Computational model for fuel component supply into a combustion chamber of LRE

To cite this article: A V Teterev et al 2017 J. Phys.: Conf. Ser. 937 012054

View the article online for updates and enhancements.
Computational model for fuel component supply into a combustion chamber of LRE

A V Teterev\textsuperscript{1,2}, P A Mandrik\textsuperscript{1}, L V Rudak\textsuperscript{1} and N I Misyuchenko\textsuperscript{1}

\textsuperscript{1}Belarusian State University, Nezavisimosti Av., 4, 220030 Minsk, Belarus
\textsuperscript{2}National Research Nuclear University “MEPhI”, Kashirskoe Av., 31, 115409 Moscow, Russia

teterev@bsu.by

Abstract. A 2D-3D computational model for calculating a flow inside jet injectors that feed fuel components to a combustion chamber of a liquid rocket engine is described. The model is based on the gasdynamic calculation of compressible medium. Model software provides calculation of both one- and two-component injectors. Flow simulation in two-component injectors is realized using the scheme of separate supply of "gas-gas" or "gas-liquid" fuel components. An algorithm for converting a continuous liquid medium into a "cloud" of drops is described. Application areas of the developed model and the results of 2D simulation of injectors to obtain correction factors in the calculation formulas for fuel supply are discussed.

1. Introduction

Fuel is supplied to a mixing chamber by means of injectors. The main requirement for injectors is to provide, as much as possible, a finer and more uniform spray of fuel at a sufficiently small pressure drop on injectors \cite{1, 2}. Usually, two main types of injectors are used in rocket engines: jet and centrifugal. The application of injectors of both types is possible. To cool combustion chamber walls, slit injectors are also used as a kind of jet injectors having a slot-like shape of the outlet. A jet injector is an outlet hole in the engine chamber head or a separate element shaped as a tube that communicates a fuel or oxidizer cavity with the inner volume of the combustion chamber. The main advantage of jet injectors is a large capacity of the head with jet injectors. In engines with one-component injectors, to ensure a good mixture formation, fuel and oxidizer injectors should uniformly alternate. In general, one-component injectors on the mixing head of the chamber are placed in concentric circles. In this case, either the belts of oxidizer and fuel injectors or the injectors of oxidant and fuel alternate in one concentric circle. Two-component injectors can be placed according to any scheme, but more often they are placed in concentric circles.
When designing a new engine, injectors must be calculated in order to ensure necessary fuel consumption. Numerical simulation of flow inside injectors and engine chambers is usually performed either in the project approximation [3-5], when analytical dependences and formulas are used, or in the gasdynamic approximation [6-8], when finite-difference methods are adopted to solve systems of partial differential equations.

2. The physical model of the fuel supply through jet injector

The developed model of injector calculation is intended for closed-type engines, when one or both fuel components pass through a gas generator. The scheme of organization of the working process in the combustion chamber of such engines is "gas-gas" or "gas-liquid". I.e., at least, one fuel component is in the gas phase. The mathematical model is a system of gasdynamic equations for compressible medium with given initial conditions and corresponding boundary conditions. These are the non-flow conditions on all rigid walls of an injector and the inflow conditions at the fuel tank inlet and the outflow conditions at the combustion chamber outlet. Input parameters for calculation are a selected type of fuel or oxidizer or both components, gasdynamic parameters and a mixture composition in the combustion chamber, fuel component consumption and injector geometries. The shape of the channel edge of a gas jet injector at its inlet can be sharp, radically rounded or with a chamfer. In addition, the injector can be cylindrical or cone-shaped.

When simulating the injectors supplying a fuel component into the liquid phase, the most complex flow occurs in the zone where its continuity undergoes discontinuity. In this zone, a lot of drops are born that seem to emerge from the continuous medium region and to continue moving in the form of discrete formations with definite physical characteristics. Based on the parameters of the continuous medium in the injector, the model describing this process should form a granulometric composition of drops, their thermodynamic parameters and should make a smooth transition of the flow description from the continuous medium approximation to the discrete one. A continuous medium of the fuel liquid component is transferred into a set of drops at a time when the medium density falls below a certain critical value. As the model for the interaction of drops and the surrounding gas flow, we used the Sand Bag model that was successfully used in describing the dynamics of condensed particles with a gas stream of combustion products in the Laval nozzle [9].

When constructing mathematical and computational models describing the flow of liquid or gaseous components of fuel into the combustion chamber of a liquid rocket engine, we will proceed from the following basic assumptions:

- Motion of liquid and gaseous media is described by non-stationary gasdynamic equations written for a multicomponent compressible medium.
- The transition of the liquid jet into a drop-like state is carried out depending on its parameters at the injector section or inside the combustion chamber.
- The interaction of the emerging droplet liquid component with the gaseous medium in the combustion chamber is described by a non-stationary system of ordinary differential equations.
- The destruction of a liquid jet inside the combustion chamber is modeled using the concentration method, which makes it possible to describe contact boundaries in multiply connected regions.
- Maintaining the initially set pressure drop on the injector and the pressures in the fuel tank and combustion chamber is accomplished by setting the appropriate boundary conditions.

