SPACE PROPULSION SYMPOSIUM (C4)
Interactive Presentations (IP)

Author: Mr. Florin Mingireanu
Romanian Space Agency (ROSA), Romania, florin.mingireanu@rosa.ro

Prof. Florin Frunzulica
Romania, ffrunzi@yahoo.com
Dr. Marius STOIA-DJESKA
University POLITEHNICA of Bucharest - Research Center for Aeronautics and Space, Romania, marius.stoia@gmail.com

VALIDATION WITH CFD SIMULATIONS OF A SIMPLIFIED MODEL OF A HYBRID ROCKET MOTOR

Abstract

The hybrid propulsion offers some remarkable advantages and thus it is considered a promising technology for future launchers and space systems. The complex interaction between (turbulent) flow and combustion in hybrid rockets arise some scientific challenges like regression rate modeling, the increase of combustion efficiency and instabilities mitigation by active control. Furthermore, the pressure sensitivity of the combustion process, the coupling between the motor and the oxidizer feed system, the vortex shedding in the aft mixing chamber makes indispensable a study of the mutual coupling between the main flow stream, the fuel gasification and combustion and the thermal conduction into the solid fuel.

The purpose of this work is to validate through complex computational fluid dynamics (CFD) simulations a design tool for hybrid rocket motors (HRM). A simpler model is required for performing overall analysis and parametric studies, which are necessary for optimisation and for the development of control techniques. The FLUENT commercial code is used for the numerical simulation of the mutually interacting three-dimensional flow, combustion and heat conduction processes in a very rigorous manner. In order to take into account the modification of the geometry of the combustion port due to the consumption of the solid fuel, one uses an UDF to implement the effects of the regression rate on the moving boundary.

The simplified HRM model is based on the coupling of the hybrid combustion process with the complete unsteady flow, modeled with the one-dimensional Euler equations. The flow equations and the fuel regression rate law are solved in a coupled manner. Each propellant combination has its unique regression rate formula and thus such a model includes some key operational parameters that are known and used by engineers in the design process. The platform of the numerical simulations is an implicit fourth-order Runge-Kutta high order cell-centred finite volume method.

The numerical results allow the comparison of the pressure and temperature in the combustion chamber and of the overall thrust with those obtained with the simpler model. The numerical results obtained with both models show a good agreement between them and with published experimental and numerical results.