



On the use of bi-species ZDES for multi-nozzles Space Launcher Configurations

Jolan Reynaud, Pierre-Elie Weiss, Sebastien Deck

► To cite this version:

Jolan Reynaud, Pierre-Elie Weiss, Sebastien Deck. On the use of bi-species ZDES for multi-nozzles Space Launcher Configurations. ETMM13, Sep 2021, Rhodes, Greece. hal-03368494

HAL Id: hal-03368494

<https://hal.science/hal-03368494>

Submitted on 6 Oct 2021

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

ON THE USE OF BI-SPECIES ZDES FOR MULTI-NOZZLES SPACE LAUNCHER CONFIGURATIONS

J. Reynaud¹, P.-E. Weiss¹ and S. Deck¹

¹ *ONERA The French Aerospace Lab, 8 Rue des Vertugadins, F-92190 Meudon, France*

jolan.reynaud@onera.fr pierre-elie.weiss@onera.fr sebastien.deck@onera.fr

Abstract

In order to study the unsteady flows occurring in the wake of space launchers, numerical models have to provide an accurate description of both the turbulent behaviour of base flows and the thermodynamic properties of the multi-species and reactive propulsive jets. As the complexity of models dedicated to each of these objectives can have a significant impact on computational cost, especially for reusable designs including several nozzles, authors have often to establish a compromise between an accurate treatment of turbulence and the taking into account of chemistry effects. To contribute to the improvement of launcher base flow predictions, this paper presents the study of a four-nozzle space launcher configuration with a dedicated numerical workflow focusing on the treatment of turbulence. Indeed, such a case has only been studied with RANS approaches so far, and is treated here with the automatic mode of the Zonal Detached Eddy Simulation (ZDES Mode 2 2020) and hybrid numerical methods, based on Ducros' sensor and allowing to resolve well the turbulent fluctuations of the base flow while capturing the strong normal shock structures inside the jets. The computations are furthermore performed thanks to inert bi-species numerical methods enabling to take into account some thermodynamic properties of the jets with an acceptable computational cost. The bi-species scale resolving numerical workflow is able to reproduce the main features of multi-jets afterbody flows and provides a satisfying agreement with experimental pressure measurements. Compared with RANS calculations, the use of ZDES not only gives access to additional information regarding the unsteadiness of the flow but also provides distinct predictions of the mean aerodynamic field.

1 Introduction

To support the development of multi-nozzles reusable launcher designs, the accurate prediction of afterbody flows occurring behind the base and around the nozzles during flight is of primary importance to assess the aerodynamic performance and the integrity of the vehicles. Indeed, the reactive multi-gas interactions occurring between the hot propulsive jets and the base flow can induce adverse phenomena such as base

drag, due to the low pressure recirculation zone forming behind the base, pressure loads which may damage mechanical structures and base heating linked to the recirculation of hot gases towards the base. As a result of this complexity, the aerodynamic design of new generation launcher afterbody remains a challenge for both experimental and numerical tools.

Focusing on numerical approaches, two main conclusions can be drawn from the literature. First, the need to resolve the large scale turbulent eddies driving the dynamics of base flows in order to perform accurate predictions of the wake flow topology and wall pressure levels has been well documented on generic space launcher configurations (Reynaud et al. 2021a, Statnikov et al. 2016). RANS approaches have indeed been shown to have limited capacities to reproduce such massively separated flows whereas RANS/LES approaches, combined with low dissipative numerical schemes, have displayed a good cost/accuracy compromise (Pain et al. 2014). Then, authors such as (Gusman et al. 2011) have shown the need to take into account the thermodynamic properties of the hot reactive propulsive jets to increase the representativeness of space launcher simulations. As the computational cost of both RANS/LES approaches and multi-species reactive flow models is important, authors have often to establish a compromise between an accurate treatment of turbulence and the taking into account of chemistry effects.