3. Mathematical model of motion of continuous media in injectors

The mathematical model constructed on these assumptions is a nonstationary system of partial differential equations describing the gas-dynamic flow of a multicomponent multiphase compressible medium. The conservative form of the non-stationary Euler equations in the cylindrical coordinate system \((r, \varphi, z)\) can be written as follows:
\[
\frac{\partial [rU]}{\partial t} + \frac{\partial [rF(U)]}{\partial r} + \frac{1}{r} \frac{\partial [rG(U)]}{\partial \varphi} + \frac{\partial [rH(U)]}{\partial z} = S(U)
\]  

(1)

Here the conservative state vector \( U \) is defined as

\[
U = (\rho, \rho v_r, \rho v_\varphi, \rho v_z, E)
\]

(2)

The flux vectors \( F(U), G(U) \) and \( H(U) \) and the source term \( S(U) \) are

\[
F(U) = \begin{pmatrix} \rho v_r \\ \rho v_r v_\varphi \\ \rho v_r v_z \\ [E + p] v_r \end{pmatrix}, \quad G(U) = \begin{pmatrix} \rho v_\varphi \\ \rho v_\varphi v_r \\ \rho v_\varphi v_z \\ [E + p] v_\varphi \end{pmatrix}, \quad H(U) = \begin{pmatrix} \rho v_z \\ \rho v_z v_r \\ \rho v_z v_\varphi \\ [E + p] v_z \end{pmatrix}
\]

(3)

\[
S(U) = (0, p + \rho v_\varphi^2 - \rho v_r v_\varphi, 0, 0)
\]

(4)

To close system (1)-(4), we use the caloric and thermal equations of state, as well as the expression for the total density of a medium in terms of the density of its components:

\[
p = p(\rho, \epsilon) \quad T = T(\rho, \epsilon), \quad \rho = \sum_i \rho_i
\]

(5)

The mathematical model of the interaction of the droplet fraction of the fuel components with the gaseous medium with respect to flows in the combustion chamber and Laval nozzle is described in [9], constructed on the basis of the model set forth in [10].

In addition, for multiphase regions condition

\[
p_g = p_l
\]

(6)

where \( p_g \) and \( p_l \) are the gas and liquid pressures, respectively. An iterative process ensuring that condition (6) is satisfied allows us to correct the volume concentrations of the liquid and gas components of the medium in the calculated cell.

In the case of the two-dimensional version of the model, the same system of equations in the cylindrical coordinate system \((r, z)\) is used.

4. Statement of boundary conditions

The task statement is completed by setting the initial conditions and the corresponding boundary conditions. These are the non-flow conditions on all rigid walls of an injector and the inflow conditions at the fuel tank inlet and the outflow conditions at the combustion chamber outlet. Input parameters for calculation are a selected type of fuel or oxidizer or both components, gasdynamic parameters and a mixture composition in the combustion chamber, fuel component consumption and injector geometries. The shape of the channel edge of the gas jet injector at the inlet can be sharp, radically rounded or with a chamfer. In addition, the injector can be cylindrical or conical in shape.

The boundary conditions in subsonic flow tasks play an important role, since they are a constant source of flow disturbances in the computational domain. In order to maintain a constant pressure drop between the fuel tank and the combustion chamber, different boundary conditions were tested at the tank inlet and at the chamber outlet. In calculations, the following boundary conditions can be set:

- inflow at a given mass flow rate

\[
\dot{m} = \text{const}, \quad \rho_{\text{bd}} = \rho_N + \Delta \rho, \quad v_{\text{bd}} = v_N + \Delta v, \quad E_{\text{bd}} = E_N + \Delta E
\]

(7)

- outflow at a given pressure

\[
p_{\text{bd}} = \left(p_N + \rho_{\text{inj}}\right) \times 0.5, \quad v_{\text{bd}} = v_N - 3v_{N-1}
\]

(8)

- outflow with a given hole
\[ r_{\text{hole}} = (2 + 3) r_{\text{inj}} \]  \hspace{1cm} (9)

outflow with a semi-transparent hole

\[ r_{\text{hole}} = (2 + 3) r_{\text{inj}}, \quad m_{\text{out}} = m_{\text{inj}} \]  \hspace{1cm} (10)

At conditions (7) and (8), the index \text{bnd} means the boundary between the calculated cell N and the cell N+1. At all formulas the subscript inj means the values in the injector.

The application of conditions (7) usually changes the fuel tank pressure and condition (9) as a rule decreases the combustion chamber pressure. The injector pressure drop can be most accurately maintained under boundary conditions of form (8) and (10).

