Numerical Simulations of Flow and Fuel Regression Rate Coupling in Hybrid Rocket Motors

Marius STOIA-DJESKA*[,1], Florin FRUNZULICA[1], Florin MINGIREANU[2]

*Corresponding author
[1]"POLITEHNICA" University of Bucharest, Faculty of Aerospace Engineering,
Gh. Polizu Street 1-7, Sector 1, Bucharest 011061, Romania,
marius.stoia@rosa.ro*, ffrunzi@yahoo.com
[2]Romanian Space Agency,
Mendeleev Street 21-25, sector 1, Bucharest 010362, Romania,
florin.mingireanu@rosa.ro

DOI: 10.13111/2066-8201.2017.9.1.9

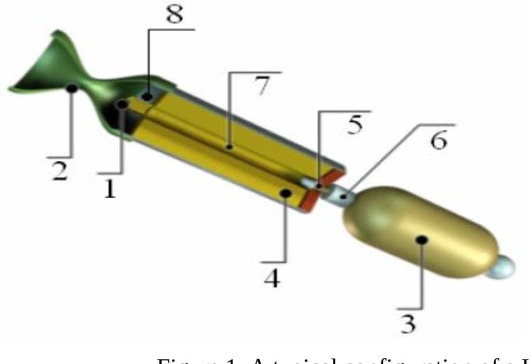
Received: 30 January 2017/ Accepted: 18 February 2017/ Published: March 2017

Abstract: *The hybrid propulsion offers some remarkable advantages like high safety and high specific impulse and thus it is considered a promising technology for the next generation launchers and space systems. The purpose of this work is to validate a design tool for hybrid rocket motors (HRM) through numerical simulations.*

Key Words: *hybrid rocket motor, HRM numerical simulation, CFD, hybrid propulsion system*

1. INTRODUCTION

The Hybrid Rocket Motor (HRM) is an efficient propulsion system for different purposes, [2], [4]. The hybrid propulsion offers some remarkable advantages like high safety and high specific impulse and thus it is considered a promising technology for the next generation launchers and space systems. The commonly used butadiene based fuels are nontoxic, easy to store and transport and safer to manufacture. Another very important feature of a HRM is the possibility of thrust modulation through a throttle system. A direct hybrid rocket motor uses a solid fuel and a liquid oxidizer. The typical configuration of a HRM is shown in Figure 1.



1. Combustion chamber,
2. Nozzle,
3. Oxidizer tank,
4. Fuel grain,
5. Injector valve,
6. Head of the injector valve,
7. Burning port,
8. Solid propellant.

Figure 1. A typical configuration of a Hybrid Rocket Motor

The physical mechanism characterizing hybrid rockets is the boundary layer combustion, [4]. Unfortunately, the direct numerical simulation of the mutually interacting flow, combustion and heat conduction processes is too difficult to be performed in the preliminary design phase of a HRM.

The complexity of the physical processes results from Figure 2.

