Journal of Engineering Physics and Thermophysics, Vol. 79, No. 4, 2006

SIMULATION OF THE INTERNAL DYNAMICS OF SOLID-FUEL ROCKET ENGINES ON THE BASIS OF THE STAR-CD SUITE

K. N. Volkov, S. V. Denisikhin, and V. N. Emel'yanov

UDC 532.529

Flows in the channels of solid-fuel charges with cross sections having different views in plan and flows in the prenozzle volume and the nozzle unit of a solid-fuel rocket engine have been simulated on the basis of the STAR-CD suite for different types of charges and different designs of the input part of the engine nozzle. The influence of the compressibility, turbulence, geometric factors, and flow rate on the distributions of gasdynamic parameters in the working region of the engine has been investigated.

Introduction. The development of numerical methods, the progress in computers, and the appearance of modern computer technologies gave birth to a new investigation method — computational simulation, which represents a complex approach to the theoretical study of physical processes and effects and involves construction of mathematical models (systems of equations), development of numerical methods for solving these equations, and computer realization of the indicated methods.

An automated engineering analysis performed on the basis of numerical methods represents an integral part of the process of designing different-application products. Specialized program complexes (computational suites) are used for this analysis. They make it possible to perform simulation of complex and expensive (for natural experiments) processes. Depending on the set of service functions used for determining the initial data and processing the calculation results given to a user, computational suites are classified as light, medium, and heavy suites. The "weight" of a suite characterizes its potentialities and efficiency.

At present, "heavy" packages represent an integral part of the computer-aided design/computer-aided manufacturing/computer-aided engineering (CAD/CAM/CAE) general chain of development of new technical means and belongs to the category of CAE technologies.

A modern approach to the mathematical simulation of processes occurring in different-application technical mean is based on the use of computational gas hydrodynamics. Among the packages used for solving problems of gas hydrodynamics and their associated problems on the heat load of constructions are the STAR-CD, CFX, and FLUENT suites. Such problems can be also solved with the use of CAE-technology packages. In this connection, of interest is determination of the potentialities of these packages and estimation of the reliability of models constructed with them and the results of numerical investigations carried out on the basis of such models, which can be made by comparison of the results of a numerical simulation with test solutions and the results of a physical experiment.

In the present work, we investigated the possibilities and peculiarities of use of the STAR-CD suite for solving problems on the internal gas dynamics of solid-fuel rocket engines (SFRE). The internal gas dynamics of an SFRE is determined by a large number of interrelated processes of different physical nature occurring in spaces of complex geometric configuration changing with time. To solve the problem on the gas dynamics and the internal ballistics of an SFRE, it is necessary to determine of the distributions of the velocity, pressure, density, and temperature of a flow of combustion product along the length of the inner channels of solid-fuel charges as well as the average parameters of the flow in the prenozzle and nozzle volumes at any instant of time.

Let us determine the conditions on which the program suite being considered should be used the peculiarities of formulation of concrete problems, the most suitable boundary conditions for them, difference schemes, and other parameters necessary for correct solving complex problem, which follow from the solution of simpler problems.

Voenmekh D. F. Ustinov Baltic State Technical University, 1st Krasnoarmeiskaya Str., St. Petersburg, 190005, Russia; email: dsci@mail.ru. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 79, No. 4, pp. 50–56, July–August, 2006. Original article submitted January 14, 2005; revision submitted March 9, 2005.

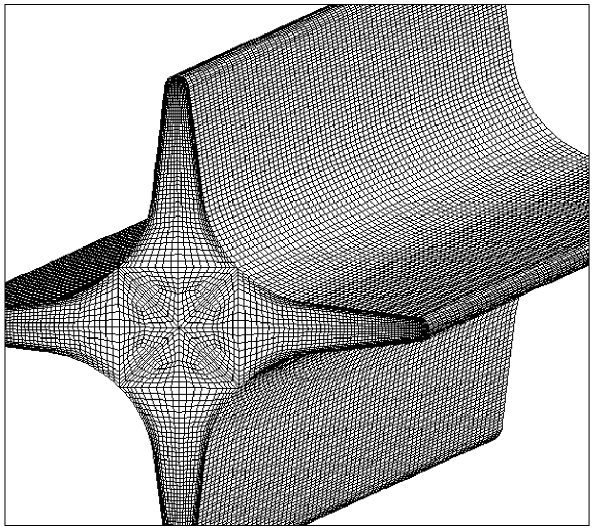


Fig. 1. Computational grid in a channel with a star-like cross section.