5. Simulation results

To calculate the flow velocity of fuel liquid components from jet injectors, we use the theoretical flow velocity of incompressible liquid from the hole, which is determined by the formula [1]:

\[ w = \left( \frac{2 \Delta p_{\text{inj}}}{\rho_{\text{liq}}} \right)^{1/2} \]  \hspace{1cm} (11)

where \( \Delta p_{\text{inj}} = (p_{\text{inj}} - p_{\text{ch}}) \) is the pressure drop at the injector, \( \rho_{\text{liq}} \) is the liquid density. In this case, the liquid flow rate through the injector with the hole cross-sectional area \( F_c \) is given by equation

\[ m_{\text{inj}} = \mu_{\text{inj}} F_c \sqrt{2 \Delta p_{\text{inj}} \rho_{\text{liq}}} \]  \hspace{1cm} (12)

where \( \mu_{\text{inj}} \) is the flow rate coefficient taking into account the jet constriction and the flow velocity decrease in comparison with the theoretical one.

The quality of the factory produced injectors is checked by hydraulic tests, conducting the so-called "spillage" with water [2]. Figure 1 shows the dynamics of water consumption through a jet injector with a diameter of 5 mm at a pressure of 250 atm in the combustion chamber and at a pressure drop of 10 atm. It also illustrates the graph fragment, according to which the frequency of consumption oscillations is determined to be approximately 40 kHz, which corresponds to the frequency of pressure pulsations on the reservoir symmetry axis that come from the boundary, at which the fuel is supplied. The dash-dotted line corresponds to the water consumption calculated from equation (12) at \( \mu = 1 \).

![Figure 1. Dynamics of water consumption through the injector.](image)

To identify the factors that affect the process of drop formation, an illustrative calculation was performed by the model with a contact boundary. A jet of liquid (water) flowing from the jet injector into the region of the cold compressed gas (air) was simulated. The initial gas parameters were:
temperature – 298 K, density – 292 kg/m³, pressure – 250 atm. The pressure drop across the injector was Δр = 35 atm and its diameter was 0.5 mm. The flow dynamics can be seen from the fragments of density fields depicted in Figure 2 for three time moments. The time moment \( t = 1.3 \cdot 10^{-4} \) s in Figure 2(a) corresponds to the beginning of the process of destruction of the already formed "large" drop, which is an expanded front part of the jet that propagates into the gas reservoir. By this time moment, the jet penetrates to a distance of ~ 3 mm from the injector section. Destruction begins at the back surface of the drop directly near the cylindrical jet surface. By the time \( t = 1.73 \cdot 10^{-4} \) s in Figure 2(b), the jet penetrates to a distance of ~ 6 mm into the reservoir, and a "crack" spreads approximately to the half the length of the drop. It can be seen that the stern part of the separating part of the drop continues in turn to disintegrate, thus disintegrating into smaller formations spraying in the ambient air. By the time moment \( t = 2.41 \cdot 10^{-4} \) s in Figure 2(c), when the jet penetrates to a depth of 7 mm, the side part of the drop is almost completely separated from the central part of the drop representing the initial jet.

![Figure 2](image-url)

**Figure 2.** Fragments of the density fields at different time moments.

The described drop destruction process can be explained on gasdynamic grounds and, apparently, it is primarily associated with a vortex flow arising in the immediate vicinity of the side peripheral region of an elongated drop. The vortex dynamics on the side surface of the jet can be seen in the velocity vector fields. Developed vortex flow, arising in the lower part of the jet, intensively destroys a water flow. Similar destruction was observed in simulating the destruction in dense layers of the atmosphere of water formations that occur when small comets fall [11]

To calculate the rate of gas-generating gases through a jet injector, the expression [1] is used:
\[
W = \left( \frac{2\gamma}{\gamma - 1} RT_{\text{in}} \left( 1 - \left( \frac{P_{\text{out}}}{P_{\text{in}}} \right)^{\frac{\gamma - 1}{\gamma}} \right) \right)^{1/2}
\]

(13)

where \( R, T_{\text{in}} \) is the gas constant and the gas temperature in front of the injector respectively and the flow rate is given by formula

\[
\dot{m}_{\text{ij}} = \mu_{\text{ij}} w F_c \rho_{\text{out}}
\]

(14)

In Figure 3, the lines represent the velocities calculated by (13) as a function of pressure drop for two gas temperatures and two pressures in the combustion chamber.

**Figure 3.** Hydrogen velocity at the injector outlet.

Here, the points stand for the numerical simulation results of hydrogen supply through an injector with a diameter of 5 mm and a length of 20 mm. The dotted lines with a corresponding flow rate coefficient are drawn through the points.