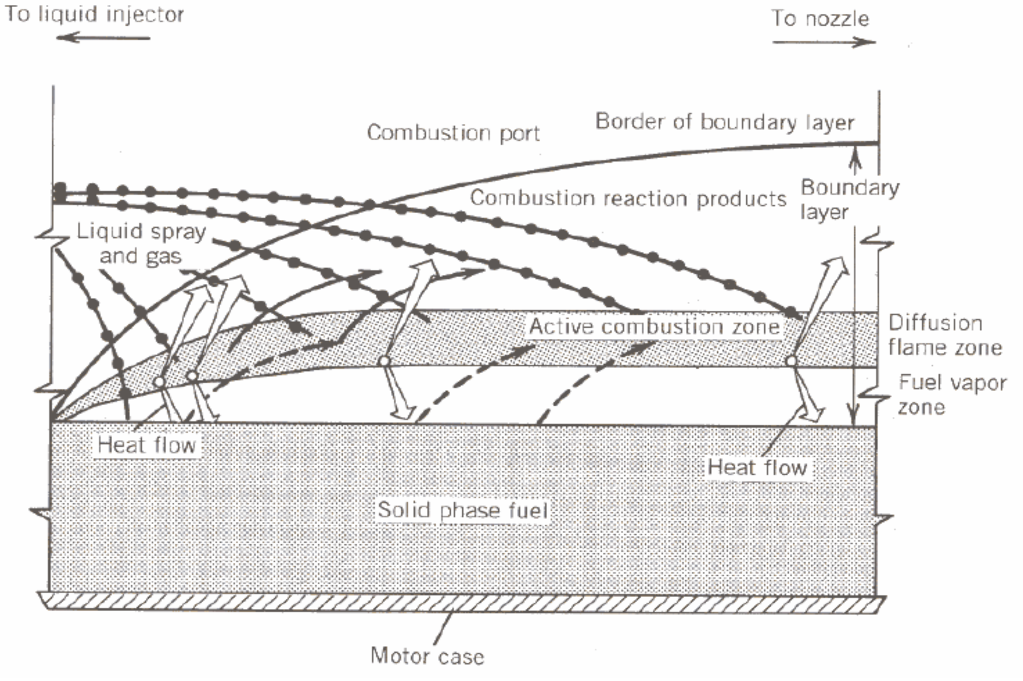


Figure 2. The complexity of the physical processes in a HRM

From a numerical point of view, a CFD based investigation of the flow and of the other physical processes faces with some difficulties arising, among others, from the dynamics of the boundary between the fuel and the combustion port, see Figure 3.
The burn of the fuel produces not only the injection in the combustion port of mass and energy fluxes, but also the movement of the previously mentioned boundary, [1].

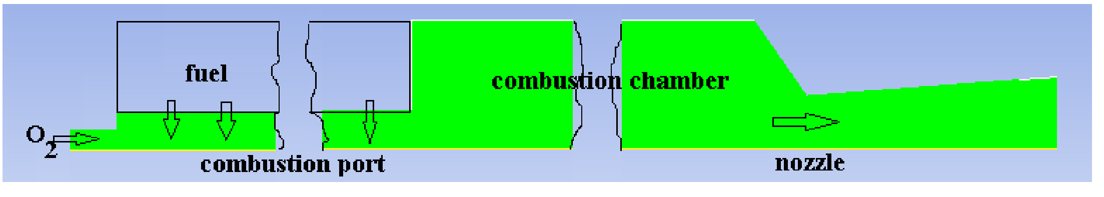


Figure 3. Schema of a HRM (axis-symmetric geometry)

A simpler model is therefore required for performing overall analysis and parametric studies during the design phase of a HRM.

Of course, such a simpler model must be verified and validated before being used for engineering purposes.

The validation of such a simpler model through numerical simulations is the objective of this work.

2. THE SIMPLIFIED MODEL OF FLOW

The flow through the HRM combustion port is assumed to be unsteady, compressible and essentially one-dimensional, [1]. At each cross-section the kinematic and thermodynamic flow parameters are homogeneous. The flow parameters are supposed to vary continuously and smoothly along the channel. The mass, momentum and total energy conservation laws can be applied to describe the flow.

The viscous effects are neglected. The burning surface of the propellant supplies continuously the gases flowing through the channel with combustion products and also with energy. The heat exchange in flow is dominated by the convection and thus the conduction along the combustion port axis is neglected.

The mixture density, temperature and pressure are constant on the cross section. The assumption about the one-dimensionality of the flow leads to the use of only the axial velocity explicitly in the model. The flow model adopted in this work is the 1-D Euler equations of gas dynamics with source terms:

$$\frac{\partial}{\partial t}\begin{Bmatrix}\rho\\ \rho u\\ \rho E\end{Bmatrix}+\frac{\partial}{\partial x}\begin{Bmatrix}\rho u\\ \rho u^2+p\\ u\left(\rho E+p\right)\end{Bmatrix}=-\frac{1}{A}\begin{Bmatrix}\rho u A_x+\rho A_t\\ \rho u^2 A_x+\rho u A_t\\ u\left(\rho E+p\right)A_x+\rho E A_t\end{Bmatrix}+\frac{1}{A}\begin{Bmatrix}\dot m\\ 0\\ c_p\,T_f\,\dot m\end{Bmatrix},\ A=\pi r_A^2 \qquad (1)$$

where $p=\rho RT=(\gamma-1)\left[\rho E-\frac{(\rho u)^2}{2\rho}\right],\ e=c_v T=\frac{1}{\gamma-1}RT,\ \dot m=\dot r\,\rho_p\,2\pi\, r_A.$

The regression rate model is given by $\dot r = a\,G^n,\ G=\rho u$, $\frac{dr_A\left(t,x\right)}{dt}=\dot r\left(t,x\right)$ with the constants a=0.087 and n=0.615 (these coefficients are depending on the fuel-oxidizer combination).

The geometry of the computational domain adopted in this work consists in the combustion port, the post-combustion chamber and the nozzle, see Figure 4.