The turbulent flow and heat exchange in the spaces being considered are described on the basis of the Navier–Stokes equations averaged in accordance with Reynolds' concept and closed by the equations of the two-parameter k–ε model of turbulence [1, 2]. The velocity and pressure fields are related using the SIMPLE pressure-correction procedure [1, 2]. For discretization of time derivatives, an entirely implicit scheme is used. The discretization of diffusion flows is performed on the basis of the centered finite differences of second order of accuracy, and convective flows are discretized using the QUICK scheme of third order of accuracy.

The calculations were carried out on a Celeron computer (500 MHz) having a 450-Mbyte random-access memory.

Flow in a Channel with Blowing. The inner space of the combustion chamber of an SFRE represents a complex system of channels, the walls of which are formed by the burning surface of a solid-fuel charge and the inner surface of the engine body. The processes of heating a fuel, decomposition of it into components, and chemical interaction of these components proceed in a thin near-surface layer. The mathematical model of a flow of solid-fuel decomposition products in the combustion chamber of the engine being considered is the model of a flow in a channel with penetrable walls that defines the most important process occurring in this case — the mass transfer from burning surface of the charge [3, 4].

The combustion chambers of SFREs have different geometries providing both the solution of arrangement problems and the obtaining of a mass-transfer surface corresponding to their operating conditions. Charges having channels with multislot (star-like) cross sections are widely used in SFREs, in particular in the Space Shuttle, Titan IV, Arian V, and H-1 engines [5–8]. Solid-fuel charges of slot-free design make it possible to gradually change the combustion surface: in this case, the maximum deviation of this surface from its average value does not exceed 2–5%. The use of charges with kerfs (slots) and channels having a star-like cross section makes it possible to change the combustion surface in a wide pressure range [5].

Let us consider a turbulent flow of an incompressible viscous fluid in cylindrical channels with cross sections having round and star-like views in plan. The working body is the combustion products of a solid fuel, which have the following parameters: p = 40 atm, T = 3000 K, G = 306 kJ/(kg·K), Pr = 0.442, and γ = 1.17. The thermophysical properties of the working medium were taken from a reference book. As the characteristic size of a channel R we used its radius (for a round channel) or the elongation of a ray (for a star-like channel) representing the radius of the circle circumferential about a star (distance from the symmetry axis to the farthest point of the ray).

It is assumed that the fluid spreading is symmetric relative to the left boundary of the computational region, at which the condition of flow symmetry is set; and the pressure in the output cross section of the channel $p = 1.013 \cdot 10^5$ Pa.

The rate of gas supply is calculated on the condition that the flow rate of the burnt fuel is equal to the flow rate of the blown gas $\rho_f q_f = \rho_g q_g$, where ρ_f and q_f are respectively the density of the charge and the rate of combus-

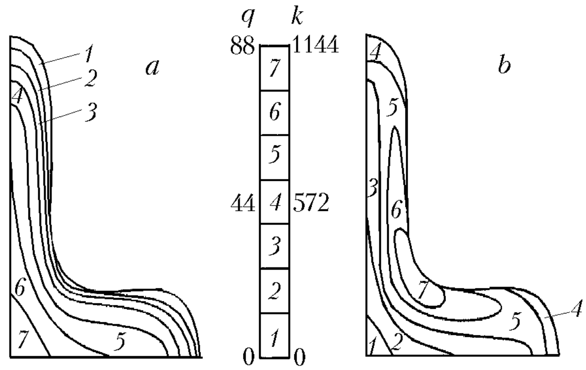


Fig. 2. Contour curves of the velocity (a) and the kinetic turbulent energy (b) of the flow in a channel with a star-like cross section.

tion of the solid fuel, and ρ_g and q_g are respectively the density of the combustion products and the rate of blowing. To find the rate of combustion of the solid fuel, it is necessary to determine the initial pressure in the channel and the rate of blowing. Thereafter, the field of the flow in the flow channel is calculated and the pressure distribution over the combustion surface is determined. Then the parameters of the combustion and blowing are recalculated. The process is repeated until the convergence is attained.

