



Spyros Tsentis¹

Centre for Propulsion and Thermal Power
Engineering,
Cranfield University,
Bedfordshire MK43 0AL, UK
e-mail: spyros.tsentis@cranfield.ac.uk

Ioannis Goulos

Centre for Propulsion and Thermal Power
Engineering,
Cranfield University,
Bedfordshire MK43 0AL, UK
e-mail: i.goulos@cranfield.ac.uk

Simon Prince

Centre for Aeronautics,
Cranfield University,
Bedfordshire MK43 0AL, UK
e-mail: simon.prince@cranfield.ac.uk

Vassilios Pachidis

Centre for Propulsion and Thermal Power
Engineering,
Cranfield University,
Bedfordshire MK43 0AL, UK
e-mail: v.pachidis@cranfield.ac.uk

Vladeta Zmijanovic

Reaction Engines Ltd.,
Culham Science Centre,
Abingdon OX14 3DB, UK
e-mail: vladeta.zmijanovic@alumni.polytechnique.org

Cavity Impact on the Base Flow Unsteadiness for a High-Speed Exhaust System

Future propulsion systems could enable high-speed flight and routine space access. Such concepts employ base-embedded nozzles and cavities to allow space for nozzle gimballing, thus altering the flow dynamics at the base in comparison to contemporary launch vehicles. This article presents a numerical investigation on the impact of a cavity on the base flow unsteadiness for a sub-scale, high-speed exhaust at over-expansion. The delayed detached eddy simulation approach is used to investigate the base flow. The configuration featuring the cavity is compared to a baseline apparatus, where the cavity is removed, thus allowing for its impact to be identified. The computational approach is validated against experimental data. Results show that the cavity region can reduce the base pressure fluctuations by up to 20% and acts in a damping-like manner for the base flow unsteadiness. The total energy of the base pressure fluctuations spectrum can be reduced by up to 38% compared to the baseline configuration. Finally, the cavity has a notable effect on the nozzle side loads, which are reduced by an order of magnitude compared to the baseline case. This indicates that the cavity could reduce undesirable off-axis loads. [DOI: 10.1115/1.4069222]

Keywords: base flow dynamics, base cavity, base drag, hybrid RANS/LES, DDES, nozzle side loads

1 Introduction

1.1 Background. High-speed propulsion is usually linked to space access as well as supersonic, and hypersonic flight. When it comes to space access, the options are typically limited to launch vehicles and rocket-based transportation systems. On the contrary, sustainable, high-speed flight has been traditionally related to air-breathing propulsion systems [1]. In this context, the most commonly found approaches include systems which feature no rotating components, namely ramjets [2] and supersonic combustion ramjets (scramjets) [3]. These systems however, do not feature the capability of generating static thrust. Combined-cycle concepts have been developed to address this issue, including turbine-based [4] and rocket-based [5] combined-cycle concepts, among others.

Additionally, the development of re-usable systems has been the focus of intense research [6]. A technology that appears promising

is based on the concept of pre-cooled combined-cycle engines [7]. This new class of engines, cools the air prior to compression, therefore extending the operational range of the downstream turbomachinery. Such systems have been investigated extensively from a thermodynamic point of view, and have been found to exhibit notably improved efficiency compared to contemporary rocket-based systems [8,9].

While the advantage of these systems is well-established from a thermodynamic perspective, their overall aero-propulsive behavior is not yet fully understood. These concepts usually feature high-speed nozzles at the base, similar to rockets. However, the nozzle in most rocket-based systems extends downstream of the base, while in the aforementioned concepts, the nozzle is embedded within the base. This poses a major difference from a flow physics perspective, and therefore, specific attention must be given to these concepts. Additionally, these systems require base cavities to facilitate their operation and allow space for nozzle gimballing. Such cavities have been recently found to have a severe impact on the mean base flow and nozzle operation, and can notably increase the base drag at specific flow conditions by as much as 12% [10]. However, their potential impact on the unsteady base flow characteristics and nozzle side loads, has not been reported in previous studies.

Paper presented at the Turbo Expo, June 24–28, 2024.

¹Corresponding author.

Contributed by International Gas Turbine Institute (IGTI) of ASME for publication in the JOURNAL OF TURBOMACHINERY. Manuscript received August 10, 2024; final manuscript received July 18, 2025; published online September 12, 2025. Tech. Editor: David G. Bogard.