Regarding wind tunnel experiments, only few studies have been dedicated to multi-nozzles space launcher configuration with hot propulsive jets (Mehta et al. 2013). Among them, the case of a four-JP4/LOX-nozzle space launcher configuration studied experimentally in (Musial and Ward 1961) is considered as a reference case and has been treated numerically by several authors. These studies (Mehta et al. 2013, Pu and Jiang 2019), which employed advanced thermodynamic models but only used RANS models for turbulence modelling, have provided useful analyses of the flow field but have reported noticeable errors in the evaluation of wall pressure levels.

To contribute to the study of launcher base flow prediction methods, this paper presents a numerical study of the four-nozzle space launcher configuration with a scale resolving framework based on the use of a

simple two-species physical model and hybrid numerical methods to allow for an affordable ZDES mode 2 (2020) (Deck and Renard 2020) computation. The results will be compared with a two-species RANS computation and with RANS computations including chemistry effects from the literature in order to investigate the benefits of a scale resolving approach for multi-nozzle afterbody computations.

In this article, we first describe the test case and the features of the numerical framework used for the bi-species ZDES and RANS computations. We then characterize the launcher wake flow thanks to instantaneous and mean flow visualizations. In particular, the focus is put on the resulting wall pressure distribution on the base and on the nozzles of the launcher.

2 Test case and mesh setup

The present study is based on a wind tunnel experiment provided by Musial and Ward, the geometry of the model for this study is shown in Fig.1. It consists of a four-nozzle subscale rocket model with parameters such as $r_b = 152.4$ mm, $L_n/D_e = 1.53$, and $D_s/D_e = 1.67$. The four rocket engines operate with a propulsive mixture combining JP-4 as a fuel and LOX as an oxidizer. The selected test flow conditions, corresponding to a flight altitude of 14.9 km, are summarized in Table.1 together with the total pressure P_c and total temperature T_c inside the combustion chamber. These test conditions correspond to underexpanded jets with an exit Jet Pressure Ratio (JPR=3.4) (Mehta et al. 2013). The wind tunnel model was instrumented with pressure transducers and temperature probes on its base.

As shown in Fig.2b a structured mesh has been designed by including one secondary O-H topology for each nozzle in a main O-H topology built around the launcher body. The radial point distribution shown in Fig.2a is designed in order to ensure $\Delta y^+ \approx 1$ for every attached boundary layers and to cluster points in the mixing layer forming behind the main body base. With 240 points in the azimuthal direction ($\Delta\theta = 1.5^\circ$) the mesh contains 55.6×10^6 cells overall.

Parameters	Values
P_c	41.37 bar
T_c	3469.8 K
P_∞	12209.5 Pa
T_∞	297.7 K
M_∞	2.75

Table 1: Low altitude case initial flow conditions (Musial & Ward 1961)

3 Computational set-up

Two-species model

To study the 4 nozzle launcher configuration, an implicit finite volume formulation of the two-species Navier-Stokes equations, presented in (Reynaud et al. 2021b) is used. This approach enables to define the propulsive gas mixture and the surrounding air as two perfect gases. To provide adequate properties for the propulsive mixture, JP-4/LOX equilibrium flow data from (Huff et al. 1956) are used to define constant values for the Sutherland's law constants $[\mu_{0_j}, S_j, T_{0_j}]$, the specific heat ratio γ_j , the volume constant heat ratio c_{v_j} and the Prandtl number Pr_j of the jets gas (see Tab.2). The turbulent Schmidt number is chosen as 0.5 for the mixing between air and the propulsive mixture.

μ_{0_j} (N.s/m ²)	$1.85 \cdot 10^{-5}$
T_{0_j} (K)	370
S_j (K)	168
γ_j	1.224
c_{v_j} (J/kg)	1997.5
Pr_j	0.78

