Numerical and experimental evaluation of the performance of a cavitating valve for the control of oxidizer flow in a hybrid rocket engine

Petr Dvořák and Rob Tijsterman***
**Institute of Aerospace Engineering, Faculty of Mechanical Engineering, Brno University of Technology*
Technicka 2, 616 69 Brno, Czech Republic, dvorak.p@fme.vutbr.cz
***Moog Bradford of The Netherlands*
De Wijper 26, 4726 TG, Heerle, The Netherlands, r.tijsterman@bradford-space.com

Abstract
Part of a collaborative European effort to advance throttleable hybrid rocket technology, Bradford Engineering is developing a cavitating flow control valve. In order to improve performance of the valve Brno University of Technology has performed extensive numerical simulation campaign. The paper describes design considerations of the valve assembly as well as characterization of the valve performance by experimental and numerical means.

1 Introduction

Both unmanned and manned missions to extra-terrestrial bodies require reliable means to perform soft and precise landing on the surface of a planet. On Mars, this is usually performed by a combination of deceleration parachute and rocket-thrusters [1]. Advanced functionality, such as obstacle avoidance and dynamic flight path adjustment are also increasingly being incorporated into the mission requirements. One of the most promising concepts to allow for deep-throttleability and hence soft landing is hybrid rocket engine, consisting of solid propellant and liquid oxidiser [1]. Unfortunately, such a technology has not been available within Europe so far [2].
To develop a deeply-throttleable hybrid engine is the primary objective of FP7 European Commission funded project SPARTAN (SPAce exploration Research for Throttleable Advanced eNgine). This effort is in agreement with International Space Exploration Coordination Group and ESA development guidelines [3], [4]. The three-year long project has commenced 1st March 2011. It is coordinated by Thales Alenia Space Italia S.p.A. and incorporates both commercial and academic partners from seven European countries.
During course of the project, hybrid propulsion technology based on HTPB (hydroxyl-terminated polybutadiene) solid propellant fed by hydrogen peroxide as an oxidiser has been selected due to a number of favourable characteristics - namely safety, minimum environmental impact, low life cycle costs, responsiveness, competitive performances, increased reliability, soft ignition and shutdown [2].
Outcome of the project, the SPARTAN demonstrator (aiming to raise the Technology Readiness Level of throttleable hybrid engines to level 6) is going to employ four of these rocket engines ensuring vertical deceleration from 30m/s to 2m/s , which is considered to be a nominal landing condition. The demonstration mission profile is designed so as to reproduce the Mars landing dynamic requirements. To achieve this goal, the engines need to be capable of operation with 10:1 thrust ratio.
In hybrid engine, modulation of thrust is achieved by controlling the oxidizer inflow to the combustion chamber. In this application, fast and accurate control is vital. Mass flow changes must be accomplished within a few hundred milliseconds to an accuracy of within 1% or 2% of the full range at most. Not only does this require a fast actuator, it also requires an actuation with excellent open-loop characteristics. The run-of-the-mill valve with a flow characteristic dependent on the pressure drop over the valve will simply not suffice for this application: the exact flow rate depends on the outlet pressure as well as the inlet pressure to the valve. Since we have a rocket engine firing downstream of the valve, we not only have to cope with the start-up transient, but also any combustion induced oscillations in back pressure which are propagating upstream to the valve outlet. This implies a back-pressure independent valve and valves that are operating in cavitation are back-pressure independent. We therefore decided to proceed with a cavitating valve as the baseline.
This approach is not new. In fact, cavitating valve designs have been used in throttleable rocket engines at least since the Apollo program, where they were used for thrust modulation of the Lunar Module Descent Engine [5]. Most recently, cavitating valves were used in the Mars Science Laboratory SkyCrane [6].
The major design issue for our cavitating valve was limiting the pressure drop over the valve. Initial development tests were immensely successful, but for the issue of pressure recovery. While this type of valve should be capable of

pressure recovery well over 80 %, the figures that we obtained were much lower. This necessitated the need to look into the design of the venturi section in detail and consequently Institute of Aerospace Engineering, Brno University of Technology, was invited to provide numerical simulation support of the process.
The paper focuses on the design considerations and experimental as well as numerical characterization of the flow control valve.

2 Valve design overview

The valve is based on a pintle in nozzle configuration, whereby a moveable pintle is inserted into a venturi nozzle. The mass flow through the valve is directly controlled by varying the position of the pintle with respect to the nozzle.

By applying Bernoulli's principle for incompressible flow between valve inlet and valve throat, the basic performance relation of a cavitating venturi is obtained:

$$m = c_d\, A_t\, \mathrm{sqrt}(2\, \rho\, (p_i - p_v)) \qquad (1)$$

where A_t stands for throat area [m^2], c_d for coefficient of discharge [-], m for mass flow rate [kg/s], p_i for inlet pressure [Pa], p_v denotes outlet pressure [Pa] and ρ is for density [kg/m^3].
For the above relation to hold true, the pressure at the valve outlet must be sufficiently low to ensure that the liquid is cavitating in the valve throat.

