] Wilcox D C. Turbulence modeling for CFD [M]. La Canada, CA: DCW industries, 1998. 87-90.
[8] Varga C M, Lasheras J C, Hopfinger E J. Initial breakup of a small-diameter liquid jet by a high-speed gas stream[J]. Journal of Fluid Mechanics, 2003, 497: 405-434.