Paper ID: 56838
oral

IAF SPACE TRANSPORTATION SOLUTIONS AND INNOVATIONS SYMPOSIUM (D2)
Technologies for Future Space Transportation Systems (5)

Author: Mr. Serhii Furkalo
Yuzhnoye State Design Office, Ukraine, Furkalo@science.yuzhnoye.com

COMPLEX NUMERICAL SIMULATION OF THE LAUNCH SYSTEM GAS-DYNAMICS

Abstract

The launch is one of the most important phases of the mission requiring special technical solutions to ensure trouble-free and reliable launch. The source of increased risk is the intense thermal and pressure impact of the launch vehicle plume on the elements of the launch system and on the launch vehicle itself. Taking into account the significance of the possible consequences of design errors, all possible measures are applied to optimize the design. The most accurate parameters can be obtained during bench gas-dynamic and flight tests which are necessary to confirm the operability of the structure, as well as to define more precisely the parameters and configuration of the equipment and systems of the launch system. However, the full-scale tests are expensive and significantly increase the development time of the launch system. Therefore, a numerical simulation of the launch process is quite important in the design of the launch system. The complexity of such a simulation is the need to take into account a number of factors. Propulsion systems of launch vehicles are structurally composed of several nozzles, during whose operation, due to the interaction of the plumes, zones of high pressure and intense heat fluxes are formed. In the plume, the combustion of the propellant occurs, which significantly affects the thermodynamic parameters of the flow. The possibility of reusable launch systems requires the use of additional systems to decrease loads. The water supply system got the greatest practical application. Having extensive experience in developing launch systems, an approach has been developed for the practical application of CFD methods and high-performance computing systems for modeling launch processes. The results of that simulation determine the parameters of the pressure and thermal effects on the launching equipment. Using numerical simulation allows us to optimize the volume of gas-dynamic tests and significantly reduce the cost of the launch system development. This paper presents a numerical simulation technique with ANSYS Fluent for a multicomponent gas plume, taking into account chemical kinetics, high-speed and thermal interaction of the gas and dispersed phases of water droplets fed into the plume. The technique was validated by comparing the reference calculations with the experimental data of existing launch systems. The simulation results allow choosing the optimal design of the trench, determining the need for additional protection systems for the launch system, as well as reducing the amount of bench tests.