Modulation of mass flow rate is achieved through variation of throat area. In contrast to non-cavitating valves, where mass flow rate is a function of pressure drop over the valve, for a cavitating venturi the mass flow rate is a function of the square root of the difference in pressure at valve inlet and vapour pressure of the liquid medium. Given that the temperature of the liquid remains relatively constant, so does the vapour pressure. The mass flow rate through a cavitating valve in a particular configuration is effectively a function of the inlet pressure.

The mass flow range to be covered by the valve required development of a dedicated flow test setup at Bradford Engineering. The baseline oxidizer, hydrogen peroxide, was deemed too troublesome to handle for development test purposes. Instead, water was selected for testing. From a theoretical performance viewpoint, influence of the medium on the performance of the valve is governed by the density and vapour pressure.
The influence of vapour pressure is minor. For hydrogen peroxide 87.5 % at 20 °C vapour pressure is ~400 Pa [7], compared to ~2350 Pa [8] for water. At the nominal operating pressure of 65 bar, this results in m_{H2O2} / m_{H2O} = sqrt((6500000 – 400) / (6500000 – 2350)) = 1.00015, or a change of 0.015 % in the mass flow rate. This is effectively negligible.

The density has a significant effect on the performance of the valve however. With a density of 1376 kg/m^3 at 20 °C for hydrogen peroxide against 998 kg/m^3 of water [9], the mass flow rate will increase sqrt(1376 / 998) = 1.17 or 17 % with respect to that measured in a water based test.

The valve is designed such that pintles and nozzles are easily exchangeable. This offers flexibility in flow control range by modifying the throat diameter. It also allows modification of flow rate control ability by altering pintle shape.
For the initial experimental campaign, a pair of pintles was produced, as depicted in Figure 1. One of the pintles was manufactured with a conical tip with half-angle 10 °, while the other featured a parabolic contour. The latter was expected to give a desirable linear control characteristic.
A pair of nozzles was also produced (Figure 2). One of the nozzles was manufactured with a strictly converging-diverging section. Both nozzles have a convergent half-angle of 22 ° and converge to a throat 2.6 mm in diameter. The divergent for nozzle 1 has a half-angle of 15 ° and immediately follows the convergent. The divergent for nozzle 2 has a half-angle of 20 ° and is separated from the convergent by an 8.2 mm long straight section. Both nozzles were equipped with pressure ports to enable pressure measurements on different locations inside the nozzles.

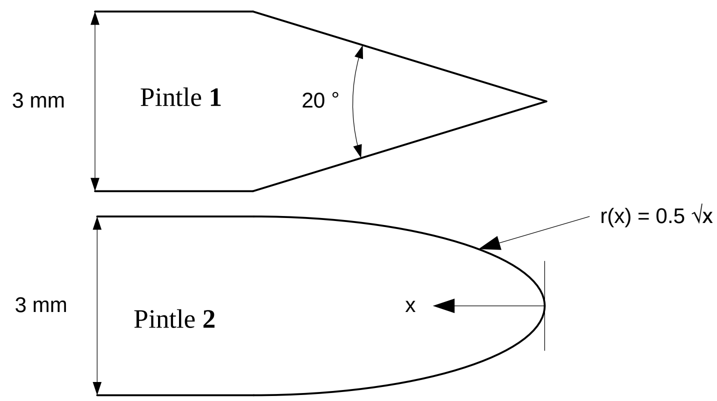

Figure 1 Pintle geometry comparison

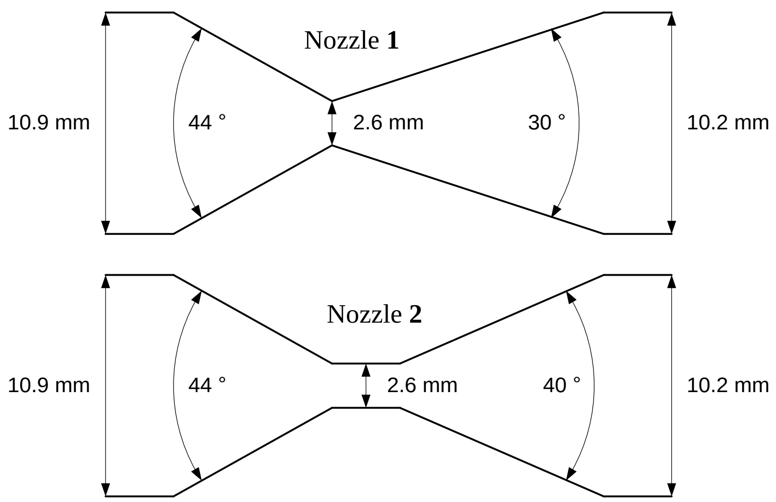

Figure 2 Nozzle geometry comparison

The operational scenario for the valve foresees cavitation over the entire flow range.