It is assumed that the characteristics of turbulence at the penetrable surface of the channel are constants ($k = 10^{-3}$ m^2/sec^2, $\varepsilon = 10^{-4}$ m^2/sec^3). The results of calculations indicate that a change in these values in a reasonable range (when the intensity of turbulence at the mass-transfer surface changes from 0.1 to 2.5%) does not lead to the appearance of large errors in the results of the numerical simulation.

A stationary solution of the problem is obtained by the method of ascertainment. It is assumed that the gas is at rest at the initial instant of time $t = 0$ ($\mathbf{q} = 0$, $P = 1.013 \cdot 10^5$ Pa, $T = 288$ K).

We investigated the influence of the compressibility and turbulence on the distribution of the flow characteristics with the use of a computational grid having 330,000 cells, which is shown in Fig. 1. It has taken 150 h of the processor time to perform 6700 iterations.

The results of calculations performed for the star-like channel were represented in the form of contour curves of desired functions; which are shown in Fig. 2. In accordance with the results obtained, a flow in this channel can be conditionally divided into three parts. In the initial cross section ($x = 0$), the velocity of the gas is equal to zero. Then, as the gas penetrates through the side walls, the velocity of the gas flow increases. In the region adjacent to the initial cross section of the channel, the compressibility of the gas can be neglected. Here, the rate of mass transfer through the walls is larger than that in the axis mass flow and the lines of flow are pushed aside such that only the gas supplied to this cross section is found near the walls. The distribution of the axial velocity components over the cross section of the channel is described by a cosine function (at $x/R < 30$), and the dependence of the pressure on the coordinate x is parabolic [3, 4].

The velocity of the axial flow increases downstream of the initial cross section. Once the momentum is approximately equal to the axial component of the momentum carried out by the gas flowing from the walls, gas particles of the main flow begin to penetrate to the wall and decelerate there. In this case, there arises a boundary layer that thickens rapidly and fills the cross section of the channel. The viscous interaction of the gas flowing from the walls with the flow formed in the channel, taking place in the next region, influences the distribution of the gas parameters over the whole cross section of the channel.

For an incompressible fluid, the distribution of the gas flow velocity along the axis of the channel is linear in character, which follows from the continuity equation. In the case of a compressible fluid, the flow rate can change with a change in the density of the fluid. The results obtained indicate that, at $x/R < 30$, the influence of the compressibility of the gas on its velocity distribution is negligibly small.

The results of calculations carried out with the use of a k–ε model of turbulence were compared with the exact solution obtain without regard for the turbulence [3, 4]. The data of the numerical simulation indicate that the turbulence influences the gas velocity distribution at $x/R > 32$.

