The Technique for CFD-Simulation of Fuel Valve from Pneumatic-Hydraulic System of Liquid-Propellant Rocket Engine

L S Shabliy[1], D V Malov[1] and D S Bratchinin[1, 2]

[1] Samara National Research University, 34, Moskovskoe shosse st., Samara, Russia
[2] PSC Kuznetsov, 29, Zavodskoye shosse, Samara, Russia

E-mail: shelbi-gt500@mail.ru

Abstract. In the article the description of technique for simulation of valves for pneumatic-hydraulic system of liquid-propellant rocket engine (LPRE) is given. Technique is based on approach of computational hydrodynamics (Computational Fluid Dynamics – CFD). The simulation of a differential valve used in closed circuit LPRE supply pipes of fuel components is performed to show technique abilities. A schematic and operation algorithm of this valve type is described in detail. Also assumptions made in the construction of the geometric model of the hydraulic path of the valve are described in detail. The calculation procedure for determining valve hydraulic characteristics is given. Based on these calculations certain hydraulic characteristics of the valve are given. Some ways of usage of the described simulation technique for research the static and dynamic characteristics of the elements of the pneumatic-hydraulic system of LPRE are proposed.

1. Introduction

The hydraulic processes occurring in the liquid-propellant rocket engine (LPRE) are the subject of close attention of the research engineers, since it largely determines the efficiency and reliability of the engine. The calculated prediction of the nature and characteristics of the fluxes in the engine elements allows to identify possible design errors even before the product is embodied in the metal, and find ways to improve existing engines which much lower costs in comparison with the experimental study. However, using of popular in recent years CFD-simulation technologies of hydrodynamic processes of liquid rocket engines [1, 2], allows to perform even optimization researches on high-level models (in fact, virtual prototypes), but in the domestic rocket-engines industry it is still not widespread. Therefore, at the present time, young specialists are faced with the task of developing and mastering CFD-simulation technologies, which in the conditions of modern re-equipment of computational tools (the emergence of cluster computing technologies, supercomputers), will provide a fast pace of designing, manufacturing and testing LPRE aggregates [3, 4, 5]. This work shows clearly modern simulation tools applicable to space rocket industry.

2. Pneumatic-hydraulic system of LPRE

The work of the LPRE chamber is provided by the pneumatic hydraulic system (PGS) of the propulsion system. The term "PGS" is understood to mean a set of pneumatic-hydraulic devices and pipes, providing storage of fuel components and gases on board aircraft, their feeding during engine operation under a certain pressure and with a certain flow rate in the combustion chamber and engine gas generator, starting and stopping the engine, as well as performing some other operations determined by the purpose and specificity of the operation of the aircraft [6, 7]. Modeling the flow in pipelines, connectors and other elements, which does not change its shape of flowpath when the engine is running, usually does not represent complexity. Another situation occurs with valves - necessary elements of any PGS.

The article describes the approaches to the modeling of valves of the pneumohydraulic LPRE system by computational fluid dynamics (CFD) on the example of differential valve simulation. Such valves are used, for example, in the main trunking of the components of the LPRE NK-33 (AJ-26 in NASA classification). The NK-33 engine is a multiple-launch engine, those allows providing multiple start-up (acceptance test and operation as part of carrier rocket) without bulkheading of the main units and blocks [8, 9]. Despite the fact that after each start-up, it is necessary to replace certain single-acting units, the usage of differential valves allows not to remount the engine after the test.

2.1. Design of differential valve

The differential valve under consideration has a straight-flow construction (Figure 1). The body 1 consists of an outer shell and a central body mounted on pylons. In the central body is installed the stem of the movable valve 3, pressed by a spring 4 to the seat of the cover 2, connected to the body. To ensure the tightness of the valve in the closed state, the valve and cover contact is provided by the seal 5, which can be constructed from rubber or, in the case of a cryogenic component, a fluoroplastic.