3 Experimental characterization

3.1 Experimental setup

The test setup is relatively straightforward. It consists of a 25 litre pressurized water reservoir feeding a test section containing the device under test. Pressure is supplied in the form of nitrogen from a standard 200bar 50l gas cylinder. The pressure is regulated before being supplied to the reservoir. Water is discharged into a container which is open to atmosphere. The facility is sized based on a flow rate of 0.6 kg/s to be maintained over 30 seconds at a nominal supply pressure of 65 bar. Monitoring equipment consists of upstream and downstream pressure transmitters and inline Coriolis flow meter upstream of the test section. The test setup schematic is given in Figure 3.

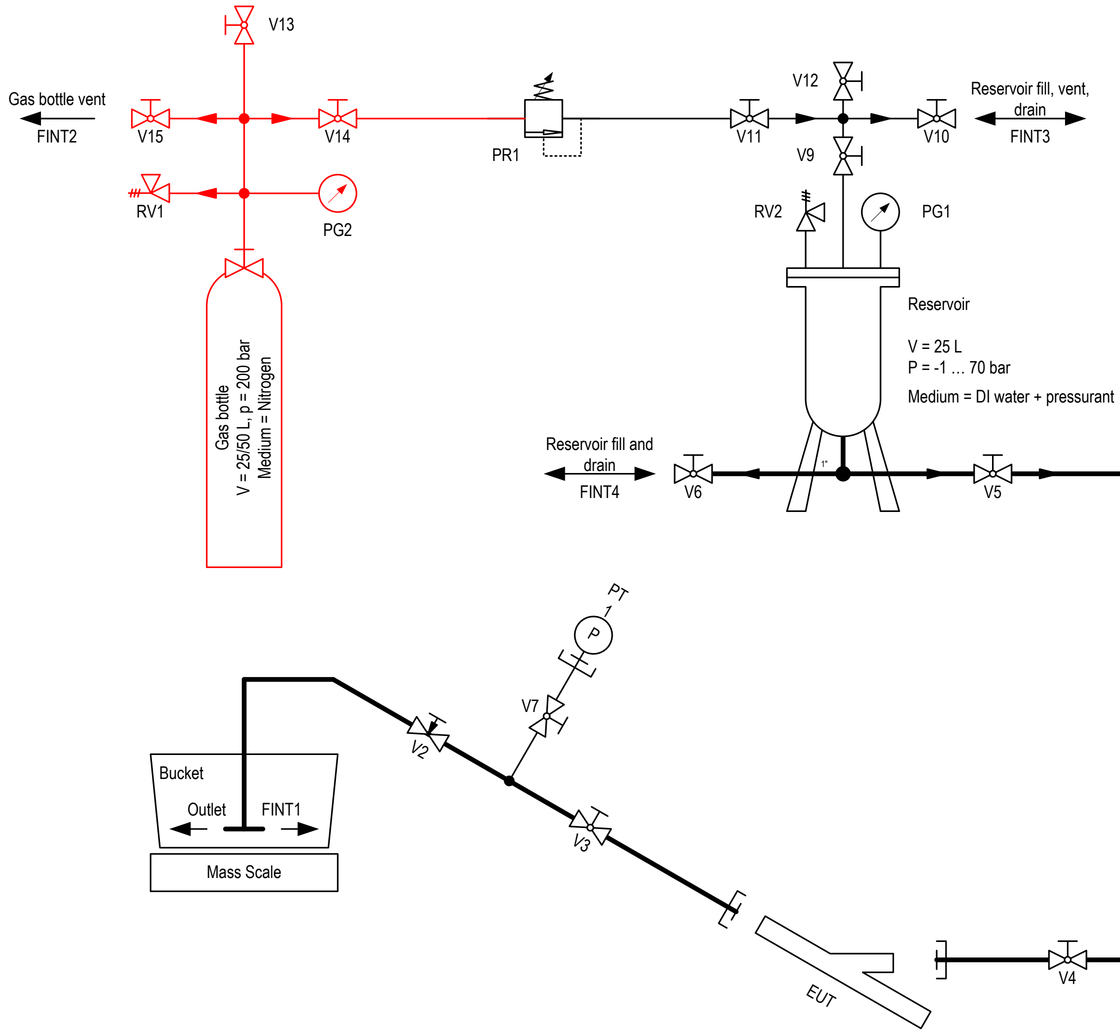

Figure 3 Schematic of the test setup

The procedure consists of setting the valve stroke to a specific value, then pressurizing the valve, with back pressure controlled by a downstream valve, and enabling flow through the valve. This procedure is performed for pintle positions of 0.70 mm, 1.75 mm, 3.50 mm, 5.25 mm, and 7.00 mm open as illustrated in Figure 7. For each pintle position, flow rates and pressures have been measured and recorded for a number of back pressures ranging from 2 to 60 bar. Data points have been gathered in the direction of increasing pintle position starting from the smallest pintle position. Hysteresis from the pintle drive mechanism is thereby eliminated.

