Modeling of atomization and distribution of drop-liquid fuel in unsteady swirling flows in a combustion chamber and free space

To cite this article: A A Sviridenkov et al 2018 J. Phys.: Conf. Ser. 980 012042

View the article online for updates and enhancements.

You may also like

- Heat transfer simulation of unsteady swirling flow in a vortex tube
  S V Veretennikov, Sh A Piralishvili, O A Evdokimov et al.

- Investigation of the Plunging Pressure Pulsation in a Swirling Flow with Precessing Vortex Rope in a Straight Diffuser
  S Muntean, C Tras, A I Bosioc et al.

- A benchmark test case for swirling flows: design of the swirl apparatus, experimental data, and numerical challenges
  R F Susan-Resiga, C Popescu, R Szakal et al.
Modeling of atomization and distribution of drop-liquid fuel in unsteady swirling flows in a combustion chamber and free space

A A Sviridenkov¹ P D Toktaliev¹ V V Tretyakov¹
¹Central Institute of Aviation Motors named after P.I. Baranov, Moscow 111116, Aviamotornaya 2, Russia

E-mail: tretjak@ciam.ru

Abstract. Numerical and experimental research of atomization and propagation of drop-liquid phase in swirling flow behind the frontal device of combustion chamber was performed. Numerical procedure was based on steady and unsteady Reynolds equations solution. It's shown that better agreement with experimental data could be obtained with unsteady approach. Fractional time step method was implemented to solve Reynolds equations. Models of primary and secondary breakup of liquid fuel jet in swirling flows are formulated and tested. Typical mean sizes of fuel droplets for base operational regime of swirling device and combustion chamber were calculated. Comparison of main features of internal swirling flow in combustion chamber with unbounded swirling flow was made.

1. Introduction
Operating process in combustion chamber has multidisciplinary nature, it includes gas dynamic, phase transition, reaction kinetics and others subjects. Correct numerical modeling of combustion process should reproduce physical features of the flow and should be based on verified models in each of these fields of study. Among them, atomization and propagation of drop-liquid fuel in turbulent flow, typically swirling flow.

The aim of this work is to study by simulation and experiment unsteady swirling flows in the combustion chambers of gas turbine engines and the effect of nonstationarity of the flow on the atomization and distribution of fuel in the primary volume of the combustion chamber. The object of these studies is the swirling frontal device of the model combustion chamber with a gas-dynamic stabilizer mounted at the inlet [1]. The scheme of device is shown at figure 1. Main air enters into primary volume of combustion chamber through blade swirler by annular gap with height \( h_0 = 20 \) mm, additional air is injected by transverse jets at the end of cylindrical stabilizer, fuel is supplied in section between swirler and additional air holes.
2. Numerical simulation

The method for calculating velocity fields is associated with modeling the unsteady nature of airflow and fuel droplets. First, the air velocity fields are calculated from the steady-state Reynolds equations solution with imposed k-ε turbulence model at effective Reynolds numbers of the air flow $Re_{eff}>300$ (based on $h$ - annular gap of main channel). In this range, the flow under consideration, as can be shown both from calculations and experiments, is unstable, and repeated patterns of velocity distributions occur at different stages of the iterative calculation process. Random samples of these structures are accepted as instantaneous fields of gas flow velocity, and the series of these instantaneous fields is a model of unsteady flow. This procedure is similar to described in [2]. At the second stage, instantaneous concentration fields corresponding to the instantaneous velocity fields are calculated. And, finally, in the third stage, the instantaneous concentration fields are averaged, resulting in concentration fields that are accepted as averaged. The averaging of the instantaneous velocity and concentration fields is realized without weights. Detailed comparison of the calculated and experimental profiles of velocities can be found in [1] and it shows acceptable agreement. An example of the calculated axial velocity field in the longitudinal section of the combustion chamber with gas dynamic stabilizer obtained using the described approach for regime with swirl intensity $S_w=1.0$ (blades angle 45°) and dimensionless transverse air jets velocity $V_j=1.0$ (relative to $u_0=20$ m/s) are shown at figure 2.

The obtained velocity field used for modeling the propagation of a fuel jet into the swirling air flow and for modeling fuel jet disintegration into droplets. This simulation includes three stages: determining the shape of the liquid jet by solving the equations of mass and momentum conservation in a curvilinear coordinate system associated with the jet surface, using the primary disintegration model of the fuel jet, using the model of further fuel fragments disintegration and droplets formation.
The modeling of disintegration processes, details see [3], is carried out within the framework of the linear stability theory. It is assumed that the primary disintegration of the jet is caused by its instability in the field of aerodynamic forces of the air flow. In this case perturbations acting on the jet lead to the formation of waves on its surface, the amplitude of which increases and when a certain "critical" value is reached, the jet breaks into fragments. Similarly, the decomposition of fragments and droplets is occurred. We note that the technique for calculating the formation and breakup of liquid jets, described in [2], makes it possible to describe the shape of the jet in the flow of atomizing air, to determine the place of its breakup, and to obtain data on the droplet size in the spray cone. In the cases under consideration, the disintegration of the jet occurs at a distance of 3d to 4d (d - injection holes diameter) from the nozzle edge, and the average spray angle is 30° to the axis of the stabilizer. The average median sizes of the formed droplets are for the regime with $S_w = 0.6$, $V_j = 0.6$ $D_m = 40 - 45 \mu m$, and for regime with $S_w = 1.0$, $V_j = 1.0$, $D_m = 29 \mu m$. These data are the basis for numerical calculations of droplet motion and fuel distribution in the working volume of the combustion chamber. Note that the size of droplets formed in the flow for the second regime is in good agreement with the experimental data [1]: $D_{m, exp} = 30 \mu m$.