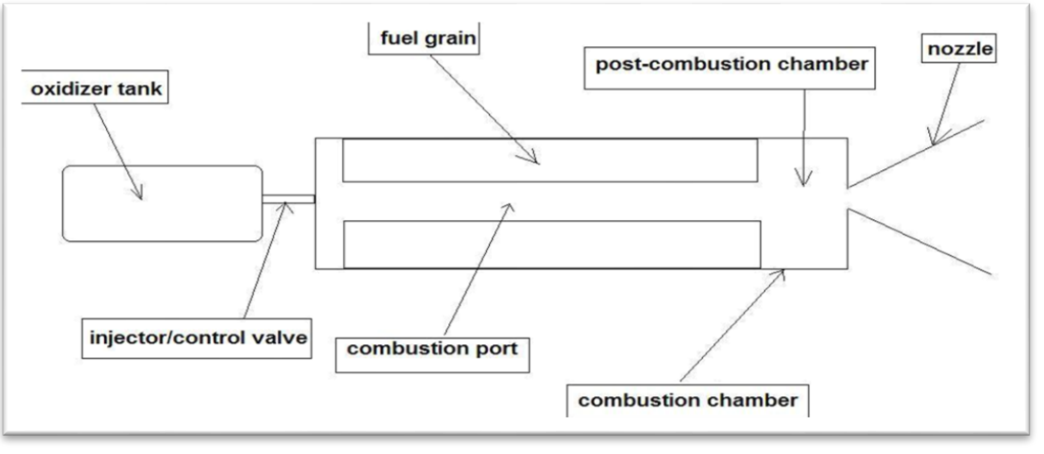


Figure 4. The simplified geometry of the computational domain used in the 1-D approach

The left boundary is the entrance in the combustion port. The outlet boundary coincides with the nozzle exit.

The numerical solution procedure is based on the standard cell-centered finite volume scheme of Godunov type, [5], coupled with an implicit second-order discretization in time. The numerical fluxes at cells faces are evaluated using Roe's flux-difference scheme. The second order spatial accuracy is obtained by linearly expanding the cell-centered values to each cell face. To maintain the monotonic character of the scheme we use the MINMOD type limiter applied to gradients in the regions where steep gradients and/or shocks are present. Due to the presence of the very strong source terms in the governing equations and in order to avoid the use of very small type steps, an implicit algorithm is used for time advancement and this is done using a dual time approach, [6]. The numerical simulation of the 1-D compressible flows coupled with the fuel regression rate law allows the calculation of the kinematic and thermodynamic variables throughout the flow field and fuel grain. The burning chamber pressure, temperature and density as functions of time are typical examples. Further, used as virtual test facility, the code offers also global performances of the HRM like the total impulse and the thrust versus time.

3. CFD SIMULATIONS AND COMPARISON OF THE RESULTS

For CFD simulation we have used the ANSYS Fluent commercial software with the following settings: axis-symmetric inviscid flow, transient density based solver, implicit second order - flux treatment, implicit one-order transient formulation, dynamic mesh for regression surface. The time step based on physically formulation is about $\sim 10^{-7}$ sec. The geometry of the computational domain adopted in this work consists in the combustion port, the post-combustion chamber and the nozzle.

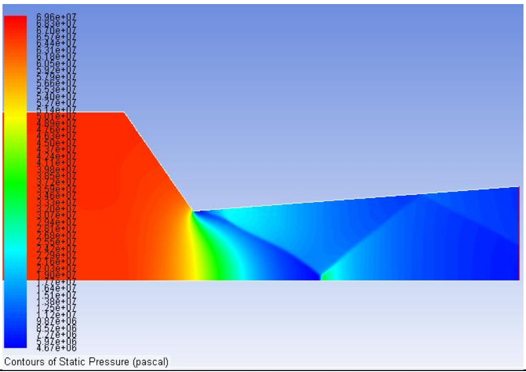
The main dimensions are: length - 1524 mm, exterior diameter - 80 mm, diameter of oxygen valve - 8.89 mm, critical diameter of nozzle - 33.02 mm, outside diameter of nozzle - 44.80 mm. The burning process involved HTPB (Hydroxyl-Terminated Poly-Butadiene) and as oxidizer, Nitrous - Oxide. The constants used in simulations are: adiabatic coefficient of burning products - 1.5, burning gas constant - 336.6777 J/Kg/K, oxidizer adiabatic coefficient - 1.4, oxidizer constant 259.8 J/kg K, initial oxidizer pressure 50×10^5 Pa. The mesh has mixed elements: triangular and quadrilateral elements (317562 cells and 202212 nodes).

Figure 5 shows the snapshots of static pressure and Mach number at time = 1 sec., in the nozzle and in the combustion port. The numerical results show that the flow in the nozzle has some specific features: a) very high values of the pressure and temperature in the combustion chamber, b) the fluid is strongly accelerated in the nozzle, c) the occurrence of shock waves and d) the very low Mach number in the combustion chamber. All these aspects have also been revealed by the simpler 1-D model. Further, in the table 1 the specific impulse and total impulse obtained from the results calculated with the 1-D model are presented and they are in good agreement with those obtained from CFD simulations and experimental data, [3].