3.2 Experimental results

The performance parameters of interest consist of the valve discharge coefficient, pressure recovery and linearity of valve response. As the nozzle 2 / pintle 2 configuration clearly outperformed all other variants in all aspects [10], only data characterizing this combination are presented further.

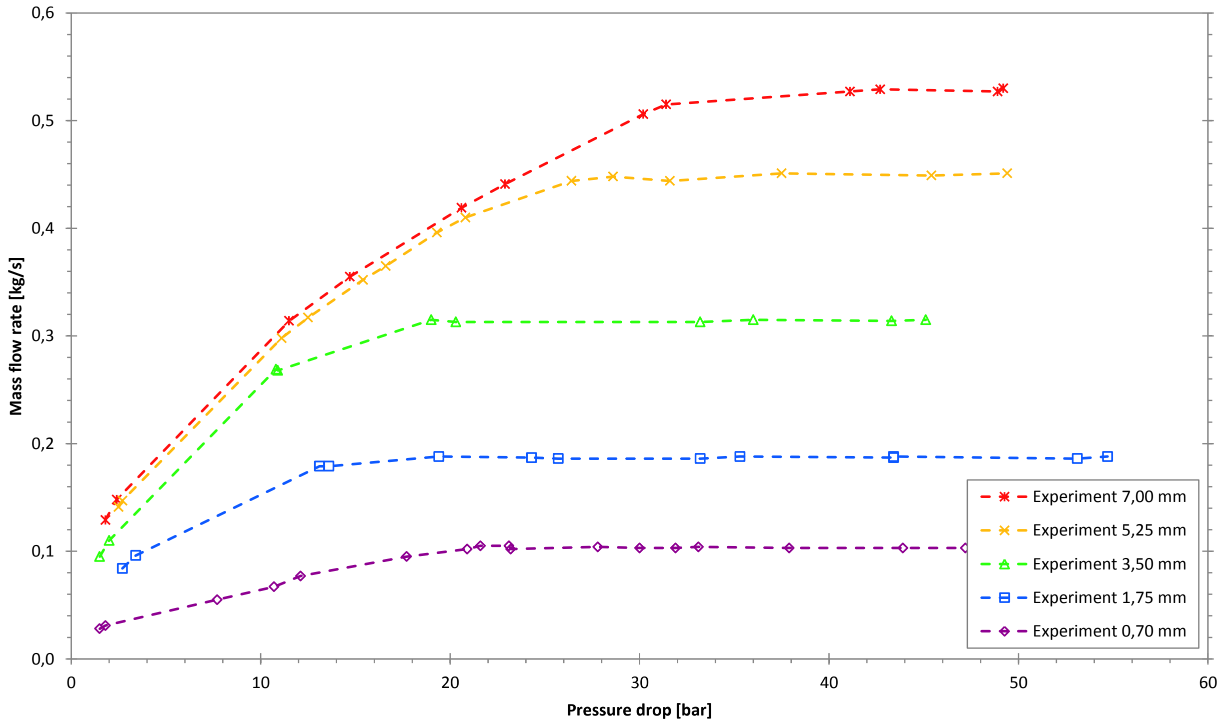

Figure 4 Test results of Nozzle 2/ Pintle 2 combination for various pintle strokes

The pressure drop required to achieve back-pressure independence is larger than initially expected – approximately 30 bar for the 7mm stroke case compared to 10 bar anticipation. Given the differences between nozzle 1 and nozzle 2 this is most likely a nozzle design issue. In particular, the changes in flow direction are smaller for nozzle 2 than for nozzle 1, due to the inclusion of a straight section in nozzle 2. Therefore, the current effort to minimize pressure drop needed for the valve to operate in cavitating mode is aimed at redesign of nozzle 2.

4 Numerical simulation

4.1 Numerical setup

Initial examination of the problem revealed four combinations of boundary condition geometries (pintle with conical and parabolic contours combined with two convergent divergent nozzle variants) with five blockage configurations each, totalling in 20 required simulation combinations. For each combination, study of behaviour based on differing pressure drop across the valve was required.

Therefore, the simulation was decided to be treated as 2D, axi-symmetric problem to allow for rapid solution turnaround and swift reaction to Bradford Engineering requirements. Furthermore, the baseline computations were treated as steady-state to mimic the real stabilised flow at constant boundary conditions.

Spatial discretization of the domain was performed within the Ansys ICEM environment to obtain multi-block fully structured meshes for the respective geometry configurations. Typically the mesh (Figure 5) consists of approximately 150 000 cells. Compared to the first simulation campaign [11], [12], the near-wall modelling was improved, aiming for y+ lower than 5 to support the Enhanced Wall Treatment of the k-epsilon turbulence model. However, no significant deviation of results has been observed compared to the original unrefined meshes (y+ up to 80).