The method for calculating the fields of droplet fuel concentrations is based on models of the breakup of fuel jets onto droplets, droplets fragmentation in the air flow and analysis of the motion of individual droplets. It is assumed that the droplets sizes distribution obeys the Rozin-Rammiller law with exponent $n = 3$: $F=1\exp[-(D/D_{m})^3]$, and the droplets distribution in the circumferential (along the angle $\phi$) and longitudinal (along the angle $\psi$) directions - the normal law. Here $F$ is the relative fraction of droplets whose diameter is less than $D$, $D_{m}$ is the average median diameter of the droplets. The initial release velocities $V_0$ of the droplets are given by the velocity module $V_0$ and the values of the longitudinal and circumferential angles: $\phi_l$ and $\psi_l$. The initial coordinates of the entry points of droplets are determined from the solution of the problem of the breakup of the fuel jet injected into the air stream, the release angles - from experimental atomization data in the free space, the initial droplets temperature is assumed equal to 300 K. The mean median diameter of the droplets in the spray cone in the calculations varied depending on the regime from 30 to 50 $\mu m$, the fuel consumption is assumed equal to $G_j = 2.5$ g/s. Fuel - kerosene TS-1.

Figure 3 represents the results of the calculations of the average distributions of the mass concentrations of droplets obtained for regime $S_w = 1.0$, $V_j = 1.0$. The mass concentration of droplets $C_k$ is referred to the air density, fuel injection was carried out in the radial direction.

![Figure 3. Average distribution of fuel concentration $C_k$ in the longitudinal section of combustion chamber. $S_w = 1.0$, $V_j = 1.0$.](image)

From the presented results it can be seen that significant part of the fuel falls on the outer wall of the chamber ($Y/h_0 = 0.6$). Analogous results were obtained for other operational regimes with sufficient high $V_j$ ($V_j>0.2$). We note that this phenomenon is determined exclusively by the unsteady nature of the flow. Similar results were obtained for the distribution of vapor fuel. The importance of taking into account transport processes nonstationarity is also evidenced by a comparison of the distributions of droplets fuel concentrations obtained in calculations with steady and unsteady...
approach. Figure 4 and figure 5 show a comparison of the droplets concentrations numerical profiles with experimental data for steady and unsteady calculations. It can be seen that the results of unsteady calculations are consistent with the experimental data in the entire flow region, while steady results are only in the internal region. Thus, the calculation from the steady model shows that the fuel drops in the cross section at \( d = 30 \) mm (here \( d \) is the distance counted from the end of the stabilizer, section correspond to dimensionless axial coordinate \( X/h_0 = 3.2 \) at figure 2, origin of Cartesian coordinate system placed at device symmetry axis and coincided in axial direction with frontal device outer vertical wall, see 2 at figure 1) are completely absent at radial distances greater than 20 mm, and in the section \( d = 50 \) mm at distances greater than 30 mm. At the same time, the calculation with the unsteady model demonstrates complete agreement with the experimental distributions. Thus, the presence of droplets in the vicinity of the walls of the flame tube of the combustion chamber observed in experiments can be explained only with unsteady flow model. It should be noted, that for the presented direct comparison of the results of experiments and calculations, a special method for processing the calculated concentration distributions has been developed: droplets mass concentrations were recalculated into volume one and local fuel distributions were averaged along the beam of the laser knife, in accordance with the experimental procedure. In addition, to investigate the process of breakup of fuel jets into droplets, a special technique was developed, based on visualization of the fuel cone in pulsed light. The experiments showed that the fuel jet in the considered regimes performs rotational-vibrational motions with an amplitude of spatial oscillations approximately equal to the output section of the swirler. The visually observed shape of the jet in non-pulsed lighting represents only the averaged distribution of droplets in a fuel-air cone.

![Figure 4](image1.png)

**Figure 4.** Average distribution of fuel concentration in the cross sections of the spray cone, \( X = 30 \) mm, \( Sw = 0.6, V_j = 0.6 \). Steady, unsteady numerical models, experimental data.

![Figure 5](image2.png)

**Figure 5.** Average distribution of fuel concentration in the cross sections of the spray cone, \( X = 30 \) mm, \( Sw = 1.0, V_j = 1.0 \). Steady, unsteady numerical models, experimental data.

