Gas dynamics and mixture formation in swirled flows with precession of air flow

V V Tretyakov and A A Sviridenkov
Central Institute of Aviation Motors named after P I Baranov, Moscow 111116, Aviamotornaya 2, Russia
Email: tretjak@ciam.ru

Abstract. The effect of precessing air flow on the processes of mixture formation in the wake of the front winding devices of the combustion chambers is considered. Visual observations have shown that at different times the shape of the atomized jet is highly variable and has signs of precessing motion. The experimental data on the distribution of the velocity and concentration fields of the droplet fuel in the working volume of the flame tube of a typical combustion chamber are obtained. The method of calculating flows consisted in integrating the complete system of Reynolds equations written in Euler variables and closed with the two-parameter model of turbulence \( k-\varepsilon \). Calculation of the concentration fields of droplet and vapor fuel is based on the use of models for disintegration into droplets of fuel jets, fragmentation of droplets and analysis of motion and evaporation of individual droplets in the air flow. Comparison of the calculation results with experimental data showed their good agreement.

1. Introduction
This work is the result of experimental and computational investigations of non-stationary swirling flows in the combustion chambers of gas turbine engine and the influence of these flows on the spraying and distribution of fuel in the working volume. It is known that with increasing flow swirl number of inlet combustion chamber vortex devices there is an improvement in their spray properties: decrease of the droplet size and increase of the uniformity of their distribution in the cross section of the spraying torch. Experiments have shown that with an increase of flow swirl angle up to 60° and higher a precessing gas flow occurs. The presence of this motion was observed experimentally, see [1], in the flows behind the gas-dynamic stabilizer (flame holder), the scheme of which is shown in Figure 1. It turned out that the shape of the atomized jet varies greatly at different times and has signs of precession motion. The appearance of this precession is associated with the hydrodynamic instability of the flow and is a non-stationary process.

The methodology for calculating fuel distributions presented in this paper is as follows. First, a velocity field is calculated from the stationary \( k-\varepsilon \) model of turbulence at effective Reynolds numbers of the air flow leaving the front swirling device \( Re_{ed} > 300 \). In this range the flow under consideration is unstable and repeated patterns of velocity distributions occur at different stages of the iteration process. Random samples of these structures are accepted as instantaneous fields of gas flow velocity, and the considered set of these instantaneous fields is a model of non-stationary flow. The calculation procedure is described in detail in [2]. The second stage calculates the instantaneous field concentrations corresponding to the obtained instantaneous velocity fields. Finally, at the third stage
the concentration fields are averaged to result in the corresponding concentration fields. The averaging of the instantaneous velocity and concentration fields is carried out by realizations.

2. Experimental determination of droplet fuel concentrations

The experimental investigation method is based on the use of a small-angle light wave scattering. The resulting values of the volume concentration of particles refer to a certain conditional value associated with the longitudinal dimension of the scattering volume of the laser beam, which is 1 cm in this study. In experiments the operating regimes were chosen in such a way as to provide an unsteady flow with the precession of a vortex at the exit of the swirl device. The results of visual observations showed that at all swirling device operating regimes the sprays are essentially non-stationary. It is confirmed by direct measurements of the droplet concentration pulsations in a fixed point of the spray at different instants of time, as illustrated in Figure 1. The sinusoidal nature of the variation in time of the mean concentration ripple values is explained by the presence of precessing air flow motion. In the figure, $C_v$ is the volume concentration of droplets (the ratio of the total volume of droplets in the cell to its volume). Experiments have shown that the size of fuel droplets lies in the range of 20 to 100 $\mu$m near the spraying site. In subsequent sections downstream the dimensions of the fuel droplets remain practically unchanged. These results indicate that in this case the main spraying of fuel occurs in the immediate vicinity of the fuel supply site and at considerable distances it only mixes with air.

![Gas dynamics stabilizer scheme and fuel pulse concentration behind stabilizer on time](image)

Figure 1. Gas dynamics stabilizer scheme and fuel pulse concentration behind stabilizer on time

1- swirl device, 2- frontal device, 3-fuel channel, 4-air jet channel

Experiments have shown that the distribution of droplets along the radius of the spraying torch is fairly uniform while the distribution of their concentrations in the combustion chamber test volume is largely determined by the characteristics of the air flow. The same is confirmed by the results of numerical calculations.