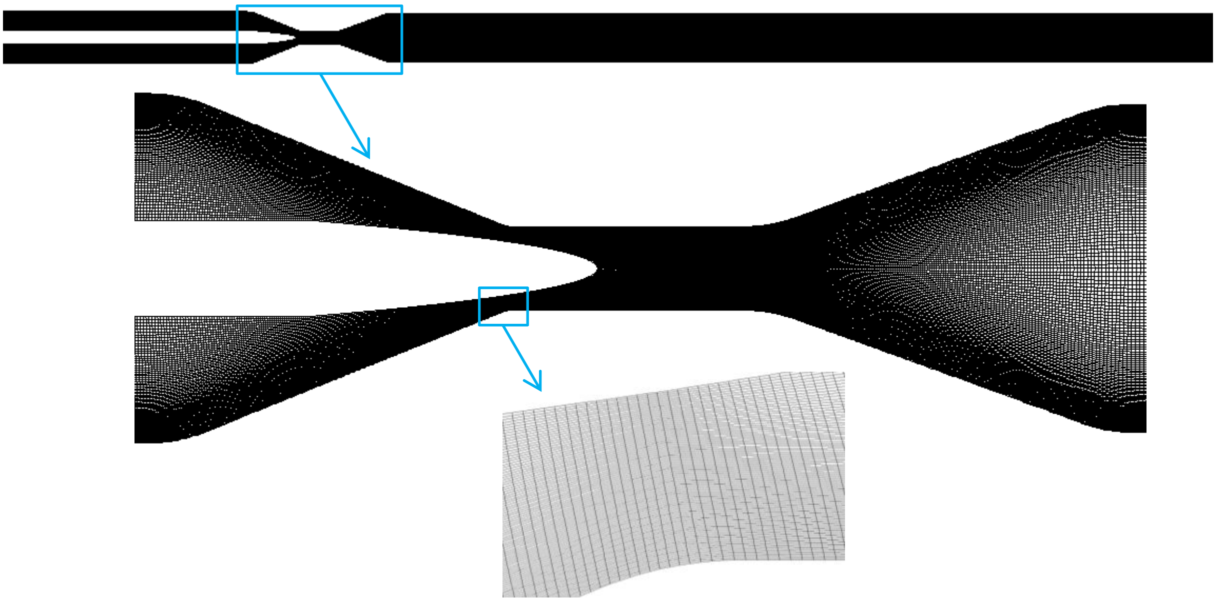

Figure 5 Fully structured computational grid

During the simulation, incompressible Reynolds-Averaged Navier-Stokes equations are solved in their steady state form in the frame of Ansys Fluent 14.5 pressure-based solver. The partial differential equations are treated by finite-volume method. Coupled iterative approach to solve the system of algebraic equations is deployed. Second order upwind scheme is selected to discretize the convective as well as diffusion terms of the governing equations. Green-Gauss Node Based method is used to treat the gradient data.
The simulation is isothermal at 293.15K. The cavitating multiphase flow is modelled by a mixture model of water and water vapour. This is in agreement with the breadboard experimental evaluation. Computations with hydrogen peroxide are part of the on-going work as a support to further stages of experimental characterization of the valve. Comparison of the material properties for the considered temperature is given in Table 1.

Table 1 Material properties used in CFD code

Material	**Liquid density [kg/m^3]**	**Liquid viscosity [kg/m-s]**	**Vapour density [kg/m^3]**	**Vapour viscosity [kg/m-s]**
H_20	998	0.001003	0.0256	1.26e-06
H_2O_2 87.5%	1376	0.001300	0.0030	9.40e-06

To consider the cavitation phenomena, mass transfer mechanism is defined from water-liquid phase to water vapour phase. The code is modelling bubble dynamics with generalized form of Rayleigh-Plesset equation [13], [14], deploying Schnerr-Sauer [15] cavitation model with the parameters set as detailed in Table 2. Parameters for hydrogen peroxide are given for reference only and are not considered in the frame of this setup.

Table 2 Schnerr-Sauer cavitation model parameters

Material	**Vaporisation pressure [Pa]**	**Bubble Number Density [-]**
H_20	2350	1e+13
H_2O_2 87.5%	400	1e+13

Turbulence is accounted for by standard k-ε model with Enhanced wall treatment deployed as recommended by [16] in favour of widely used RNG k-ε formulation ([17], [18], [19] among others). Stability of the deployed model proved to be superior to other models.
Surfaces of the pintle and nozzle are modelled by no-slip wall boundary condition. The domain is fed by pressure-inlet boundary condition with constant value of absolute pressure (62 bar) to account for pressure regulator droop and piping pressure losses of the experimental setup [10]. Outlet is modelled as pressure-outlet with defined constant value of gauge pressure. The computational cases are initialized gradually with outlet pressure values ranging from 60 bar to 24 bar in steps not higher than 2 bar. The axisymmetric computational domain with boundary condition specification is depicted in Figure 6.