Table 2: Properties of the perfect gas simulating a JP4/LOX mixture

Turbulence modelling and numerical scheme

The RANS computation is performed with the Spalart-Allmaras turbulence model (Spalart and Allmaras 1992) and an AUSM+ scheme (Liou 1996) for convective fluxes. As this scheme is too dissipative for scale resolving computations, the numerical fluxes for the ZDES computation are evaluated thanks to a hybrid numerical scheme (Reynaud et al. 2021a) built to switch from the AUSM+ scheme in presence of shocks to a low dissipative AUSM+(P) formulation (Mary and Sagaut 2002) in vortical regions. The time step for the ZDES computations is $\Delta t_{CFD} = 2 \cdot 10^{-7}$ s which enables to ensure that $CFL < 6$ in the base flow mixing layer. As the vortex shedding period linked to the base diameter was estimated to be $T_s = \frac{2 \cdot r_b}{0.2 \cdot U_\infty} \approx 1.6$ ms, a period of 32 ms = $20 \times T_s$ was adopted to clear the flow from transient effect before using a 80 ms $\approx 50 \times T_s$ period to compute statistics.

4 Results

Flow topology

The instantaneous visualisation of the wake flow shown in Fig.3a enables to observe that the main body boundary layer separates at the base and rolls up into azimuthal vortices to form a mixing layer which bends towards the launcher axis under the effect of an expansion wave. As displayed in the instantaneous pressure field shown in Fig.3b, a realignment shock forms as the bended flow meets with the propulsive jets boundaries and induces an adverse pressure gradient on the outer side of the jet boundaries. Looking at the underexpanded propulsive jets, we observe that in coherence with results from (Mehta et al. 2013, Pu and

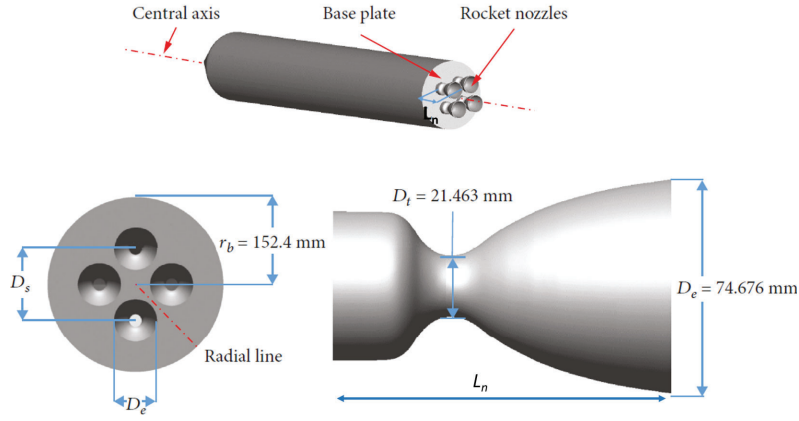
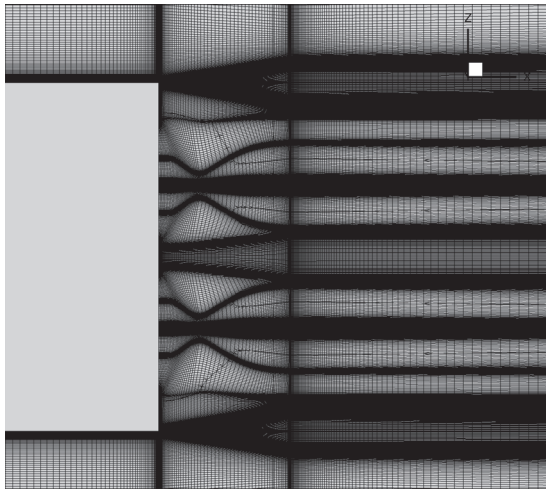
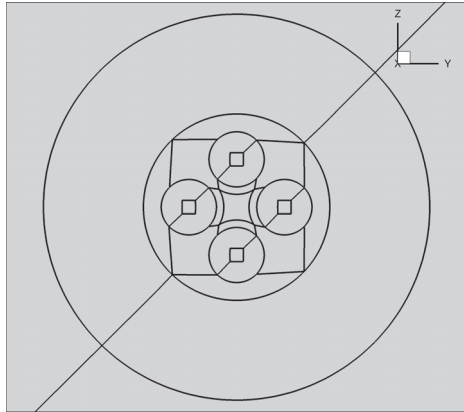