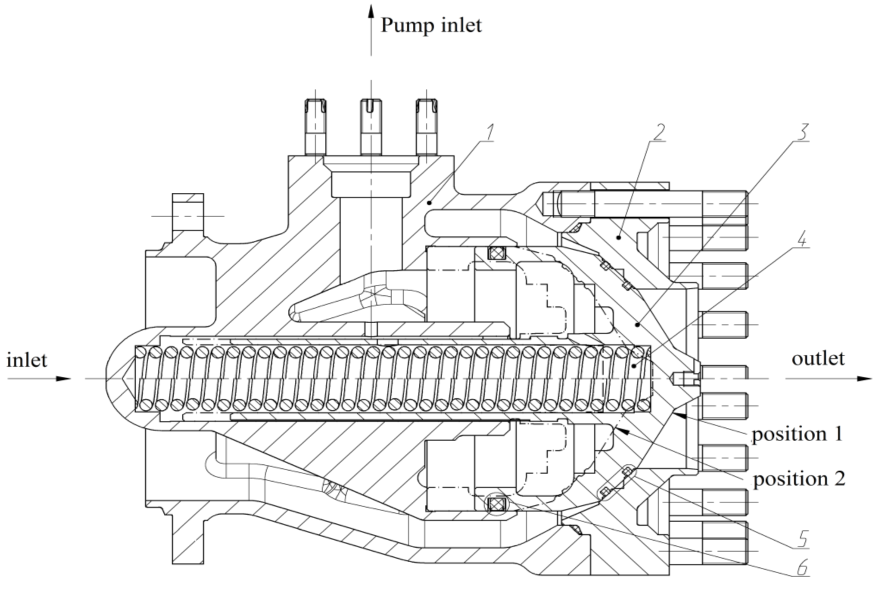

Figure 1. Longitudinal section of the differential valve under consideration: 1 – body; 2 – cover; 3 – valve; 4 – spring; 5, 6 – seals

2.2. Principle of operation of the differential valve

The valve is designed to control the timing of the fuel supply to the gas generator or the engine chamber, and also for stopping its feeding when the engine is switched off. Before starting the engine, the valve is closed by the action of a spring. Fuel is piped from the pump to the valve inlet, filling the cavity A before the seals. Also filled cavity B, connected through the flange to the pump inlet. While the pressure in cavities A and B is the same, the valve remains closed under the action of the spring, and the fuel does not flow past the valve to the consumer. After the turbo-pump unit (TPU) is unleashed by the pyro-turbine, the pressure in the supply line and the cavity A connected to it increases. The valve 3 (figure 1) is shifted to the left due to the fuel pressure in the gap between the valve 3 and the cover 2 on the tapered surface of the valve. When the valve opens, the fuel begins to act on the entire surface of the valve, ensuring its full opening. At the same time, on the side of the cavity B, the pressure remains equal to the pressure at the pump inlet. Thus, the valve remains in the open state as long as the pressure in the cavity A exceeds the pressure in the cavity B, which corresponds to the operation of the pump. The pressure difference at which the valve starts to open is determined by the initial spring force and the pressure at the pump inlet. For closed-circuit LPRE, when starting with a safe feed sequence, the components are the one at which an oxidizer first enters the consumer. At given pressures at the pump inlet, determined by their cavitation characteristics, and the dynamic characteristics of the pumps, the opening moment of the valves in the engine system is given by the initial spring compression force.

When the engine exits to the main thrust stage, the rotational speed of the TPU rotor increases, and the increasing pressure in the supply line increases the force that presses the valve 3 to the body 1. In this case, the valve remains in the open position. After operation the engine is switching off by stopping the supply of fuel to the gas generator from the control. In this case, the combustion in the gas generator ceases, the rotational speed of the rotor TPU drops and the pressure at the outlet from the pumps drops too. As a result, the pressure between the cavities A and B is equalizing, the valve 3 (see figure 1) under the action of the spring 4 returns to its original state - it is pressed against the cover 2, ensuring the closing of the valve.

3. Technique of CFD-simulation for differential valve

The above principle of operation of the differential valve shows the complexity of the valve operating process due to the presence of a differential pressure at the pump inlet and outlet. In this study, flow was simulated through the valve cavity A at different positions of the movable element: from fully open to minimally open. Also, the forces acting on the moving part of the valve were evaluated. The flow in the cavity B was not simulated.

As with any CFD simulation, the calculation process consisted of five stages: geometric modelling, meshing, imposition of boundary conditions, calculation and analysis of the obtained results.

3.1. Geometry

The geometrical model of the flow part was obtained by modifying the geometric model (Figure 2) created by the authors [10, 11] during the work on the project "Development of technology and methodological support for the creation of a virtual LPRE", implemented as part of the activities of block 2 "Development And increasing the effectiveness of scientific and innovative activities "and block 3 "Development of the information science and education environment and infrastructure" Development programs of the SSAU for 2009 - 2018. It is limited by the internal surfaces of the valve body, as well as the supply and discharge pipeline, the surfaces of the central body holding its pylons, and the movable part of the valve. Geometry reflects the main surfaces that affect the flow. At the same time, elements that do not have a significant effect on the flow (connecting gaps, small blind holes for connecting elements, etc.) were not reflected in the model, so as not to complicate the further construction of the mesh.

