Numerical simulation of combustion processes in combustion chamber of hybrid solid fuel engine

Lyuben Stamov, Veronika Tyurenkova, Elena Mikhalchenko

Federal Science Center Scientific Research Institute for System Analysis of Russian Academy of Sciences, 36-1 Nakhimovskiy pr., Moscow, Russia, 117218

lyubens@gmail.com

Hybrid solid-propellant engines, which use fuel mixture components in various states of aggregation, are very attractive. This type of engine has a number of advantages over liquid or solid propellant engines. For example, compared to solid fuel engines, it has greater safety due to the storage of fuel components separately from each other. It also allows you to control the traction of the device. Compared to liquid fuel engines, hybrid engines are simpler in design and easier to maintain. One of the fastest and relatively cheap ways to develop such engines, as well as other engines, is computer simulation. Carrying out predictive computational modeling of such devices due to its rather high complexity is difficult without the use of high-performance computing systems and the development of parallel algorithms and programs.

In this work, a three-dimensional numerical simulation of the processes occurring in the combustion chamber of a solid-fuel engine, into which the oxidizer enters in gaseous form at supersonic speed, was done. Gaseous oxygen and air were used. It is assumed that in the course of chemical reactions occurring on the surface of a solid fuel as a result of interaction with the oncoming flow, a gaseous combustible substance, decomposition product of a solid fuel, is released into the chamber, which also begins to mix and react with the oncoming flow of the oxidizer. The HTPB (Hydroxyl-terminated polybutadiene) and PMMA (Polymethyl methacrylate) solid fuels were used. Modelling the release of gaseous fuel through a solid fuel surface was done by near-wall functions calculated on the basis of the parameters of the medium near the wall surface obtained in the course of numerical calculation. The characteristic dimensions of the engine are based on experimental data from the work [1,2]. A specialized author's program code was developed for the simulation. The numerical model is based on the MUSCL method of flow interpolation, the AUSMP method for compression conditions on a uniform cubic grid and the semi-implicit Novikov method for a rigid system of kinetic equations. An increase to the second order of accuracy was carried out according to the McCormack method. Wilcox's k-ω turbulence model was used. In the course of the study, a comparison with experimental data was done. A series of test computational experiments was carried out to determine the distributions of physical parameters inside the combustion chamber. The diffusion mode of the combustion process in the combustion chamber, large flow turbulence, vortex formation and strong process asymmetry have been obtained.

Russian Foundation for basic research is acknowledged for financial support (RFBR 20-07-00587).

References

1. A. Ben-Yakar, B. Natan, A. Gany, *Journal of Propulsion and Power* **2015**, 14 (4) 447–455.
2. X. Sun, H. Tian, Y. Li, N. Yu, G. Cai, *Acta Astronautica* **2016**, 119, 137–146.