Figure 1: Schematic view of the 4 nozzle launcher geometry (from (Pu and Jiang 2019))



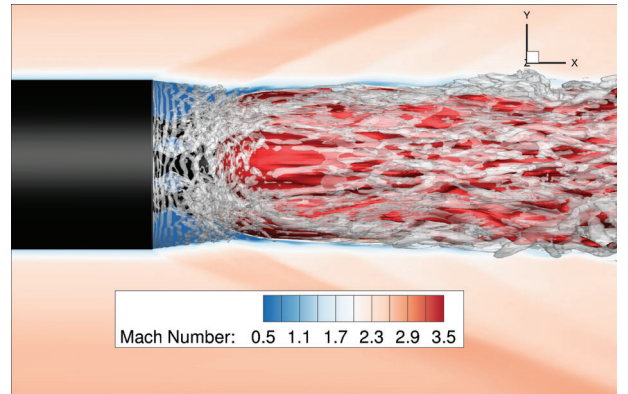
(a) Mesh distribution in the nozzles exit area



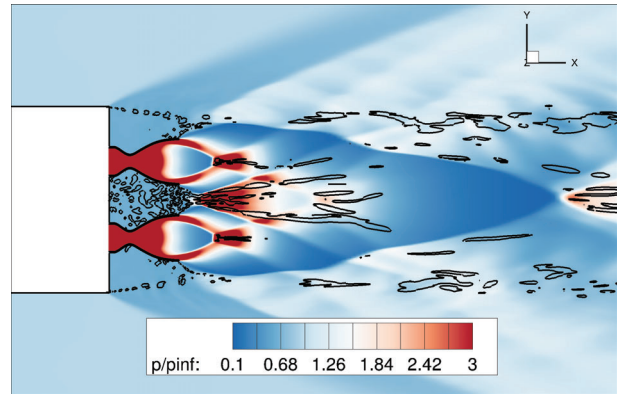
(b) 4 secondary O-H topologies included in a main O-H topology

Figure 2: Mesh description

Jiang 2019), the jet boundaries collide near the axis of the launcher and create a high pressure zone on the inner side of the jet boundaries. These high pressure gradient zones together with the normal shocks visible inside the jets justify the use of hybrid numerical methods able to capture shocks while maintaining



(a) Mach number distribution and coherent structures in the wake of the launcher

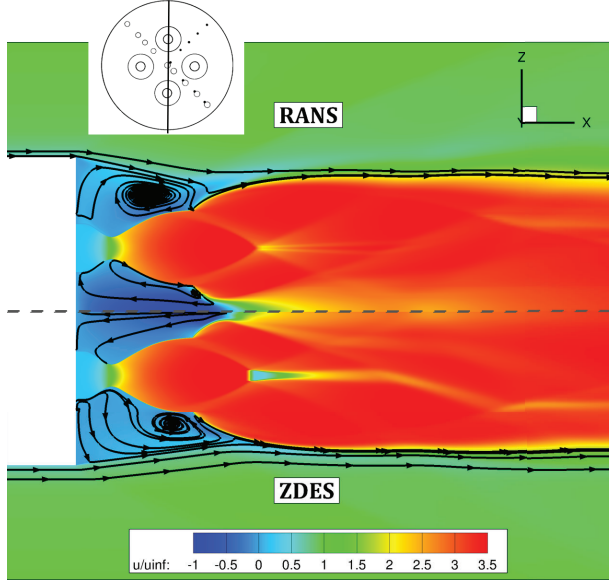


(b) Instantaneous pressure distribution and coherent structures (black iso-lines) in the wake of the launcher