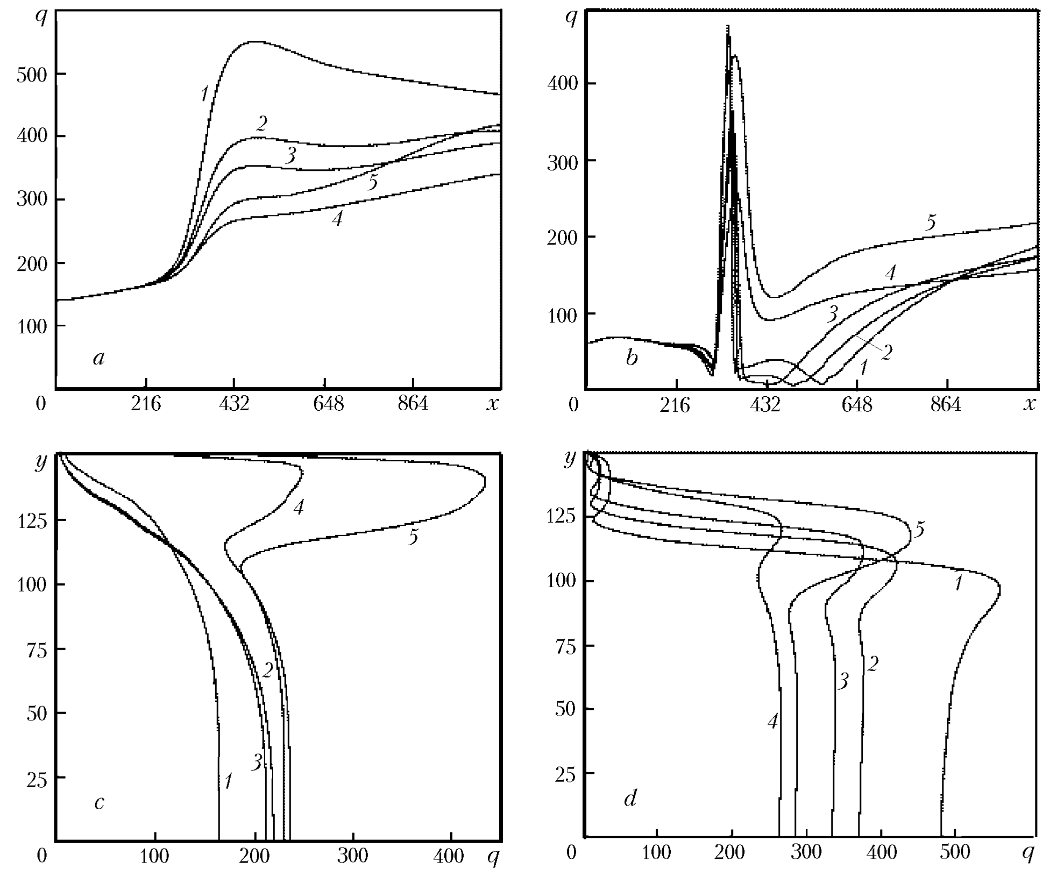


Fig. 3. Velocity distributions of the flow in the cross sections y/R = 0.5 (a) and 9 (b), x/R = 2.30 (c) and 2.73 (d) of a slot channel obtained for different angles of inclination of the compensator: φ = 45° (1), 70° (2), 90° (3), 115° (4), and 135° (5).

As does the gas velocity distribution, the character of convective heat exchange also changes along the length of the channel. In the initial region, the flow is isolated from the convective heat transfer because of the injection of the combustion products. Then the boundary layer grows and the heat flow directed to the wall begins to increase. In the next region, the convective heat exchange is close in character to the heat exchange in the turbulent flow in the main region of the tube [3].

Calculations carried out for a channel with a cross section having a view of a four-beam star in plan (Fig. 2) have shown that the gas velocity distribution in the cross section between the rays of such a star is described fairly well by a cosine function [3, 4]; the larger the length of the ray, the better this conformity.

As the rate of blowing increases, the gas flow in the channel turbulizes (at the cost of the increase in the velocity gradient in the central region of the channel and the turbulence appearing according to the k–ε-model), excepting the near-wall and near-axis regions, and the maximum of the pulsation energy shifts from the wall to the flow. In this case, the intensity of pulsations in the core of the flow begins to decrease and, at the surface of the channel, there arises a near-wall layer (zone of pushing back) in which the turbulence energy is vanishingly small (the flow in this region is close in its properties to a laminar flow).

The results of calculations performed for the channel with a star-like cross section are in fairly good qualitative agreement with the data obtained in [7–9]. The difference between the quantitative estimates obtained by us and the authors of these works can be explained by the difference in the shapes of the cross sections (especially at the sites of connection of rays with the central part of the channel).