Since the task of this study was to simulate the flow at different valve positions, the geometric model was supplemented by another geometric body modelling the volume occupied by the flow part valve at its intermediate positions (shown in figure 2 in a darker color). Thus, the modified geometric model contained two geometric elements: a solid body representing the entire flowing part of the fully open valve (see figure 2) and the second solid body representing the moving element of the valve. This division allowed the relative displacement of two bodies to be modelled without the need to rebuild the computational mesh, since during the simulation of the "opening" of the valve, the reduction in the cavity B (see figure 1) was not modelled (see figure 2), and the displacement occurred unobstructed penetration of the moving body into the stationary one.

This division allowed the relative displacement of two bodies to be modelled without the need to rebuild the computational grid, since during the simulation of the "opening" of the valve, the reduction in the cavity b (see figure 1) was not modelled (see figure 2), and the displacement occurred unobstructed penetration of the moving body into the stationary one.

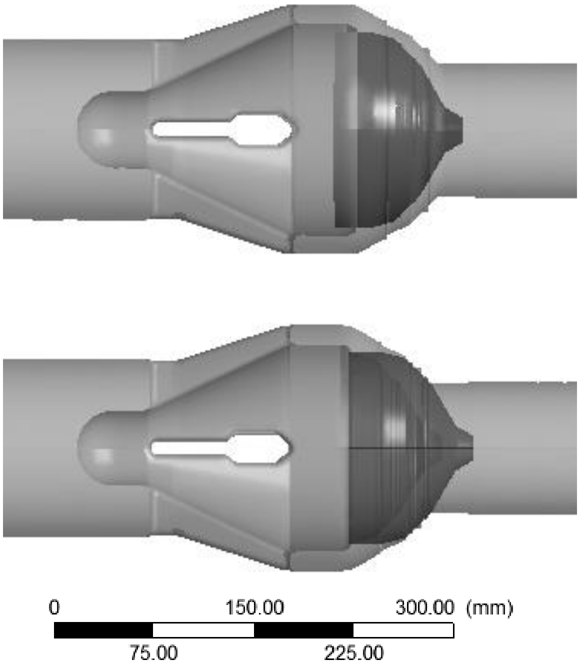

Figure 2. Geometrical model of the valve flowpath. The position of an additional geometric body that fills the volume of the movable part of the valve when its position corresponds to the fully closed state (below) and the semi-open (upper) valve mode

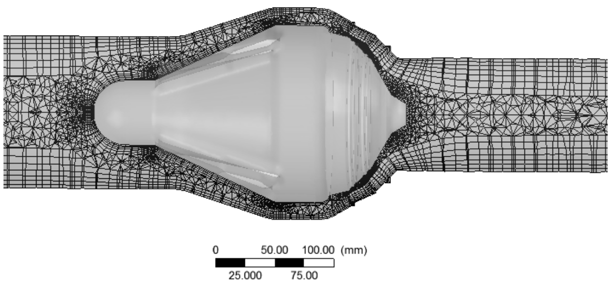

Figure 3. Mesh visualization for the model of a fully open valve in a plane at an angle of 45 ° to the plane of symmetry of the pylons of the central body

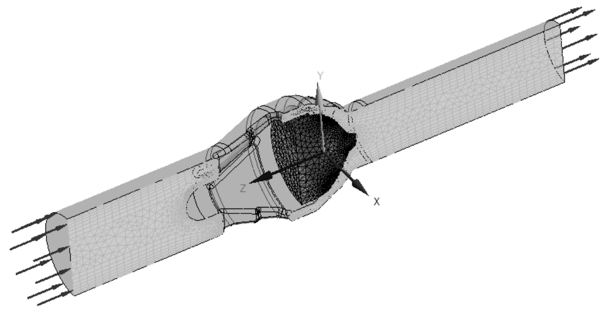

Figure 4. Visualization of the calculation model of the flowpath with immersed solid body represented via the mesh model

3.2. Meshing

The mesh model was created in the ANSYS Meshing program. The following settings were used:

- association of the dimensional function of grid elements - to the curvature (Use Advanced Size Function → On: Curvature);
- the limiting sizes of elements: (Min Size) – 0.1 mm, maximum (Max Size) – 23 mm, maximum size of the side of the element (Max Face Size) – 12 mm.
- coefficient of ratio of the sizes of neighbouring cells (Growth Rate) – 1.2
- smooth transition to thickening of the boundary layer (Inflation Option → Smooth transition)