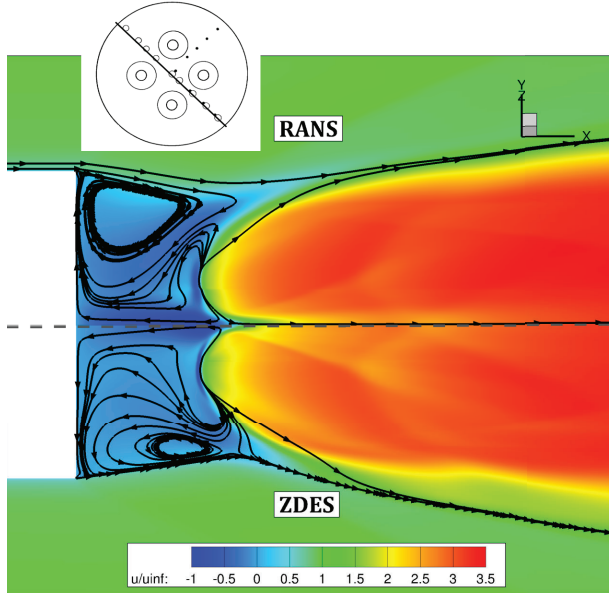
Figure 3: Instantaneous flow topology

a low numerical dissipation in the rest of the flow to resolve turbulent structures. One can indeed note in Fig.3b that in addition to the azimuthal vortices issuing from the base, the mixing process between the jets engender the formation of a wide variety of turbulent structures in the inter-nozzle area.

The time averaged topology of the base flow results from the combination of the two adverse pressure gradients previously mentioned. We compare in the



(a) Distribution of longitudinal velocity and streamtraces in a plane containing the nozzles.



(b) Distribution of longitudinal velocity and streamtraces in a plane between the nozzles

Figure 4: Average velocity distribution in the wake of the launcher

following the average flow fields obtained with RANS and ZDES computations. First, looking at a longitudinal plane containing two nozzles (Fig.4a), one can observe the formation of a recirculation zone on the outer side of the nozzles. This recirculation zone interacts with the propulsive jets which entrain the external flow through a mixing layer. This entrainment effect appears stronger in the RANS computation as the recirculation bubble reattaches closer to the nozzle exit than in the ZDES computation. This first flow pattern corresponds to the one usually observed for axisymmetric base flow with a single nozzle (Reynaud et al.

2021a, Statnikov et al. 2016). In the present case of 4 propulsive nozzles, additional flow interactions occur. Indeed, due to the impingement between the supersonic propulsive jets, the fluid located between the nozzles is subject to a strong adverse pressure gradient that leads to a reverse flow heading towards the base. We note that the RANS computation provides a higher reverse velocity than the ZDES one. This reverse flow impacts the base and induces the formation of radial wall jets. In a plane containing nozzles, these wall jets will be deflected as they go around a nozzle and part of the reverse flow will regain a positive longitudinal velocity. In a plane located between the nozzles (see Fig.4b), these wall jets will flow towards the base outer radius and feed the external recirculation zone. In such a plane, the prediction of the recirculation zone topology again differs between the two computations as the S-A model predicts a center position of the recirculation closer from the base than ZDES. Notwithstanding some local differences, the two bi-species computations are able to reproduce the main flow features of a 4-nozzle launcher wake flow (Mehta et al. 2013).

To investigate further the interaction between the propulsive jets and the base flow, the average distribution of the propulsive mixture mass fraction (Y_{fuel}) in the afterbody area is displayed on Fig.5. One can note that the RANS computation predicts a much intense mixing between the two gases than the ZDES one. This is shown by the greater predicted growth rate of the jet external mixing layer and by Y_{fuel} levels in the inter-nozzle area nearly twice as high as those found with ZDES. Such discrepancies in the amount of hot propulsive gases flowing back to the base area can be attributed to the strong influence of the turbulent Schmidt number on RANS mixing prediction (Reynaud et al 2021b) and is expected to induce differences in base temperature predictions. One can indeed observe in Fig.6 that the RANS computation predicts higher base temperature levels and a wider high temperature zone. Due to the absence of chemistry effects and to the use of adiabatic wall boundary conditions, both computations overestimate the temperature ratio $\frac{T}{T_c} = 0.55$ measured experimentally at the center of the base.

