Validation of a numerical strategy to simulate the expansion around a plug nozzle

Marco Daniel Gagliardi[1,a], Luca Fadigati[1,b], Nabil Souhair[1,c] and Fabrizio Ponti[1,d *]

[1]Alma Propulsion Lab, Alma Mater Studiorum-Università di Bologna, Via Fontanelle 40, 40121 Forlì, FC, Italy

[a]marco.gagliardi2@studio.unibo.it, [b]luca.fadigati2@unibo.it, [c]nabil.souhair@unibo.it, [d]fabrizio.ponti@unibo.it

Keywords: Aerospike Engine, Plug Nozzle, HLLC Scheme, Numerical Simulation, OpenFOAM

Abstract. Rocket engines currently use traditional bell-shaped nozzles that have a fixed area ratio and can only operate at maximum efficiency at a given altitude. Plug nozzles have been proposed as an alternative solution to achieve higher performance over a larger altitude range. Unlike bell nozzles, the flow is free to expand along the plug, as it is no longer surrounded by solid boundaries. Therefore, plug nozzles can adapt to different altitudes by expanding the flow to ambient pressure, resulting in continuous altitude adaptation. Due to the high surface area that needs to be cooled, one of the main challenges of plug nozzle design is thermal management. However, the introduction of aerospike geometry, which is essentially a truncated plug nozzle, has helped mitigate this issue. Simulating an aerospike engine is challenging due to the interaction between the plume and the external flow, which is necessary to accurately predict thrust. In this work, a numerical strategy for predicting the performance of an aerospike engine, during a static fire, was developed and validated.

Introduction

Rocket engines commonly employ traditional bell-shaped nozzles with fixed area ratios, limiting their maximum efficiency to a specific altitude corresponding to the design Nozzle Pressure Ratio (NPR). Therefore, bell shaped nozzles operate sub-optimally for a significant portion of a launcher's flight. Various solutions have been proposed to address these limitations, but none have proven suitable for practical flight operations. In the 1950s, plug nozzles and aerospikes were introduced to achieve higher performance across a broader altitude range. Unlike other nozzle concepts, they offer continuous altitude adaptation. The advent of additive manufacturing techniques has sparked new developments in aerospike technology worldwide, as economically viable processes can address the geometrical complexity of the engine. Several research groups are actively exploring aerospike technology. The Beijing University of Aeronautics and Astronautics has developed an optimization method to design aerospike contours that maximize total impulse from sea level to the design altitude [1]. Technische Universität Dresden has studied thrust vectoring control systems and ceramic additive manufacturing techniques [2, 3]. Accurate thrust prediction in aerospike engines requires considering the interaction between the plume and external flow, posing a significant simulation challenge. This paper focuses on the development and validation of a numerical strategy aimed at predicting the performance of an aerospike engine. By utilizing advanced numerical simulations, the aim is to obtain crucial information impractical to measure during physical tests, such as pressure distribution along the plug and in the plume. The paper is divided into two main sections: the first one describes the developed model, the second one the results obtained and the validation using experimental data coming from literature.

Model structure

The CFD software employed to carry out the simulation is OpenFOAM, the chosen solver is *dbnsTurbFoam* that is contained in foam-extend. It has been chosen because it is considered particularly suitable for simulating supersonic turbulent flows and results the only compressible solver exploiting the Harten-Lax-van Leer-Contact (HLLC) approximate Riemann solver, crucial to correctly capture shock waves without smearing them [4]. In [5] the fidelity of *dbnsTurbFoam* is assessed: taking as references the Sod's shock tube (analytical solution, [6]) and the Onera S8 Transonic Channel (experimental data, [7]) scenarios, it is shown how *dbnsTurbFoam* is suitable to study high speed compressible flows. Turbulence is taken into account by means of the k-ω SST model developed by Menter [8]. A key feature of the SST model is the implementation of the stress limiter parameter, a_1, in the definition of the eddy viscosity ν_t [9].

$$\nu_t = \frac{a_1 k}{max(a_1\omega;\ \Omega F_2)} \tag{1}$$

Such formulation of the eddy viscosity has been considered in the model in order to better account for the transport of the turbulent shear stress inside the boundary layer of compressible flows, largely improving the performance in case of adverse pressure gradients. Ω is the absolute value of vorticity, and F_2 is a blending function, equal to one for boundary layer flows and to zero for free shear layers, that, in this latter case, allows to return to the original definition $\nu_t = k/\omega$ [8]. The default value employed by Menter for a_1 is 0.31.