Figure 4 (left) shows the time dependences of the oxygen velocity at the injector outlet (left) and of the mass flow rate (right). The straight lines correspond to the mass flow rate calculated according to equation (13) with the corresponding coefficients.

**Figure 4.** Oxygen velocity and flow rate at the injector outlet.
The oxygen velocity growth with increase in the pressure difference continues until it reaches the sound speed. At this moment, the “reservoir” flow is "locked", and a further increase in the difference does not affect the fuel consumption through the injector [12]. Such flow is shown in Figure 5, where the temperature fields are shown on the left and the density fields – on the right. They are displayed on the monitor screen during the calculation. In this version of calculation, the oxygen pressure in the tank was 250 atm, and the pressure in the combustion chamber was about 10 atm; the sound speed and the limiting flow rate occurred even at a combustion chamber pressure of 100 atm. But even at a pressure of 150 atm, the flow rate was almost maximum, although the outflow velocity was far from the sound speed. It should be noted that at a pressure of 50 atm and more, a weakly expanding jet outflows from the injector, and only with a further pressure decrease its shape is similar to the shape underexpanded jet from the Laval nozzle. A sharp expansion of the jet at the combustion chamber inlet causes strong cooling, and as a result of acceleration, the temperature reaches 500 K at an initial temperature of 300 K. The depicted flows can appear at full-flow start.

![Temperature and density fields at a large pressure drop on the injector.](image)

**Figure 5.** Temperature and density fields at a large pressure drop on the injector.

6. Conclusion
The developed model of flow calculation in injectors is designed to solve different simulation problems. First, it can be used in designing injectors for parametric calculations. This will allow one to more accurately determine sizes and parameters of designed injectors in comparison with analytical approximation calculations. In addition, this simulation makes it possible to investigate the flow features of fuel components within the unsteady flow regime and how it is influenced by gas parameter pulsations in the combustion chamber. Second, based on multiparametric calculations, it is possible to tabulate correction factors for the analytical expressions, in terms of which the fuel supply into the combustion chamber is calculated. This in turn will improve the simulation accuracy of a rocket engine. Third, with the appropriate software modification, this model can be directly used in simulating the flow in the combustion chamber. The essence of such a modification is to calculate injectors and combustion chamber on various three-dimensional grids. In this case, the calculation results of injectors will be the boundary conditions for fuel component inflow into the combustion chamber; the calculation results of the combustion chamber will in turn be the boundary conditions for
calculation of flow in injectors. Such a calculation scheme will allow revealing the influence of fuel supply pulsations in the combustion chamber on the low-frequency instability of combustion.

Currently, the developed software is designed for flow simulations in jet injectors, the results of which will allow one to determine correction factors in the analytical expressions for fuel supply. As parameters, we use fuel consumption, thermodynamic parameters of fuel and oxidizer, as well as gas parameters in the combustion chamber.

References
[1] Dobrowolski M 2005 Liquid rocket engine (Moscow) p 488
[2] Barrere M A, Jaumotte B, Fracis de Veubeke and Vandenkerckhove J 1960 Rocket Propulsion (Amsterdam, London, New York and Princeton)
[3] Egorychev V S 2011 Calculation and design of a mixture in a liquid rocket engine (Samara: Izd. SGAU) p 108
[4] Egorychev V S 2009 Method of design calculation of centrifugal emulsive duplex fuel nozzle Bulletin of the Samara state aerospace university 19 248-52
[5] Egorychev V S 2011 Method of design calculation jet-centrifugal gas-liquid duplex fuel nozzle with internal mixing Bulletin of the Samara state aerospace university 27 199-204
[6] Salich V L 2011 The numerical modelling of processes in combustion chambers of low thrust rocket engines Bulletin of the Samara state aerospace university 27 120-5
[7] Matyunin O O, Bachev N L, Bulbovich R V 2015 Numerical simulation of the flow in two-component fuel injector Bulletin of PNIPY Aerospace Engineering 43 19-34
[8] Strokach E A, Borovik I N 2016 Numerical simulation of the process of kerosene spraying with a centrifugal injector Bulletin of MSTU. Ser. "Mechanical Engineering" 3 37-54
[9] Teterev A V, Mandrik P A, Rudak L V and Misuchenko N I (2017) Effect of the condensed particles on a flow of combustion products in a Laval nozzle IOP Conf. Series: Journal of Physics: Conf. Series 788 012034
[10] Teterev A V (1999) Cratering model of asteroid and comet impact on a planetary surface Int. J. Impact Eng. 23 921-932
[11] Teterev A V, Mandrik P A, Rudak L V 2009 The passage of large meteoroids through planetary atmospheres: theory and numerical simulation Proc. XVII Int. Scientific and Technical Conference "Applied problems of mathematics and mechanics" (Sevastopol)
[12] Loiteyanskii L G 1987 Mechanics of liquid and gas (Moskow, Nauka) p 840