Mass-flow-rate at the outlet as well as the pressures averaged across inlet and outlet boundaries are monitored during the computation in addition to the residual values in order to assess the stability and convergence of the computation. Flow variables across the domain are initialized with the hybrid routine and solved. Typically, the solution arrives at a stable state within 5000 iterations.

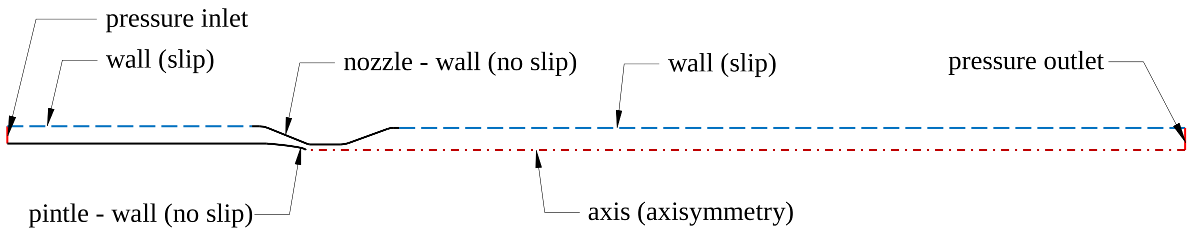

Figure 6 Simulation domain boundary condition assignment

Five various valve openings were simulated, representing the pintle strokes used during the experimental investigation as given in Figure 7.

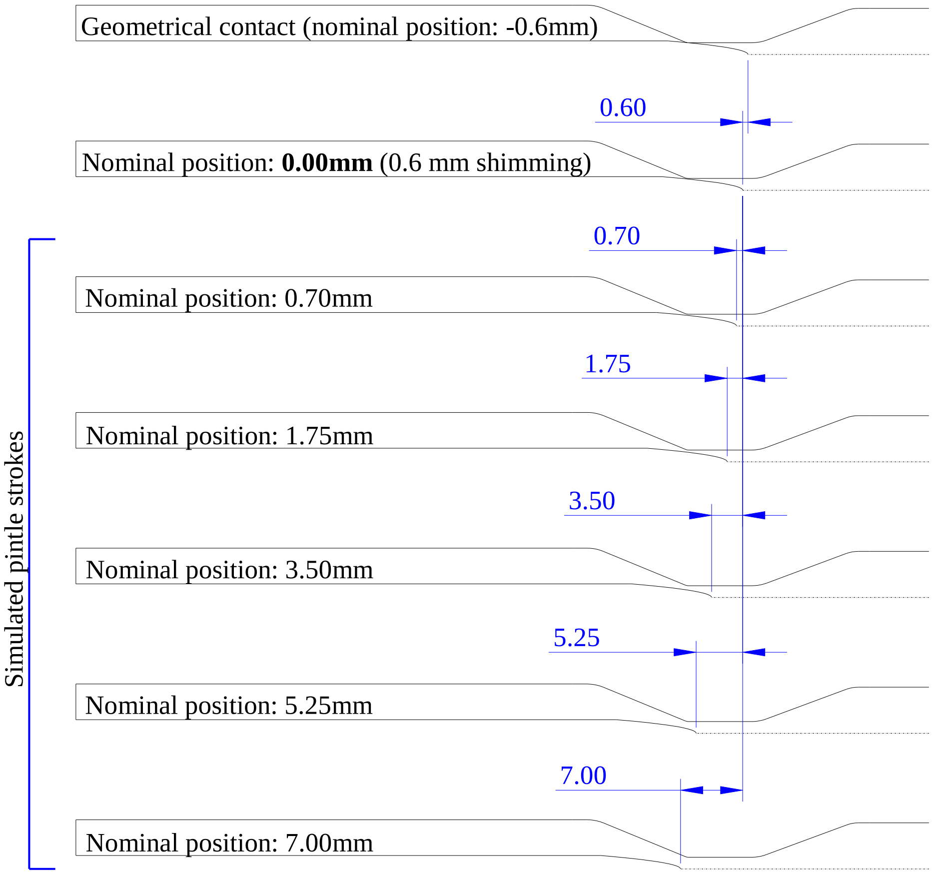

Figure 7 Respective geometrical variants: valve pintle stroke

4.2 Numerical results

All nozzle/pintle combinations are simulated, however only results characterizing nozzle 2/ pintle 2 configuration are presented further, as this setup was identified as the best performing during the experimental phase. This was also confirmed by the numerical simulation. More than 250 computational cases were created, solved and post-processed during this campaign, amounting in more than 2100 CPU-hours of computation cluster time.