Results

The experimental work of S. B. Verma and M. Viji [10] consists in testing a linear plug nozzle. The nozzle is fed with air stored in a tank at ambient temperature and pressure, but both values are not explicitly provided in the reference paper; in this work, they have been assumed of 1 bar, as done also in [11], and 300 K, respectively. Fig. 1 shows the experimental setup and the nozzle geometry as given in [10]; the design Nozzle Pressure Ratio for the internal nozzle is 14, and the overall nozzle design NPR is 36, corresponding to a Mach 3±0.1 exit flow; all tests have been performed in over-expansion conditions. The nozzle is provided with nine pressure ports distributed along the plug axis, as shown in Fig. 1, that allow measurements for the reconstruction of the time-averaged pressure distribution along the spike wall.

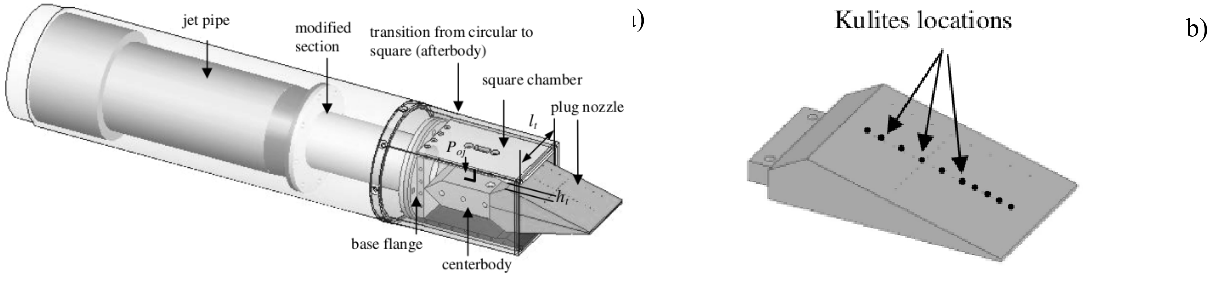

Fig. 1: Experimental setup for the analyses conducted in [10]: a) shows a section of the experimental setup, b) shows the position of the pressure measurements on the plug surface. Images from [10]

The simulations have been run until the jet achieved stationary conditions, and the obtained results have been compared with the data available in the two reference papers ([10] and [11]) in terms of flow topology and time-averaged pressure distribution along the spike wall. In Fig. 2, for example, the numerical results at NPR = 3.1, from [11] are compared with those obtained in this work. Fig. 3 shows the comparison between the experimental result and the ones from the simulations. On the horizontal axis, there is the coordinate along the spike normalized by L = 0.1m, that is the length of the spike itself from the throat section to the tip. The results are very close to the experimental.

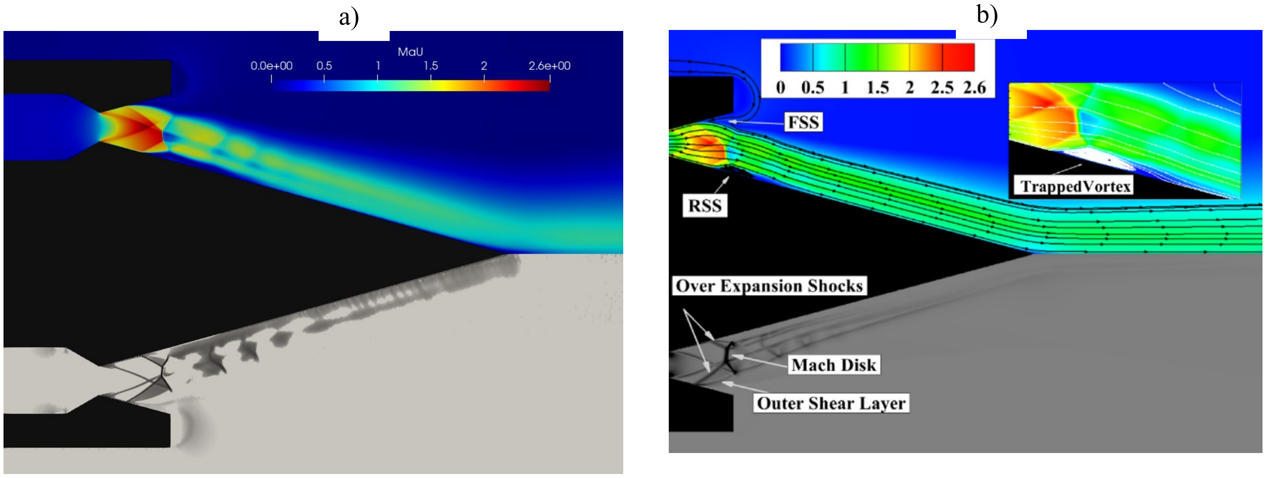

Fig. 1: A comparison between the numerical results provided by OpenFOAM (a) and ANSYS Fluent (b) at NPR = 3.1 [11].