Fully generated mesh (Figure 3) contained 2.3 million elements (tetrahedral - pyramids with triangular base, triangular prisms and pyramids with a quadrangular base)

3.3. Moving parts simulation

As already mentioned, the simulation of overlapping by the moving part of the valve of the flowing part was carried out without rebuilding of mesh, using the model of "solid body immersed in liquid" (Immersed Solid) [12]. A mesh of the flow part corresponding to the fully open valve is used (Figure 3), but a part of the cells marked by the program as cells occupied by a solid body (Figure 4) become impermeable to the flow. This is clearly seen from the current lines in different modes (Figure 5). In this case, the displacement of the Immersed Solid body was specified at the stage of the solution start as a parameter of the model "displacement along the local Z-axis" (Figure 4).

3.4. Stationary simulation

The ones shown in Figure. 5 flow patterns were obtained by calculations in a stationary setting. The data obtained from such calculations can characterize the throttling properties of the valve (Figure. 6). However, specific throttling characteristics can be obtained only for the previously known operating conditions of the valve determined by the PGS: a constant pressure difference between the valve and the variable flow or vice versa, a constant flow at a varying pressure at one or two boundaries. The ones shown in Figure. 5 and 6, the results were obtained by simulating a constant flow rate.

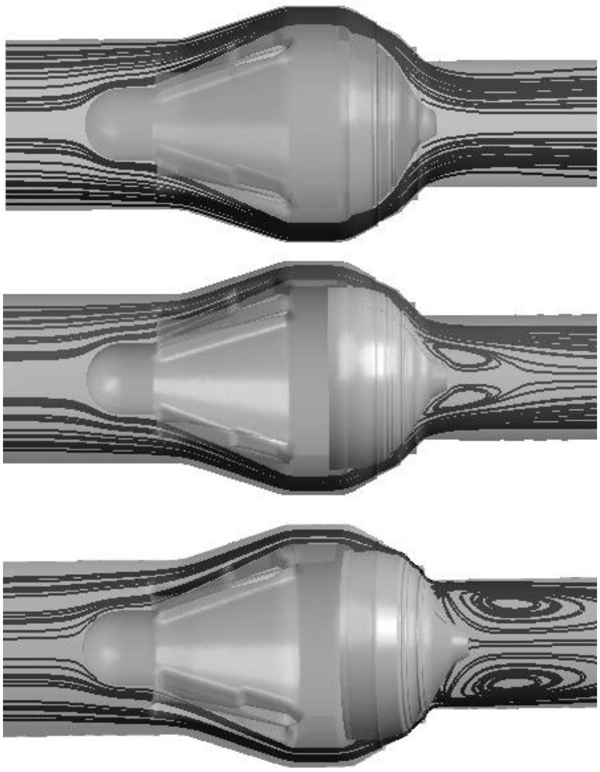

Figure 5. Lines of current in the valve: fully open (top), partially open (in the middle), almost closed (bottom) with 3 mm displacement above the fully closed position

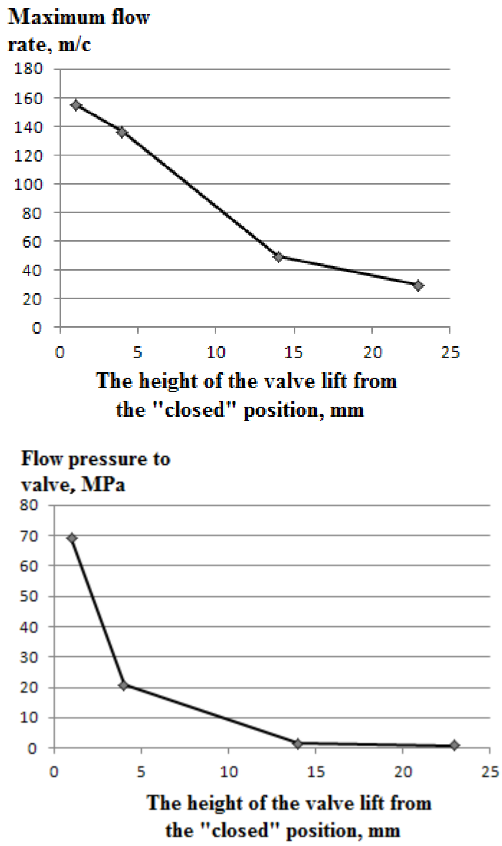

Figure 6. Valve characteristics depending on the lift height (displacement) of the valve

3.5. Transient simulation

At the second stage, the flow through the valve was modeled, taking into account the movement of its moving parts in an unsteady setting. In this case, the motion of the locking element can be set according to the laws of mechanics of solids (Rigid Body [12]). In this case, the ANSYS CFX solver takes into account the superposition of inertial forces, spring elasticity and fluid pressure based on the specified mechanical properties of the moving element (orientation, degrees of freedom, center of mass, mass, moments of inertia, initial velocity and acceleration, etc.) and calculates the displacement in Given instants of time. The visual result of this simulation is the video of the valve working process under given conditions, which can be used to study the work on virtual prototypes.