Following a check of convergence and stability of the solution, parameters of each simulated case were extracted. Several cases with higher pressure drops across the domain displayed instabilities/oscillations in the outlet pressure values, however mass flow rate was not affected by this. The oscillatory behaviour seems to be in agreement with observations of [20] and is also mentioned by [21]. Switching the simulation code into double precision mode did not affect this phenomena, it is therefore probably result of the physics-modelling rather than round-off error. However this issue concerns only minority of the cases and the integral values of interest are not affected, therefore no further attention was paid to the phenomena.
Mass flow rate of the nozzle 2/pintle 2 geometry as a dependency of pressure drop over the valve is presented in Figure 8, where experimental data are plotted as well to enable direct comparison of the results.

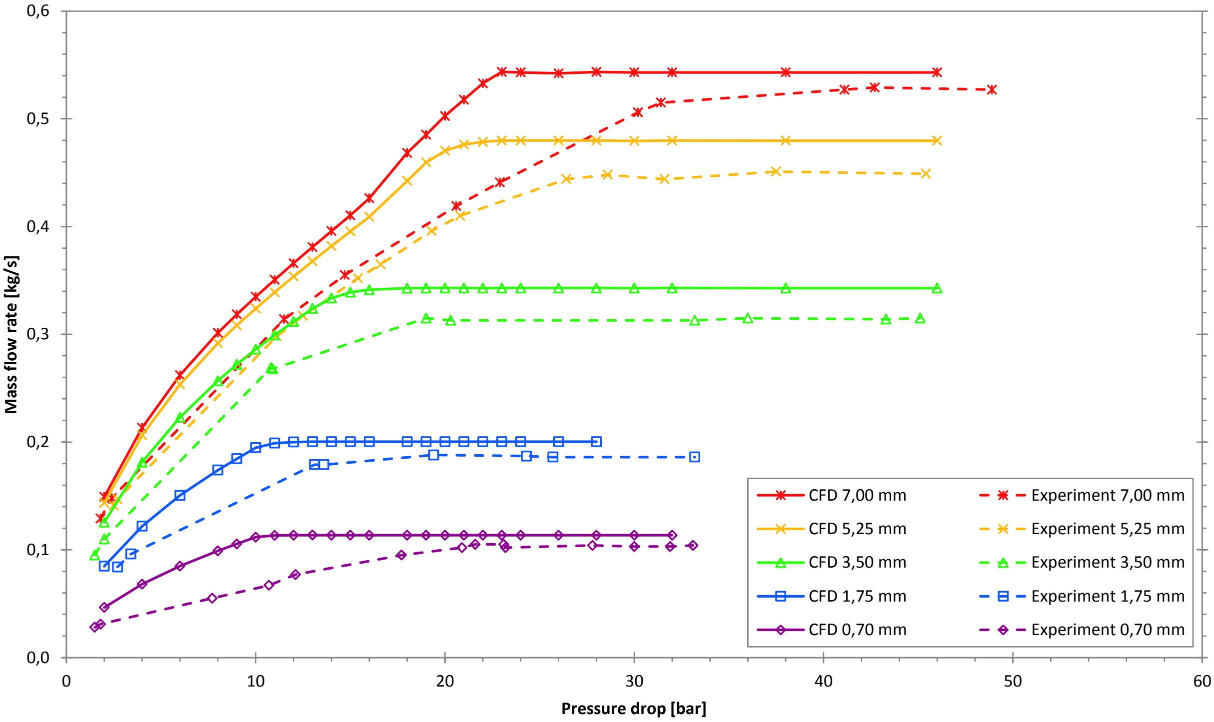

Figure 8 Nozzle 2/Pintle 2 performance as provided by CFD and experiment

The results display very good agreement in qualitative terms. Quantitatively, numerical simulation predicts the valve to enter the cavitating mode of operation at slightly more favourable values than measured experimentally, however the difference is tolerable and consistent across the whole simulation space. The discrepancy might be caused both by simplifications in simulation domain and numerical code as well as by inaccuracies during the measurement and manufacturing of the physical components.
Comparison of the obtained numerical results with computation performed by Lazzarin [22] at the University of Padua seems to be favourable. The code used was Ansys CFX and for the two evaluated configurations (100% and 50% opening) differences in forces acting on pintle were 3.5% for both cases. Differences in mass flow rates were 4% and 9%, respectively. This is considered a good agreement between the numerical codes.

5 Conclusion

Flow control valve has been designed to modulate oxidizer flow for hybrid rocket engine. The cavitating venturi concept of the valve proved to fulfil the challenging requirements of the application. This has been confirmed both by experimental and numerical evaluation of the various geometry configurations considered. The only parameter that lacks behind the expectations is the pressure drop needed for the valve to enter the cavitating mode – at approximately 30 bar for the fully opened valve, it is 20 bar higher than anticipated.
Reasonable agreement between the experiment and numerical simulation has been found, enabling the CFD to be used during optimization of the valve geometry, in order to lower the pressure drop needed for the valve to operate in fully cavitating mode.