1.2 Unsteady Base Flow Features. Base flow dynamics are of major importance for the design and control of launch vehicles due to the massive separation that occurs at the base, which is characterized by highly unsteady features [11]. This region is dominated by very low pressure, therefore, substantially increasing the base drag [12]. The flow characteristics at the base of several, generic propulsive configurations have been extensively studied over the years. Most of the studies available in the public domain deal with both planar [13] and axisymmetric configurations. In the context of axisymmetric topologies, many studies focus on blunt-based geometries [14,15] and backward facing steps (BFS) for both power-off [14,15] and power-on [16] conditions.

Depress et al. [17] investigated experimentally the unsteady afterbody flow characteristics at transonic conditions for blunt-based and axisymmetric BFS configurations, with and without a propulsive jet. They reported the length extension of the nozzle to be a key parameter for the base flow unsteadiness, primarily through the reattachment (or not) of the separated shear layer on the downstream surface. The presence of the jet had little effect on the large-scale dynamics which were found to be dominant at non-dimensional frequencies of $St_D \sim 0.2$. Deck and Thorigny [11] and Weiss et al. [18] examined the dynamics of such separating/reattaching flows for an axisymmetric BFS geometry, with and without a jet respectively, using the zonal detached eddy simulation approach [19]. They identified dominant large-scale motions at $St_D \sim 0.2$, corroborating the experimental observations. Saile et al. [20] investigated experimentally a similar BFS topology, focusing on the impact of nozzle extension length at over-expanded jet conditions and Mach 0.8, following a previous study which showed an exceptionally excited area with distinct turbulent quantities at such conditions [21]. They concluded that the nozzle extension length is important in the base flow unsteadiness, which is pronounced when the mean shear layer reattaches near the tip of the nozzle.

Studies on more realistic space launcher configurations have also been reported. In specific, the effect of side boosters on the base flow of such cylindrical BFS geometries has been investigated both experimentally [22] and computationally [23] at conditions of $M_\infty = 0.7$. A significant increase in the coefficient of pressure fluctuations of approximately 45% was reported at the base as a result of the presence of the side boosters. Furthermore, results on the base flow and side loads of detailed geometries of the European launchers, Ariane5 and Vega, can be found in the study of Schwane [24]. The studies mentioned previously, demonstrate the significant impact of base geometry on the mean and unsteady base flow characteristics. Most studies in the public domain focus on BFS geometries due to the design of contemporary launch systems. However, in high-speed, combined-cycle concepts the nozzle is typically embedded at the base [25]. Consequently, separate attention must be given to base-embedded nozzle geometries.

Base geometry modifications, including base cavities, locked vortex afterbodies, as well as boat tailing [26], are typically studied in the context of flow control. A comprehensive review on flow control techniques for base drag reduction can be found in the study of Tanner [27]. Base cavities in specific, have been studied for base drag reduction capabilities and base pressure control [26,27]. It is known that their effectiveness is a function of their length, and that they can have an adverse effect on base drag at increased lengths [28]. Tripathi et al. [29], examined experimentally the effectiveness of different base geometries for base pressure control, including a base cavity case. They reported a notable 42% increment of base pressure coefficient at jet-off conditions with respect to the sharp-base scenario. Most of the studies involving base cavities, are in the context of base flow control and base drag reduction [27,29]. However, advanced propulsion concepts with embedded nozzles, could feature relatively deep cavities as an integrated design aspect to allow space for nozzle gimballing. Therefore, the impact of such cavities on the base flow of future, high-speed exhausts must be investigated.

1.3 Nozzle Side Loads. The performance of launch vehicles is strongly connected to the characteristics and operation of the propulsive nozzle. The nozzle operation depends on the nozzle pressure ratio (NPR) and could be in over-expanded, under-expanded, or ideally-expanded state. Over-expansion occurs at low altitudes as well as during engine startup and shut-down, and involves flow separation at the nozzle walls owing to a large adverse pressure gradient, therefore creating undesirable side loads [30,31]. Two states of flow separation have been observed in convergent-divergent (C-D) nozzles, namely free shock separation (FSS) [30] and restricted shock separation (RSS) [30,32]. The latter is known to appear in nozzles that feature an internal shock due to their contour design. At this state, the separated flow reattaches on the nozzle walls, transitioning from FSS to RSS in an unsteady manner, and thus creating notable side loads [33].