4. Conclusions

Thus, the described set of software tools, models and sequence of their application can be considered as an experimented technique that allows modeling of valves, as well as other elements of the fluid hydraulic system of LPRE, and determination of their characteristics with the framework of optimization studies [13] or gas-dynamic tests [14].

Acknowledgments
The authors wishing to acknowledge assistance or encouragement from colleagues [10, 11] for the high quality of the geometric model of the flow-through part of the valve that they created in the framework of the project "Development of technology and methodological support for the creation of a virtual liquid-propellant rocket engine (LPRE)", which was taken as a basis for creating a modified geometric model used in this study.

The work was completed with the financial support of the Ministry of Education and Science of Russia within the basic part framework of the state assignment and with Grant Council of the President of the Russian Federation in the framework of the SP-3790.2015.1 scholarship.

References
[1] Zubanov V, Egorychev V and Shabliy L 2015 Design of rocket engine for spacecraft using CFD-modeling *Procedia engineering* **104** pp 29–35 doi: 10.1016/j.proeng.2015.04.093
[2] Zubanov V, Egorychev V and Shabliy L 2014 Hydrogen-Oxygen rocket engine design using CFD-modeling Research Journal of Applied Sciences 9(10) pp 660–663 doi: 10.3923/rjasci.2014.660.663
[3] Egorychev V S and Ryazanov A I 2016 A method of design calculation for the two-component jet-jet gas nozzle with outer mixing Russian Aeronautics 59(4) pp 619–622 doi: 10.3103/S1068799816040280
[4] Zubanov V M, Shabliy L S and Krivcov A V 2015 Rational technique for multistage centrifugal pump CFD-modeling Proceedings of the ASME Turbo Expo 2B doi:10.1115/GT201542070.
[5] Shabliy L and Cherniaev A 2014 Optimization of gas turbine compressor blade parameters for gas-dynamic efficiency under strength constraints Proceedings of the 4th International Conference on Simulation and Modeling Methodologies, Technologies and Applications SIMULTECH 2014 pp 523-528 doi: 10.5220/0004994905230528
[6] Belousov A I and Sedel'nikov A V 2014 Problems in formation and control of a required microacceleration level at spacecraft design, tests, and operation Russian Aeronautics 57(2) pp 111–117 doi:10.3103/S1068799814020019
[7] Vaulin S D, Vaulina O S, Salich V L and Schipicin A G 2005 The research of dynamics processes in the plant of the liquid industrial wastes utilization on the basis of the rocket engine on gas propellants Izvestiya Akademii Nauk. Energetika 1 pp 101–109
[8] Borisov V A, Zhizhkin A M and Melent'ev V S 2011 Liquid-propellant rocket engine NK-33 (Samara: Samara University Publ.) p 27
[9] Parovay F V, Borisov V A and Parovay E F 2014 Research of temperature influence on the contact pressure in radial seals Research Journal of Applied Sciences (Medwell Journals) 9(11) pp 761–766 doi: 10.3923/rjasci.2014.761.766
[10] Gimadiev A G, Bystrov N D, Gafurov S A and Bratchinin D S 2013 Design of operating process model of main fuel valves of LPRE (Samara: Samara University Publ.) p 73
[11] Gimadiev A G, Stadnik D M and Bratchinin D S 2014 Analysis of the flow force in the fuel components supply valves of the aircraft engines ARPN Journal of Engineering and Applied Sciences 9(12) pp 2864-2866
[12] ANSYS Inc. 2014 CFX Modeling Guide
[13] Shablii L S and Dmitrieva I B 2014 Blade geometry transformation in optimization problems from the point cloud to the parametric form Russian Aeronautics (Allerton Press Incorporation) 57(3) pp 276–282 doi: 10.3103/S1068799814030106
[14] Matveev V N, Shabliy L S and Krivcov A V 2017Application of stereolithography prototypes for gas dynamic tests and visualization. Journal of Physics: Conf. Series 803 012097 doi:10.1088/1742-6596/803/1/012097