3. Calculation of velocity and concentration fields

Variable parameters of the velocity field calculations were the flow swirling number at the outlet of the swirl device $Sw = V_\phi/V_z$ and the injection velocity of transverse air jets $V_j$, where $V_\phi$, $V_j$ and $V_z$ are the rotational, radial and longitudinal velocity components. Here and below all linear dimensions are assigned to the characteristic size $H_0 = 10$ mm (internal radius of the swirl device) and velocities to the characteristic airflow velocity $V_0 = 20$ m/s (medium-velocity flow through the swirl device). Calculations have shown that for non-stationary flow the instantaneous velocity distributions differ substantially from the averaged ones. In addition it turned out that the precessing jet leaving the combustion chamber has a width 3 times wider than the jet without precession. The flow in the jet is non-stationary with increased ripple values. Thus at a distance of one caliber from the nozzle cut the turbulence level is $30 \div 50\%$, which is much higher than in conventional jet streams (10%). This explains the improvement of the mixing properties of the considered flow.
The method for calculating the droplet-liquid fuel concentration fields is based on models of the disintegration of fuel jets onto droplets, second atomization of droplets in the air flow, and analysis of the movement of individual droplets. It is assumed that the droplet size distribution obeys the Rosin-Rammler law with exponent $n = 3$, and the droplet distribution in the circumferential direction (along the angle $\phi$) and the longitudinal direction (along the angle $\psi$) is normal. The initial discharge velocities are given by the velocity module $V_0$ and the values of the longitudinal and circumferential angles: $\psi_d$ and $\phi_d$. The initial coordinates of the emission of droplets are determined from the solution of the disintegration problem of the fuel jet injected into the air stream, the departure angles are determined from the experimental data of sputtering in the open space, the temperature of the drops is assumed equal to the temperature of the air flow. The mean median droplet diameter in the spraying torch is $D_m = 50 \, \mu$, the fuel consumption $G_f = 2.5 \, $g/s, which corresponds to the experimentally investigated regime $\Delta P_a = 3 \, $kPa.

Figure 2. The distribution of mass droplet concentrations obtained in calculations with a) averaged and b) instantaneous velocity fields.

Figure 2 shows the results of calculations of the distribution of the mass concentrations of droplets obtained in the averaged (Fig. 2-a) and instantaneous (Fig. 2-b) velocity fields. In the figures, the mass concentration of droplets $C_d$ is related to the air density. It can be seen that in the stationary case, fuel droplets entrained by the air flow are concentrated in the zone of return flow near the place of fuel injection (at the end of the injector) and are practically absent in the wall area of the flame tube. In the non-stationary case, droplets in the process of their motion turn out to be in the near-wall region of the chamber. Thus, the drop in the droplet fuel on the wall of the combustion chamber is determined by the non-stationary nature of the flow in question.

4. Comparison of calculation and experimental results

When comparing the results of calculations and experiments, the calculated profiles of concentration distributions were transformed: the mass concentrations of droplets were recalculated into volumetric and local fuel distributions were recalculated into averaged over the ray, in accordance with the experimental procedure. In Figure 3 and 4 show examples of comparing the calculating droplet fuel concentration field results with experimental data in two cross sections of the spraying flare: $X = 30$ mm (left figures) and $50$ mm (right figures), where $X$ is the distance from the outlet cross-section of the front swirl device. In the figures, the blue curves are experimental the data, the red ones are the results of calculations. Figure 3 corresponds to the calculations in the non-stationary air flow, Figure 4 - in the stationary one.

Comparison of these figures shows that the concentration distributions in the inner part of the fuel flare by both models are described approximately identically, however the distributions obtained in the non-stationary calculations correspond to the experimental data better than stationary ones. Thus calculation using the stationary model shows that the fuel drops in the cross section $X = 30$ mm are completely absent at radial distances greater than $20$ mm, and in the section $X = 50$ mm - at distances
exceeding 30 mm. At the same time the calculation of the non-stationary model demonstrates agreement with the experimental distributions over the entire radial range. On the whole the calculations showed that their results with practice accuracy describe the experimental data, the instantaneous fields of the droplet concentrations being significantly different from the averaged ones, and the averaged field of the non-stationary flow regime differs substantially from the stationary field. In addition the distribution of the droplet concentrations obtained by averaging their instant distributions is significantly different from the concentration distribution calculated in the averaged velocity field. These differences are illustrated by the graphs given. Their comparison shows that the presence of droplets in the vicinity of the combustion chamber flame tube walls observed in experiments can be explained only on the basis of a non-stationary flow model.