Base pressure distribution

To assess the present computations, data from the base pressure experimental probes are used together with RANS results from two studies found in the literature. First, (Mehta et al. 2013) performed computations with different levels of physical modelling for the propulsive jets; a frozen flow computation (Me1) where the propulsive mixture is considered as a perfect gas with a fixed $\gamma_j=1.15$ (value corresponding to chamber conditions); a variable γ_j computation (Me2) where γ_j is a function of temperature and a reactive computation (Me3) where the propulsive gas is modelled as a 10 species mixture and chemical reactions are taken into account. Then, in (Pu and Jiang 2019)

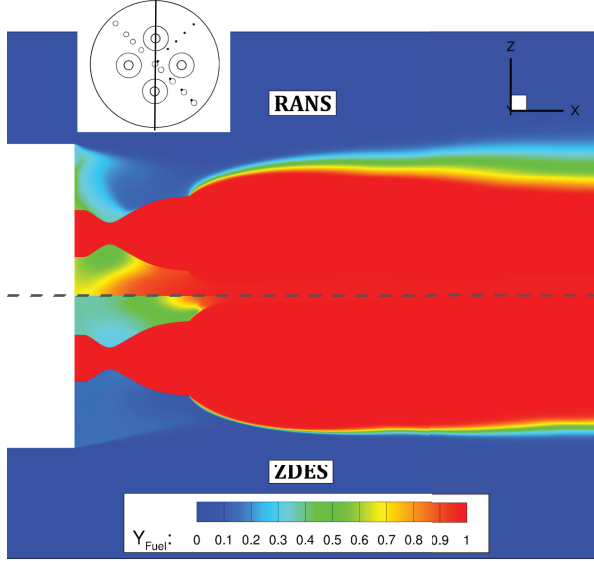


Figure 5: Distribution of fuel mass fraction in a plane containing the nozzles.

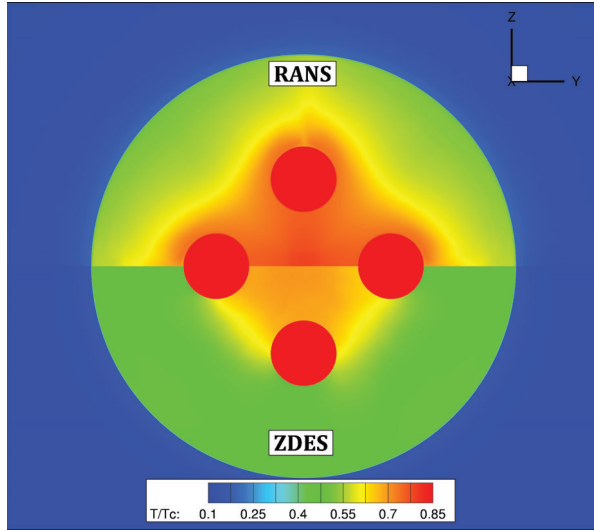


Figure 6: Temperature distribution on the base.

several RANS models were tested to perform reactive computations.

Looking at base pressure coefficient distributions displayed in Fig.7, results from (Metha et al. 2013) showed that, with the Menter BSL RANS model, the use of the simplest physical model (Me1) surprisingly provided the best predictions for base pressure levels as both the variable γ_j computation (Me2) and the reactive computation (Me3) lead to a significant underestimation of base pressure levels in comparison with experimental data. The same magnitude of error is found in (Pu and Jiang 2019) as shown here with the predictions obtained with the RNG turbulence model (Pu RNG), which provided the best results in their study. One can furthermore observe that all of the mentioned RANS computations are unable to reproduce the almost flat pressure profile found experimen-

tally as they induce an overestimation of the pressure difference occurring between the center of the base and the outer radius. These discrepancies motivate the use of more advanced turbulence modelling such as the ZDES approach used here. Focusing on base pressure coefficient distributions obtained with the present computations, one can note that the use of an equilibrium flow hypothesis in the nozzles (with $\gamma_j=1.224$) provides predictions coherent with numerical results from the literature, in between the frozen flow computation and the computations including chemistry effects. The use of ZDES improves predictions in comparison with RANS as it enables to recover a flat pressure profile and higher pressure levels near the outer radius. These disparities can be linked to the previously observed differences in the topology of the recirculation zone.