Extensive numerical parametric study is currently being executed and will be published shortly. Subsequently, experimental and numerical characterization of the valve with hydrogen peroxide as a working media will be performed to enable the valve to be flight tested in the Spartan lander demonstrator.

6 Acknowledgements

The research leading to these results has received funding from the EC FP7 project SPARTAN (SPA.2010.2.1-04 GA n. 262837).

7 References

[1] Ball, A. J., Garry, J. R. C., Lorenz, R. D., Kerzhanovich, V. V. Planetary Landers and Entry Probes. Cambridge, United Kingdom: Cambridge University Press, 2007. 978-0-521-82002-8.

[2] Parissenti, G., Pessana, M., et. al. Throttleable Hybrid Engine for Planetary Soft Landing. *4th European Conference for Aerospace Sciences (EUCASS) proceedings*, 2011.

[3] Wilson, A. ESA Achievements. More than thirty years of pioneering space activity. BR-250. 3rd. Noordwijk, The Netherlands: ESA Publications, 2005. 92-9092-493-4.

[4] International Space Exploration Coordination Group (ISECG). Annual Report: 2009. ESA. Europe, 2009.

[5] Cherne, J. M. Mechanical Design of the Lunar Module Descent Engine. TRW Systems, 1967.

[6] Udomkesmalee, S., Hayati, S. A. Mars Science Laboratory Focused Technology Program Overview. *IEEE Aerospace Conference, Big Sky, Montana.* 03/2005.

[7] van Laar, J. J. The Vapor Pressure of Binary Mixtures. *Z. Phys. Chem.* 1910, 72(723).

[8] Wagner, W., Pruss, A. Vapour pressure of water at temperatures between 0 and 360°C. *J. Phys. Chem. Reference Data.* 1993, 22, 783-787.

[9] Linstrom, P. J., Mallard, W. G., eds. NIST Chemistry WebBook. NIST Standard Reference Database. 69.

[10] Tijsterman, R. Breadboard Model Flow Control Valve Test Sequence 1 Test Report. SPAR-FCV-BE-RP-0001. Moog Bradford. Heerle, The Netherlands, 09/2012.

[11] Dvorak, P., Hradil, J. Numerical Simulation of Cavitation Valve Flow. In: Polonyiova, L., ed. *Techsoft Ansys User Meeting 2012 conference proceedings.* Praha, Czech Republic: Techsoft Engineering, 2012, pp. 33-39.

[12] Dvorak, P. Spartan Flow Control Valve: Computational support of the cavitating venturi development. LU02-2013-SPA.AD. Brno University of Technology, Faculty of Mechanical Engineering. Brno, Czech Republic, 01/2013.

[13] Brennen, C. E. Cavitation and Bubble Dynamics. New York, USA, 1995. Oxford engineering science series. 44. 0-19-509409-3.

[14] Franc, J.-P. The Rayleigh-Plesset equation: a simple and powerful tool to understand various aspects of cavitation. In: D'Agostino, L. and Salvetti, M. V., eds. *Fluid Dynamics of Cavitation And Cavitating Turbopumps.* Udine, Italy: Springer, 2007.

[15] Schnerr, G. H., Sauer, J. Physical and Numerical Modeling of Unsteady Cavitation Dynamics. *Fourth International Conference on Multiphase Flow, New Orleans, USA.* 2001.

[16] Ansys. ANSYS 13 Help. FLUENT Documentation. 13th ed. Ansys, Inc. Canonsburt, USA, 2010.

[17] Blejchar, T. Matematicke modelovani nestacionarniho proudeni, kavitace a akustickych projevu v hydraulickem ventilu. Doctoral thesis. Advisor: Kozubkova, M. Ostrava University of Technology, Faculty of Mechanical Engineering. Ostrava, Czech Republic, 2006.

[18] Frölich, K. Modeling of cavitating flow. Master thesis. Advisor: Rudolf, P. Brno University of Technology, Faculty of Mechanical Engineering. Brno, Czech Republic, 2010.

[19] Rautova, J., Kozubkova, M. Matematicke Modelovani Proudeni Stlacitelne Kapaliny s Kavitaci. In: Polonyiova, L., ed. *Techsoft & SVS FEM Ansys User Meeting 2009 proceedings*.

[20] Bily, D., Kozubkova, M. Dynamics Behaviour of The Cavitation Field. *Journal of Applied Science in the Thermodynamics and Fluid Mechanics.* 2011, 9(2).

[21] Oprea, A. I., Bulten, N. Cavitation Modelling using RANS approach. *Warwick Innovative Manufacturing Research Centre 3rd International Cavitation Forum 2011proceedings.* Warwick, UK: University of Warwick, 2011.

[22] Lazzarin, M. Pintle CFD Simulation. SPA-DII-RPT-0003. University of Padua. Padua, Italy, 05/2012.