Flow in a Channel with a Ring Neck. An axisymmetric channel with penetrable walls is mainly used for simulation of a flow of combustion products in channels of solid-fuel charges used in SFREs. In the general case,

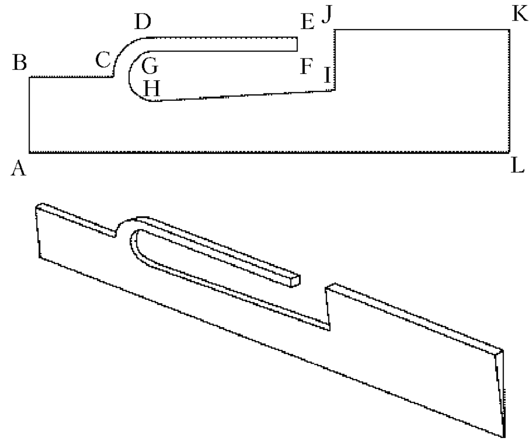


Fig. 4. Flow in a channel with a flush-mounted nozzle.

such a channel can be connected to one or several side channels (slots) inclined about its axis at any angle. In this case, the main gas flow in the channel interacts with the transverse flows of gases entering from the slots [10].

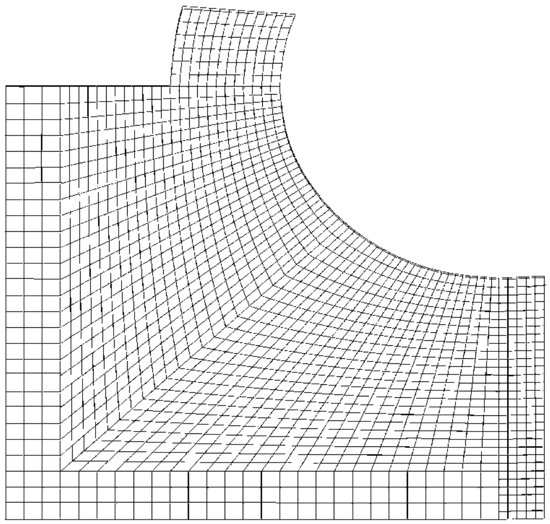
We now consider a turbulent flow of a compressible viscous fluid in a slot channel. It is assumed that the flow velocity distribution at the input boundary is described by a cosine function [3, 4], the maximum velocity of the flow is equal to 150 m/sec, and the intensity of turbulence at the input cross section changes by 1% in 1 cm intervals. The normal-blowing conditions are set at the side surface of the channel. The characteristics of the working medium are identical to the characteristics used in the previous problem, and the rate of blowing at the wall is calculated by the algorithm used in this problem. The pressure at the output cross section of the channel is assumed to be equal to p = 50 atm. The calculations were carried out for different angles of inclination of a slot with the use of a grid containing 28,000 cells. It has been 5 h of the processor time to perform 550 iterations. We investigated the influence of the disposition of the compensating element and the angle of its inclination on the distributions of the velocity, pressure, and temperature of the fluid and the characteristics of its turbulence. The results of calculations are presented in Fig. 3.

A slot in the channel disturbs a flow in it fairly strongly; however, this disturbance is localized in a comparatively small neighborhood of the boundary between the subregions. Outside the neighborhood of this boundary, the flow velocity distribution in the slot is described fairly well by a cosine function [3, 4]. In this case, the pressure at the output cross section changes within 1.5%. If a slot is inclined at any angle about the axis of the channel, near it there arises a conterflow zone; however, in this case too, only the flow near the wall of the channel is transformed significantly.

The cross flow of the gas entering from the slots disturbs the main flow only in the region near the slots, and this disturbances penetrate deeper into the main flow in the case where the slots are at the beginning of the channel where the velocity of this flow is small. The intensity of disturbance of the flow field points to the enhancement of the processes of interphase interaction.

Flow in a Channel with a Flush-Mounted Nozzle. In modern SFREs, the subsonic part of the nozzle is usually flushed into a solid-fuel charge for technological reasons, which leads to the appearance of a gas counterflow and complicates the structure of the main flow [5, 6]. The flush of the nozzle of an SFRE into the prenozzle volume decreases its longitudinal dimension but gives birth to a number of problems associated with a high-temperature flow around the nozzle [10].