Nozzle pressure distribution

The evolution of the pressure coefficient (C_p) along generatrix lines located at different angular locations (outer position (OUT), inner position (IN) and side positions (SIDE)) on a nozzle external wall is displayed in Fig.8 and reveals further deviations between the RANS and the ZDES computations. Indeed, the use of ZDES provides relatively flat C_p profiles (close to $C_p = -0.065$) on every position around the nozzle whereas the RANS calculation predicts noticeable pressure axial variations for each location around the nozzle and a substantial pressure gap between the inner position and the outer position. Each turbulence modelling approach would thus lead to different estimations of the pressure loads used to study the mechanical behaviour of the nozzles. Based on previous numerical results obtained on launcher configurations with a single nozzle (Pain et al. 2014, Reynaud et al.2021b) one can note that the ZDES predictions are used to be more representative than the RANS ones.

To complete the study of side-loads with consideration for the unsteadiness of the flow, the ZDES computation enables to investigate the evolution of the fluctuating pressure coefficient along the nozzles in Fig.9. The overall increase of fluctuating pressure levels in the direction of the flow observed with the present computation is in coherence with previous RANS/LES results and experimental measurements described in (Statnikov et al. 2016) on a launcher with one nozzle placed in an external flow at $M_\infty = 3$. The computed fluctuations levels are furthermore in the same range as the one displayed in (Statnikov et al. 2016) with $C_{p_{rms}} \in [0.0015, 0.01]$. Important differences are however observed depending on the angular position. Indeed, the inner side of the nozzle is subject to pressure fluctuations twice as high as the outer side. These higher fluctuations levels can be related to the large amount of turbulent structures observed earlier in the inter-nozzle area.

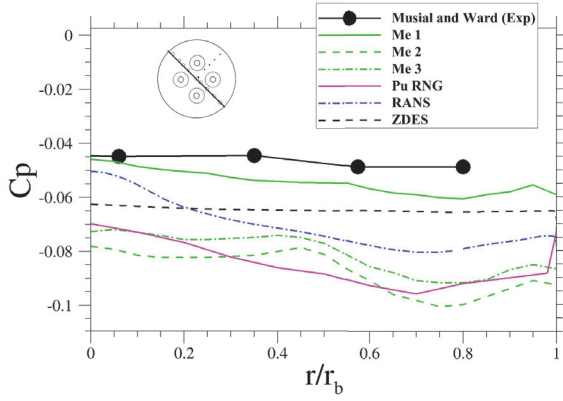


Figure 7: Base pressure distribution along an inter-nozzle radial line

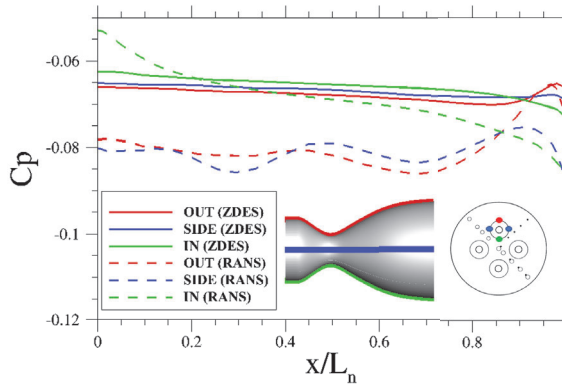


Figure 8: Axial evolution of C_p at different angular locations along a nozzle

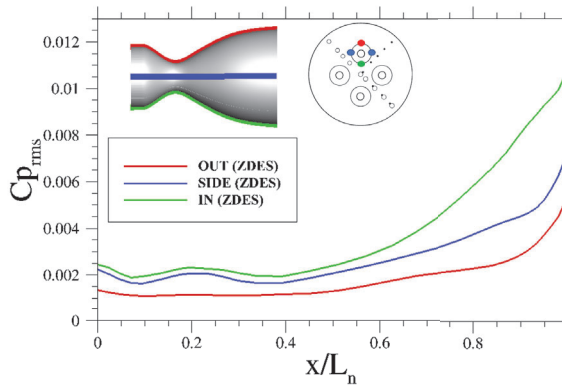


Figure 9: Axial evolution of C_{prms} at different angular locations along a nozzle