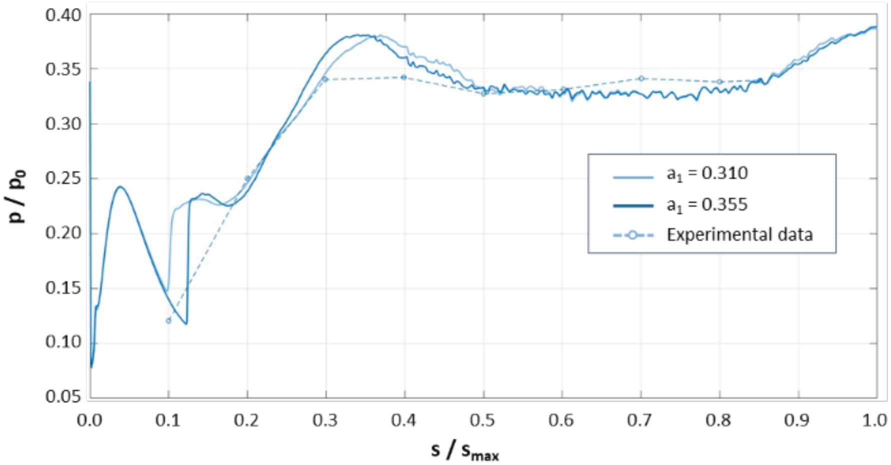

Fig. 3: Experimental spike wall pressure distribution (dots and linear interpolation) versus its numerical counterpart computed in OpenFOAM using two different values of the stress limiter in the turbulence model (NPR = 3.1).

Conclusion

The paper presents the definition of a model capable to simulate the supersonic flow at the exit of a plug nozzle. The simulation environment has been developed in OpenFOAM and based on the solver *dbnsTurbFoam*. The $\boldsymbol{k-\omega}$ SST model has been used to simulate flow turbulence and different parametrizations of the model have been discussed and simulated. The model has been validated using experimental data found in literature, collected on a linear plug nozzle, both in terms of pressure measurements on the plug wall, and in terms of Schlieren pictures of the flow. From the results that have been obtained, despite all the uncertainties associated to the reference experimental setup, the performance of the solver can be definitely considered appropriate.

References

[1] Wang C-H, Liu Y, Qin L-Z (2009) Aerospike nozzle contour design and its performance validation. Acta Astronaut 64:1264–1275. https://doi.org/10.1016/j.actaastro.2008.01.045

[2] Sieder-Katzmann J, Propst M, Tajmar M, Bach C (2021) INVESTIGATION OF AERODYNAMIC THRUST-VECTOR CONTROL FOR AERO- SPIKE NOZZLES IN COLD-GAS EXPERIMENTS

[3] Schwarzer-Fischer E, Abel J, Sieder-Katzmann J, et al (2022) Study on CerAMfacturing of Novel Alumina Aerospike Nozzles by Lithography-Based Ceramic Vat Photopolymerization (CerAM VPP). Materials 15:3279. https://doi.org/10.3390/ma15093279

[4] Toro EF (2019) The HLLC Riemann solver. Shock Waves 29:1065–1082. https://doi.org/10.1007/s00193-019-00912-4

[5] Chandramouli S, Gojon R, Fridh J, Mihaescu M (2017) Numerical Characterization of Entropy Noise With a Density Based Solver

[6] Sod GA (1978) A survey of several finite difference methods for systems of nonlinear hyperbolic conservation laws. J Comput Phys 27:1–31. https://doi.org/10.1016/0021-9991(78)90023-2

[7] Delery J (1978) Analyse du décollement résultant d’un interaction choc-chouche limite turbulent en transsonique. La Recherche Aérospatiale 6:305–320

[8] Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 32:1598–1605. https://doi.org/10.2514/3.12149

[9] Huang PG, Bradshaw P, Coakley TJ (1992) Assessment of closure coefficients for compressible-flow turbulence models. Ames Research Center, Moffett Field,California 94035-1000

[10] Chutkey K, Viji M, Verma SB (2017) Effect of clustering on linear plug nozzle flow field for overexpanded internal jet. Shock Waves 27:623–633. https://doi.org/10.1007/s00193-017-0707-y

[11] Soman S, Suryan A, Nair PP, Dong Kim H (2021) Numerical Analysis of Flowfield in Linear Plug Nozzle with Base Bleed. J Spacecr Rockets 58:1786–1798. https://doi.org/10.2514/1.A34992