On the Effects of a Flying Object inside a SRM with Finocyl Propellant Grain Geometry

M. Di Giacinto⋆, B. Favini⋆, E. Cavallini⋆ and F. Serraglia⋆⋆
⋆Dept. of Mechanical and Aerospace Engineering - Sapienza University of Rome
Via Eudossiana 18, 00184 - Rome, Italy
⋆⋆ESA-ESRIN VEGA Integrated Project Team
Via Galileo Galilei, 00044 Frascati, Italy

1. Introduction

Aim of this paper is to the analyze of the effects of a flying object, advected by the motor internal flow, on the internal ballistics of an aft-finocyl SRM, with typical characteristics of a stage or a small booster. The objective is to provide a preliminary study able to give an assessment of the effects on SRM internal ballistics of the outflow from the nozzle of pieces of the SRM components (i.e. expendable igniter) or, more realistically, pieces of thermal protections of relevant dimensions (with respect to the nozzle throat area of the SRM) detached from SRM components (floaters, igniter, casing thermal protections). The numerical simulation of the moving object will be performed with a Q1D unsteady model of the SRM internal ballistics, as presented in the following section. The idea is, in fact, to obtain from the simulations a feeling about the physics of the problem, without having to face too high computational costs and complexities for the simulation. In fact, the use of more simplified approaches like 0D models, both steady and quasi-steady, may imply the risk that too rough simplifications of the problem analyzed may brings to useless qualitative and quantitative comparisons with the experimental data. On the other hand, the computational costs and complexity of the problem faced with axisymmetric or 3D approaches is prohibitive and not useful for a preliminary study. For these reasons, the analysis will not cover the simulation of an object carried by the flow in the nozzle, with the scope to prove or not the possibility that the flying object can impact the nozzle itself (which in the typical flowfield conditions of the nozzle seems to be possible only for an object with a irregular - e.g. not a sphere). A grid convergence analysis for the computed solutions and a parametric analysis will be performed in order to achieve the objectives of the work.

2. Model for SRM Internal Flowfield carrying Big Inert Particles

The model considered for the simulation of an object of relevant dimensions carried by the flowfield is a Q1D unsteady model of SRM internal ballistics, properly adapted to account for the possible presence of a big particle in the flow. The model is a derivation of the SPIT model[1, 2, 3, 4, 5, 6] developed for a simplified study of the fluid/structural interaction during SRM ignition transient, presented Ref. [7], properly adapted for the study of the problem under investigation (Eq. (1)).

$$\begin{cases} \dfrac{\partial\left(\rho A_p\right)}{\partial t} + \dfrac{\partial\left(\rho u A_p\right)}{\partial x} = \underbrace{r_b P_b \rho_p}_{\text{propellant}} + \underbrace{\dfrac{\dot{m}_s A_p}{V}}_{\text{cavities}} \\ \dfrac{\partial\left(\rho u A_p\right)}{\partial t} + \dfrac{\partial\left[\left(\rho u^2 + p\right) A_p\right]}{\partial x} - p\dfrac{\partial A_p}{\partial x} = \underbrace{\dfrac{1}{2}\rho u^2 c_f}_{\text{friction}} + + \underbrace{\dfrac{\left(F_D\right) A_p}{V}}_{\text{particle}} \\ \dfrac{\partial\left(\rho e A_p\right)}{\partial t} + \dfrac{\partial\left[\left(\rho e + p\right) u A_p\right]}{\partial x} = -p\dfrac{\partial A_p}{\partial t} + \underbrace{r_b P_b \rho_p H_f}_{\text{propellant}} + \underbrace{\dfrac{\dot{m}_s A_p H_s}{V}}_{\text{cavities}} \end{cases} \quad (1)$$

For the droplet, the basic hypothesis is that it can be assumed inert and locally in thermal equilibrium with the surrounding flowfield. As a consequence the effects of the particle on the carrier flow are only related to the momentum equation, because of drag force acting on the particle itself and volume effects, because particle volume is not negligible. A further simplification assumed is that the particle is a sphere with imposed radius.