5 Conclusions

To investigate the benefits of a scale resolving approach for multi-nozzle afterbody computations, a bi-species numerical workflow based on the combination of the ZDES Mode 2 (2020) and hybrid numerical methods has been applied to a 4-nozzle launcher configuration. The workflow appears as a promising tool able to reproduce the main flow features occur-

ring in the wake of a 4 nozzle space launcher and to provide quantitative estimations of the resulting mean and fluctuating loads. In comparison with RANS computations, ZDES provides significant differences in the prediction of base flow topology, gas composition near the base and wall pressure distributions. These encouraging results should motivate further numerical studies combining the resolution of the large eddies forming in the wake of the launcher with more advanced thermodynamic models but also advocates for the realisation of more detailed wind tunnel experiments including hot propulsive jets and unsteady measurements.

Acknowledgments

The authors wish to thank the Centre National d'Études Spatiales (CNES) for financial support. The Ph.D. work of J. Reynaud is funded by CNES and ON-ERA.

References

- Deck, S., Renard, N. (2020) Towards an enhanced protection of attached boundary layers in hybrid RANS/LES methods, *Journal of Computational Physics*, Vol 400, p.108790
- Gusman, M.; Housman, J., Kiris, C. (2011), Best Practices for CFD simulations of launch vehicle ascent with plumes-overflow perspective *49th AIAA Aerospace Sciences Meeting*
- Huff, V. N.; Fortini, A., Gordon, S. (1956), Theoretical performance of JP-4 fuel and liquid oxygen as a rocket propellant II: equilibrium composition *NACA Technical report*
- Liou, M.-S. (1996), A Sequel to AUSM: AUSM+, *Journal of Computational Physics*, Vol. 129, pp. 364-382
- Mary, I., Sagaut, P. (2002), Large Eddy Simulation of Flow Around an Airfoil Near Stall, *AIAA Journal*, Vol. 40, pp 1139-1145.
- Musial N.T., Ward J.J (1961), Base flow characteristics for several four-clustered rocket configurations at mach numbers from 2.0 to 3.5, *Technical Report NASA*.
- Mehta, M., Canabal, F., Tashakkor, S. B., Smith, S. D. (2013), Base heating sensitivity study for a 4-cluster rocket motor configuration in supersonic freestream. *Journal of spacecraft and rockets* Vol. 50, No. 3, May-June 2013
- Pain R., Weiss P-E., Deck S., (2014), Zonal Detached Eddy Simulation of the Flow Around a Simplified Launcher Afterbody, *AIAA Journal*, Vol. 52, pp. 1967-1979.
- Pu P., Jiang Y. (2019), Assessing Turbulence Models on the Simulation of Launch Vehicle Base Heating, *International Journal of Aerospace Engineering*, Vol. 2019, pp. 1-14.
- Reynaud, J., Weiss, P-E., Deck, S., (2021a), Numerical workflow for scale-resolving computations of space launcher afterbody flows with and without jets *Computers and Fluids*, p. 104994.
- Reynaud, J., Weiss, P-E., Deck, S., (2021b), Towards Binary Gas Mixture ZDES for Space Launcher Base Flow Prediction *55th 3AF International Conference on Applied Aerodynamics*
- Spalart, P., Allmaras, S., (1992) A one-equation turbulence model for aerodynamic flows. *30th Aerospace Sciences Meeting and Exhibit, AIAA*.
- Statnikov, V. et al. (2016) Experimental and numerical investigations of the turbulent wake flow of a generic space launcher at $M_\infty = 3$ and $M_\infty = 6$ *CEAS Space Journal*, Springer, 8, 101-116