Between the surfaces of the channel and the collar (the flush-mounted part of the nozzle) there arises an annular channel. Therefore, the flow over the flush-mounted part of the nozzle of an SFRE arising at the initial instant of its operation can be simulated by the flow in an annular cylindrical channel with penetrable walls [3]. As the fuel burns, the diameter of this channel and the dimension of the annular gap above the flush-mounted part of the nozzle increase. In this case, the kinetic head of the gas flow in the channel begins to exceed the kinetic head of the counterflow of the gas entering from the annular gap and the pattern of flow over the flushed part of the nozzle changes. The break of the flow symmetry is accompanied by the formation of an asymmetric flow directed from the channel to



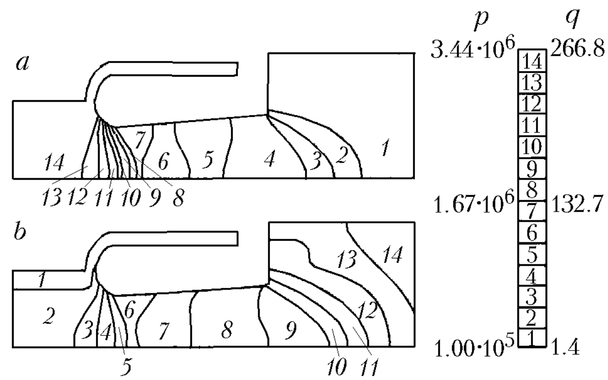


Fig. 5. Fragment of the grid in the prenozzle region.

Fig. 6. Contour curves of the pressure (a) and velocity (b) of the flow in the channel with a flush-mounted nozzle at the instant of time t = 0.027 sec.

the annular region and a flow over the surface of the nozzle cover. For calculating the gas flow in the intake conduit and in the partially flushed nozzle with account for its turn, it is necessary to use three-dimensional models.

We now consider a flush-mounted nozzle, the prenozzle volume of which is formed by linear segments AB, BC, and DE (in Fig. 4). A gas enters the prenozzle volume through the boundary AB and the penetrable walls of the contour bounded by segments BC, CD, DE, EF, and FG. The flushed part of the nozzle (collar) is formed by circular arcs and linear segments. The supersonic part of the flow in this nozzle is described by a third-order polynomial. The flow is assumed to be laminar and the fluid is assumed to be incompressible and viscous. Calculations are carried out for the sector of the nozzle of thickness equal to the size of one computational cell, the opening of whose cone is equal to 5°. A fragment of the computational grid in the prenozzle region is shown in Fig. 5. It is assumed that the distributions of the parameters at the input boundary AB are identical to those obtained in the simulation of the circular channel.

In [1, 2], it was recommended to calculate supersonic flows under soft boundary conditions or the conditions of free outflow; in this case, the variables at the output boundary are determined by extrapolation of unknowns from the inner cells. However, the use of the recommendations of [1, 2] led to the divergence of the iteration process (convergence was not obtained by varying the computational parameters). To determine the boundary conditions in the output cross section of the nozzle, we formulated the model problem on the actuation of a Laval nozzle, which has an exact solution (in the one-dimensional case). Convergence was obtained when the pressure at the output of the nozzle was equal to atmospheric pressure.

In principle, the boundary conditions for the pressure in a flow are more "natural" in the case where this flow is supersonic. For example, the structure of a flow in a Laval nozzle is determined by the ratio between the pressure in the receiver and the ambient pressure. Depending on the pressure difference, the flow at the output of the Laval nozzle is supersonic (continuous acceleration of the gas inside the nozzle) or subsonic (a compression shock is formed inside the nozzle).

The above-listed factors determine the peculiarities of the problem considered. The environment is simulated by an additional nozzle unit with a pressure at the boundaries equal to atmospheric pressure.

The calculations were carried out for different ratios between the rates of the overnozzle and channel flows (0.145 and 0.35%) on a grid containing 28,000 cells. The transient period was fairly large because the wave processes damp slowly due to the geometry of the prenozzle-volume used. It has been possible to solve this problem only in the nonstationary formulation (it was assumed that the gas is at rest at the instant of time t = 0). It has taken 5 h of the processor time to calculate 550 iterations. The results of calculations are presented in Fig. 6.