In this paper, the study of the characteristics of the mixture formation is also carried out for a fixed air flow regime: \( Sw = 1.0, V_j = 1.0 \) with azimuthal fuel injection angle variation \( \phi_k \). Calculations were carried out for three variants of fuel supply angles: \( \phi_k = 0, \pm 45^\circ \). It has been found that the degree of expansion of the spraying cone is substantially dependent on the direction of fuel jets and the swirl of the main air flow. The average size of the droplets formed in the flow also depends on these parameters. Thus, at \( \phi_k = + 45^\circ \) (in the direction of swirl of the main air flow), the spraying cone considerably narrows, and at \( \phi_k = - 45^\circ \) (opposite to the direction of swirl of the main air flow) remains practically unchanged. In addition, in the case of \( \phi_k = + 45^\circ \), there is no significant change in the
droplets size, while at the injection angles of 45° there is a reduction in the droplet sizes from 50 to 30-40 μm.

3. Experimental measurements

For verification of jet breakup and droplets atomization mathematical model and testing of pressure atomizer performance experimental research under atmospheric conditions was performed. Experimental rig consist of air-fuel feed system, cylindrical operational region with mounted atomizer, diagnostic equipment and optically transparent window, exhaust section. High-speed video recording can be done for flow visualization, characteristics of disperse phase can be measured by small-angle laser scattering technique. Velocity field for single phase flow can be extracted by particle image velocimetry (PIV). Details of testing facility and measurement technique can be found in [4-6].

It should be emphasized that the presence of unsteady flow could principally change the characteristics of fuel jet breakup. Thus, measurements of the spraying cone characteristics showed that three maxima with droplet sizes in the range 20 ÷ 100 μm are distinctly observed in the concentration distribution near the axis (see figure 4, 5). Note that this result is obtained for free space jet. As for the combustion chamber, comparison between numerical and experimental data is complicated due to droplets deposition on the walls of the combustion chamber. It's difficult to organize local mass concentration measurements in the internal combustion chamber flow. Therefore, conclusions about the features of fuel distribution in the combustion chamber can be made easier by numerical calculations. It should be noted that the presence of flow boundaries significantly affects not only the hydrodynamics of the flow, but also all the characteristics of the atomization of the fuel. Experiments with fuel jet in air flow were performed to find main features of internal two-phase flow behind the frontal device of combustion chamber. Figure 6 shows photographs of the fuel jet injected into the swirling air flow in the combustion chamber.

![Figure 6](image_url)

**Figure 6.** Photos of the fragment of the jet (a) and the whole jet (b) in the combustion chamber. ΔP_a = 3 kPa, G_f = 2.5 g/s. Flash lighting.

It can be seen from the figures that the presence of flow boundaries substantially changes the nature of the propagation of the jet and the mechanism of its breakup. Thus, in the breakup of a jet inside a chamber, waves formed on its surface are caused by the Rayleigh instability. The amplitude of these waves when interacting with a swirling air stream increases and the jet disintegrates into individual drops. At the same time, the jet does not make noticeable oscillations in the chamber, which is apparently due to the absence of significant large-scale precession-type motion in the combustion chamber.

Thus, the structure of the jet behind the swirler in the combustion chamber is radically different from its structure when it releases into free space. In the space confined to the walls of the chamber,
the fuel jet does not make significant fluctuations, which are observed with similar fuel and air mass flow rates in the free space.

4. Conclusions
As can be seen from numerical and experimental research, distributions of drop-liquid phase in swirling flow in combustion chamber for typical regime conditions have essentially unsteady nature. This internal swirling flow is more receptive to perturbations than free space one. Some features of these flow, such as fuel droplets deposition on flame tube wall, can't be modeled in numerical simulation with simple steady approach. Additionally, special treatment need to be incorporated in numerical procedure to describe two-way coupling between continuum and discrete phases. Numerical studies of presented scheme of combustion chamber with gas-dynamic stabilizer show, that Sauter mean diameter of fuel droplets for wide range of dimensionless parameters $S_w$ and $V_j$ not exceed 50 $\mu$m for cold flow. Experimental research of liquid fuel atomization, which was held for verification of proposed mathematical models, show satisfactory agreement with numerical data.

Acknowledgments
Research has been supported by the RFBR grant No. 17-01-00213.

References
[1] Tretyakov V V, Sviridenkov A A and Toktaliev P D 2017 Russian Aeronautics 3 106-12
[2] Armsfield S and Street R 1999 Journal of Computational Physics 153 660-5
[3] Tretyakov V V and Sviridenkov A A 2016 Herald of the Samara State University. Aerospace technique, technology and machinery 15 (4) 162-73
[4] Tretyakov V V and Sviridenkov A A 2011 Proc. Int. Conf. Heat and Mass Transfer and Hydrodynamics in Swirling Flows vol 1 (October Moscow)
[5] Tretyakov V V 2006 Herald of SSAU 2 (10) 136-41
[6] Krasheninnikov S Yu, Mironov A K, Toktaliev P D, Zakharov D L and Maslov V P 2014 Fluid Dyn. 49 (1) 51–62