Under such hypotheses, the motion of the particle can be expressed as given by Eq. (2), using the classical set of Lagrangian equations for a droplet in a high Reynolds flowfield, in which only the drag term is retained, since virtual mass force, lift force and buoyancy force become negligible for the problem under investigation (big and heavy particle, $\rho_p/\rho \gg 1$)

$$\begin{cases} \dfrac{dx_p}{dt} = v_p & Re_p = \dfrac{\rho \left| u\left(x_p\right) - v_p \right| d_p}{\mu} \qquad M_p = \dfrac{\left| u\left(x_p\right) - v_p \right|}{a} \\ \dfrac{dv_p}{dt} = \dfrac{u\left(x_p\right) - v_p}{\tau_p} & \tau_p = \dfrac{4\rho_p\, d_p}{3C_D \left| u\left(x_p\right) - v_p \right|} \end{cases} \tag{2}$$

Now, since a strong deviation from the situation $Re_p = 1$ and $M_p \ll 1$ is expected, a correction to the classical Stokes solution has to be considered for the C_D evaluation. This is done employing the expression of the compressibility/high Reynolds correction provided in Ref. [8], that is a modification of the Clift-Gauvin drag coefficient expression [9].

$$C_D = \frac{24}{Re_p}\left(1 + 0.15 Re_p^{0.687}\right) H_M + \frac{0.42 C_M}{1 + \frac{42500 G_M}{Re_p^{1.16}}} \quad \text{for} \quad Re_p > 45 \tag{3}$$

In Eq. (3), the following coefficient are introduced.

$$\begin{cases} G_M = 1 - 1.525\, M_p^4 & \text{for} \quad M_p < 0.89 \\ G_M = 0.0002 + 0.0008\, \tanh\left(12.77\left(M_p - 2.02\right)\right) & \text{for} \quad M_p > 0.89 \end{cases} \tag{4}$$

$$H_M = 1 - \frac{0.2581, C_M}{1 + 5141, G_M} \tag{5}$$

3. Preliminary Results

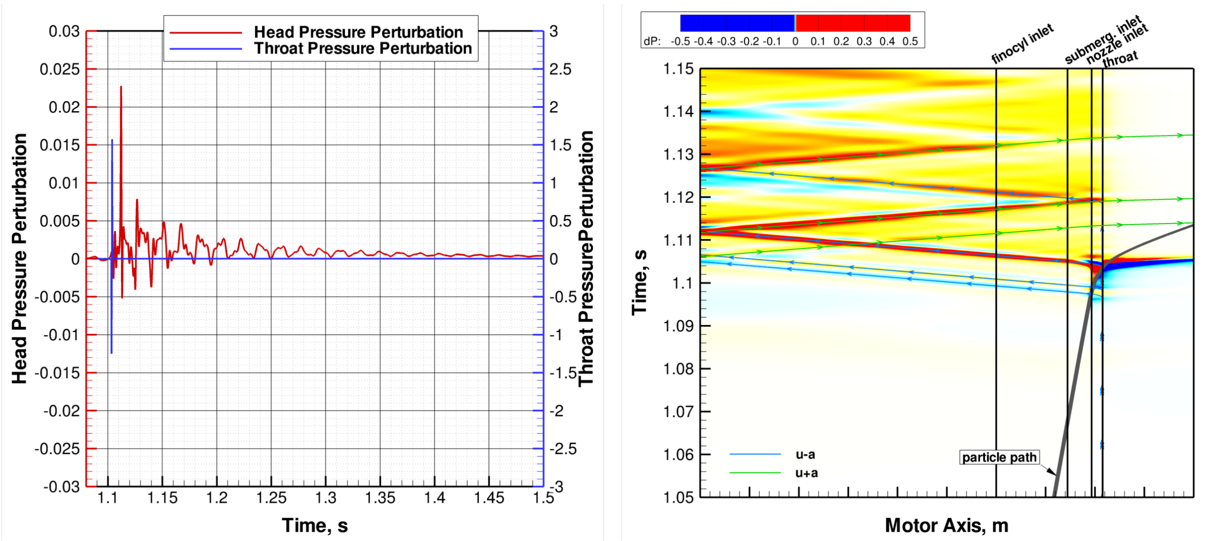
In this section, even if further simulations are going to be run and analyzed, preliminary results of the simulations already performed are briefly discussed.