![Figure 3](image1.png)

**Figure 3.** Average distribution of fuel concentration over the spray cross section calculated in instantaneous velocity fields.

![Figure 4](image2.png)

**Figure 4.** Average distribution of fuel concentration over the spray cross section calculated in stationary velocity fields.

5. **Disintegration of fuel jets in non-stationary air flow**

An essential part of the experimental study is a technique for visual observation of jet propagation and disintegration, consisting in the use of pulsed flood lighting with a flash duration of 1/500 s. The experiments showed that in all swirling device working regimes of spraying torch in the open space is essentially non-stationary. In this case the fuel jet in the considered modes performs rotational and vibration motions with the amplitude of spatial oscillations approximately equal to the output swirl device diameter. The visually observed shape of this jet is the averaged distribution of droplets in the fuel-air flare. The presence of unsteady flow is fundamental fact in relation to the mechanism of the disintegration of a fuel jet in the open space. When a fuel jet enters the air stream behind the swirl device, the jet undergoes rotational-oscillatory motion under the influence of the vortex precession. The frequency and amplitude of these oscillations are sufficiently high, and these oscillations can be observed only when the flare is pulsed.
Experimental studies of the propagation and disintegration of a liquid jet inside the combustion chamber have shown that the presence of flow boundaries significantly changes the nature of the propagation of the jet and its disintegration mechanism. Thus, waves caused by the Rayleigh instability are formed on the surface of disintegrated jet in the chamber. The amplitude of these waves when interacting with a swirling air stream increases and the jet disintegrates into individual drops. This disintegration is observed already at the jet mouth, so that a portion of the drops from the jet surface breaks off in the immediate vicinity of the fuel supply site. Note that the droplets detached from the jet fall on the chamber walls under the action of a swirling air flow. It makes directly measuring the spraying torch characteristics very difficult. Another feature of the liquid jet disintegration inside the chamber is that the jet does not make noticeable oscillations in the space confined by the chamber walls. This is apparently due to the air flow into the free space is precessing, while in the combustion chamber the precession of the flow is either very weak or nonexistent. We note that an increase in the air flow through the swirl device leads to the increase of undisturbed jet portion length. This fact, which is not obvious at first sight, is explained by the change in the structure of the air flow field behind the swirl device. In general the jet disintegration is described qualitatively by the stability curve given in [3]. At first the extent of the undisturbed portion of a fuel jet increases monotonically with increasing velocity of the jet and reaches its maximum value when the aerodynamic interaction of the air stream with the jet begins to destabilize its core. A further increase in the velocity of the jet leads to a decrease in the length of its undisturbed part due to deformation of the jet shape. With a subsequent increase in the jet velocity the length of the undamaged core of the jet decreases and it disintegrates into droplets.

6. Conclusions

When the air flow from the model chamber to the open space behind the swirl device unstable oscillations of the reverse current axial zone occur and the jet that exits the chamber has a precessing core. The distribution of fuel in the cross section of the jet behind the swirl device is fairly uniform. Calculations performed on the developed model for the decomposition of liquid fuel jets have shown that the dimensions of the droplets formed are 28 - 29 μm, which agrees well with their experimental values of 30 - 35 μm. The results of calculations of droplet fuel concentrations conducted on a non-stationary model with acceptable accuracy for practice correspond to experimental results. In this case, the fraction of droplets falling on the walls of the flame tube when calculating in a non-stationary velocity field is more than 1.5 times the calculated values in the averaged field.

Acknowledgments
This work was sponsored by the RFBR, Project N 17-01-00213

References
[1] Tretyakov V V, Sviridenkov A A 2016 Unsteady heat and mass transfer and fuel distribution in the flows behind the gas-dynamic stabilizer. Bulletin of the University of Samara. Aerospace technology and engineering 15 (4) 162-173
[2] Tretyakov V V 2006 Calculation of the distribution of droplet-liquid and vaporous fuel in the three-stage module of the combustion chamber Vestnik SGAU 2 (10) 136-141.
[3] Heister S D 2011 Plain Orifice Spray Nozzles Handbook of Atomization and Sprays Theory and Applications 643-664 Editor N.Ashgriz, Springer 935