At Re > 10^4, the flow is adequately described by the model of a nonviscous gas flow, excepting a small near-wall region at the surface of the flush-mounted nozzle where the viscosity is significant. The calculation results were substantially dependent on the geometry of the computational grid near the nose of the flush-mounted nozzle.

The comparison of the results of calculations performed for hot and cold gases has shown that the pressure on the surface of the nozzle being considered is dependent on only the ratio between the rates of the channel and overnozzle flows and is independent of the type of the working body. The larger the flow rate of the gas entering through the overnozzle gap, the higher the pressure at the cylindrical surface of the flush-mounted nozzle.

Conclusions. The mathematical model constructed using the STAR-CD suite allowed us to calculate stationary and nonstationary flows of combustion products of a solid fuel in different regions of the fuel conveying system of an SFRE (in the channel of a solid-fuel charge, in the prenozzle volume, and in the nozzle unit). The role of individual factors and physical mechanisms in the formation of the total pattern of flow in the indicated system has been determined and the main principles of simulation of inner flows with the use of the STAR-CD suite have been developed.

The results of our calculations indicate that the computational procedures involved in the STAR-CD suite allow one to fairly exactly describe the physical processes occurring in an SFRE in the process of its operation. Further investigations are associated with calculations of the chemical interaction, phase transformations, and condensed-phase motion in the system considered.

NOTATION

G, gas constant, J/(kg·K); k, kinetic energy of turbulence, m^2/sec^2; p, pressure, Pa; Pr, Prandtl number; q, value of the flow velocity, m/sec; $\mathbf{q}$, vector of the flow velocity; R, characteristic dimension of a channel, m; Re, Reynolds number; t, time, sec; T, temperature, K; x, y, coordinates, m; γ, adiabatic index; ε, rate of dissipation of the kinetic turbulent energy, m^2/sec^3; ρ, density, kg/m^3; φ, angle of inclination of the compensator. Subscripts: f, fuel; g, gas.

REFERENCES

1. STAR-CD. Version 3.10. User Guide. Computational Dynamics Ltd., London (1999).
2. STAR-CD. Version 3.10. Methodology. Computational Dynamics Ltd., London (1999).
3. V. M. Eroshenko and L. I. Zaichik, *Hydrodynamics and Heat Transfer on Permeable Surfaces* [in Russian], Nauka, Moscow (1984).
4. B. A. Raizberg, B. T. Erokhin, and K. P. Samsonov, *Principles of the Theory of Working Processes in Solid-Fuel Rocket Systems* [in Russian], Mashinostroenie, Moscow (1972).
5. A. M. Lipanov, V. P. Bobryshev, A. V. Aliev, F. F. Spiridonov, and V. D. Lisitsa, *Numerical Experiment in the Theory of Solid-Fuel Rocket Engines* [in Russian], Nauka, Ekaterinburg (1994).
6. A. A. Shishkov, S. D. Panin, and B. V. Rumyantsev, *Working Processes in Solid-Fuel Rocket Engines* [in Russian], Mashinostroenie, Moscow (1989).
7. V. N. Emel'yanov, Physical and computational simulation of three-dimensional flows in engines, in: O. Ya. Romanov (Ed.), *Interchamber Processes*, *Combustion*, a*nd Gas Dynamics of Disperse Systems* [in Russian], Izd. BGTU, St. Petersburg (1996), pp. 124–137.
8. V. N. Emel'yanov, Complex internal flows, in: O. Ya. Romanov (Ed.), *Interchamber Processes*, *Combustion*, a*nd Gas Dynamics of Disperse Systems* [in Russian], Izd. BGTU, St. Petersburg (1998), pp. 80–91.
9. K. N. Volkov and V. N. Emel'yanov, 3D turbulent flows in channels with injection, in: V. N. Uskov (Ed.), *Modern Problems of Nonequilibrium Gas Dynamics and Thermodynamics* [in Russian], Izd. BGTU, St. Petersburg (2002), pp. 43–63.
10. A. D. Rychkov, *Mathematical Simulation of Gas-Dynamic Processes in Channels and Nozzles* [in Russian], Nauka, Novosibirsk (1988).