The simulation has been set-up considering the internal geometry (propellant grain shape and nozzle profile) and the characteristics of a typical aft-finocyl SRM for stage/small booster-like applications.

Starting from a properly selected initial condition, once the steady state conditions of the SRM flowfield are attained, the particle is injected and advected within the flowfield.

Results of a case simulated, for which a grid convergence analysis has been performed in other to have confidence in the numerical solution achieved, are briefly depicted in Fig. 1 (zero time is the time in which the particle is injected in the flow). Fig. 1(a) shows the pressure perturbation, with respect the steady state solution, at the head end and at the nozzle throat. Fig. 1(b) depicts, instead, the evolution of the pressure perturbation in a *x*/*t* (*x*: motor axis) plane, where the path of the particle can be visualized, together with the characteristic lines $u - a$ and $u + a$. This last figure shows that the flowfield is almost unaltered by the motion of the particle within the bore of the SRM, before around the nozzle throat, a part from small level pressure waves emitted at the nozzle entrance section. In fact, the major pressure fluctuation due to particle motion in the flowfield (Fig. 1(a)) is directly caused by the pressure wave generated by the particle when it crosses the nozzle throat. Then, a system of pressure waves travels back and forth in the SRM and is damped in a short period of time. Looking at Fig. 1(a), moreover, the different scale of the pressure perturbation created when the particle crosses the nozzle throat, with respect to the one that reaches the head end of the SRM can be underlined.

The complete analysis of the results will be provided in the full paper, where a parametric analysis and a grid convergence analysis will be discussed.



(a) Head End vs. Nozzle Throat Pressure Perturbation Time History

(b) Pressure Perturbation Field (x, t)

Figure 1: Particle Effects on Pressure Field during the Steady State

Acknowledgements

This work is performed in the frame of the support activities of the Sapienza University of Rome - Dept. of Mechanical and Aerospace Engineering - SPqR Team to the VEGA program.

References

[1] Di Giacinto, M. and Serraglia, F., "Modeling of Solid Motor Start-up," AIAA Paper 2001-3448, July 2001, 37th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Salt Lake City, UT.

[2] Di Giacinto, M. and Serraglia, F., "Modeling of SRM Ignition Transient: Role of the Main Phenomena," AIDAA, September 2001, *XVI* Congresso Nazionale AIDAA, Palermo, Italy.

[3] Di Giacinto, M., *Numerical Simulation of Solid Motor Ignition Transient*, No. 5-ISICP-027-8-pp-MDG, Begell House Inc., New York, 2001, June 2002, 5th International Symposium on Special Topics in Chemical Propulsion, Stresa, Italy.

[4] Favini, B., Di Giacinto, M., and Serraglia, F., "Solid Rocket Motor Ignition Transient Revisited," *Proc. of the* 8th *Int. Workshop on Combustion and Propulsion*, June 2002, Pozzuoli, Italy.

[5] Favini, B., Serraglia, F., and Di Giacinto, M., "Modeling of Flowfield Features During Ignition of Solid Rocket Motors," AIAA Paper 2002-3753, July 2002, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Indianapolis, Indiana.

[6] Favini, B., Di Giacinto, M., and Serraglia, F., "Acoustic Phenomena During SRM Ignition Transient," IN-SPACE PROPULSION I-19032, Sept. 2003, 10-IWCP 10th International Workshop on Combustion and Propulsion, 21-25 September 2003, Lerici, La Spezia, Italy.

[7] Cavallini, E., Favini, B., Di Giacinto, M., and Steelant, J., "A Simplified Coupled Structural-Flowfield Analysis of Solid Rocket Motors Ignition Transient," *7th European Symposium on Aerothermodynamics for Space Vehicles, 9 May - 12 May 2011, Brugge, Belgium*, ESA Publications, 2011.

[8] Loth, E., "Compressibility and rarefaction effects on drag of a spherical particle," *AIAA journal*, Vol. 46, No. 9, 2008, pp. 2219–2228.

[9] Clift, R. and Gauvin, W. H., "Proceedings of CHEMECA 70," Vol. 1.