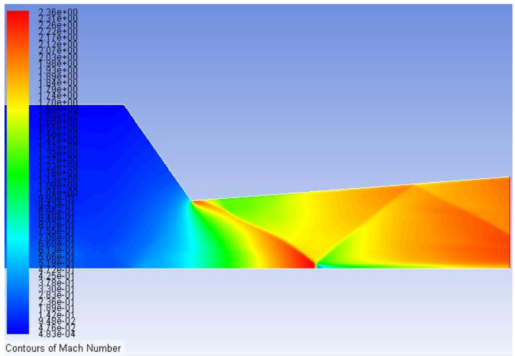
Table 1. Specific and total impulse

Impulse	Experimental	1-D model	Axis-symmetric model
Specific impulse (s) $I_{sp}=Thrust/(\dot{m}\,g)$	145.4	149	147.38
Total impulse (Ns) $I_{tot}=\int_{o}^{t_f} Thrust(t)\,dt$	1374	1404	1395.72

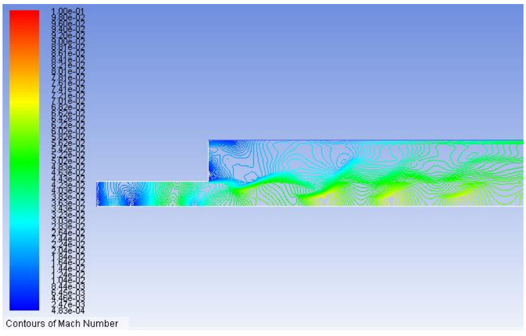
Contours of static pressure (Pa)



Contours of Mach number in nozzle (M = 0... 2.36)



Injection port and combustion port - contours of Mach number, M=0...0.1



Combustion port - contours of Mach number, M=0...0.5

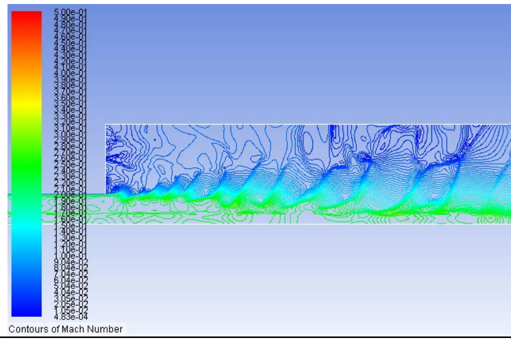


Figure 5. Contours of pressure and Mach number in the nozzle and in the combustion port

4. CONCLUSIONS

In the present work we have developed first a simplified HRM model. The flow model is based on the one-dimensional Euler equations with source terms, to which is adapted an empirical model for the fuel regression rate. The ANSYS Fluent is then used for the numerical simulation of the mutually interacting three-dimensional flow and other physical processes in a very rigorous manner. The numerical results obtained with both models show a good agreement.

ACKNOWLEDGMENTS

This paper was supported by the Romanian Space Agency, STAR Programme, research contract no. 35/2012- STRAC. A version of this work was presented at *3rd International Workshop on Numerical Modelling in Aerospace Sciences, NMAS 2015*, 06-07 May 2015, Bucharest, Romania.

REFERENCES

[1] B. Collburn, *A manual for hybrid propulsion system design*, 1st ed., Aerocon Systems, 2006.

[2] G. P. Sutton, O. Biblarz, *Rocket propulsion elements*, 7nt ed., John Wiley&Sons Inc., ISBN 0-471-32642-9, New York, 2009.

[3] F. Mingireanu, *Hybrid rocket motor internal ballistic model and oxidizer doping. Applications*, Proceedings of 4^{th} International Conference RAST 2009, ISBN:978-1-4244-3626-2, June 2009.

[4] M. Chiaverini and K. K. Kuo (ed.), Fundamentals of Hybrid Rocket Combustion and Propulsion, *Progress in Astronautics and Aeronautics*, American Institute of Aeronautics and Astronautics, Inc., Reston, Virginia, 2007.

[5] S. K. Godunov, (ed.), *Numerical Solution of Multi-Dimensional problems in Gas Dynamics*, Nauka Press, Moscow, 1976.

[6] A. Gaitonde, A Dual-Time method for Two-Dimensional Unsteady Incompressible Flow Calculations, *Int. J. Numer. Meth. Engng.*, **41**, pp.1153-1166, 1998.