A number of studies can be found regarding side loads in rocket nozzles at over-expanded conditions. Hagemann et al. [34] compared the side loads of a thrust-optimized parabolic (TOP) and a truncated, ideal-contoured (TIC) nozzle in a sub-scale, cold-flow test campaign. They reported that the side loads generated in the TIC nozzle were approximately one-third with respect to the TOP nozzle, where transition from FSS to RSS occurred. Deck and Nguyen [35] conducted numerical studies on the side loads of a thrust-optimized contour nozzle during the hysteresis regime between FSS and RSS conditions. They found that the power spectrum of the side loads in the FSS state is dominated by low-frequency energy (below 200 Hz), while for the RSS state, the spectrum was dominated by higher-frequency fluctuations (~ 700 Hz).

Additionally, experimental studies on the side loads of more advanced nozzle contours such as the dual bell nozzle which provides one-step, altitude compensating capabilities have been reported [36], and found to have reduced side loads compared to TIC nozzles. More recently, Cimini et al. [37] carried out numerical simulations for an over-expanded, dual bell nozzle, using the delayed detached eddy simulation (DDES) approach [38]. They reported that the generated side loads are an order of magnitude lower compared to a TIC nozzle. A more detailed analysis on the origins of side loads in over-expanded rocket nozzles can be found in Ref. [31]. While there are numerous studies available on side loads generation during the over-expanded state of several C-D nozzles, the impact of base cavities on the nozzle side loads has not been reported in previous works, especially for base-embedded nozzle geometries, as those typically featured in high-speed exhausts [10].

1.4 Scope of the Present Work. This study presents a numerical investigation on the impact of a cavity region on the unsteady flow characteristics at the base, for a high-speed exhaust system at over-expanded conditions. The examined apparatus is representative of future, advanced propulsion concepts which will incorporate such cavities to allow space for nozzle gimballing. The novelty of this work lies in the quantification of the cavity impact on the base flow unsteadiness and undesirable nozzle side loads. The analysis involves a model installed in the test section of a wind tunnel to facilitate the validation of the computational approach against experimental results. The DDES [38] turbulence modeling approach is used. The instantaneous and time-averaged flow structure at the base is investigated and discussed. Additionally, the effect of the cavity on the mean and unsteady base pressure is identified. To the authors' knowledge, there are currently no published works that quantify the impact of base-embedded cavities on the unsteadiness and nozzle side loads of high-speed exhausts. This is critical for their design.

2 Methodology

2.1 Experimental Details. This section describes the experimental rig used to validate the computational approach. The activities were performed at the closed-circuit, ejector-driven, transonic wind

tunnel (TSWT) of the Defense Academy of the United Kingdom. The facility has a working test section of $H \times h = 0.228 \times 0.206 \text{ m}^2$ and features solid walls without perforation. The flow conditions in the tunnel are controlled through the regulation of the pressure of the ejectors, which are located downstream of the test section. This pressure is referred to as p_{out} in this study. The total pressure of the flow approaching the model in the wind tunnel, denoted as $P_{0,t}$ is nearly atmospheric. Therefore, the tunnel pressure ratio, which is defined as $TPR = (P_{0,t}/p_{out})$ in this work, is regulated by altering the outlet pressure.

The model is mounted on the right side tunnel wall through the pylon which features a NACA 0026 profile cross section. The details of the configuration of the test section with the model mounted can be seen in Fig. 1 [39]. The maximum pylon thickness and length, non-dimensionalized over the model's diameter, are approximately 0.46 and 1.77, respectively. The relatively large pylon thickness is imposed by the required space to accommodate the secondary air flow to the nozzle. The test section dimensions relative to the diameter of the model D , are approximately equal to $H/D = 2.85$ and $h/D = 2.6$, where H and h are depicted in Fig. 1. Additionally, owing to the high blockage ratio of approximately 16.5%, the approaching flow conditions are dependent on the flow conditions in the vicinity of the model. Consequently, to avoid ambiguity in the design variables, in the context of this work, the nozzle pressure ratio is defined as the ratio between the nozzle inlet total pressure over the outlet static pressure, $NPR = (P_{0,n}/p_{out})$. The highest achievable NPR in the experiments was 6.6, and was limited by the maximum mass flowrate of the high-pressure, air-supply system. Consequently, this NPR value is used for validation of the computational approach.

2.2 Investigated Systems. Figure 2 shows the exhaust systems that are investigated numerically in this work. The full, three-dimensional models, mounted in the test section of the TSWT are employed in the computations. Both configurations comprise the nacelle surface of total length L , which is formed by an ogive-shaped nose and the main cylindrical body of diameter D . They both employ an identical, high-speed, TIC nozzle embedded at the base. The configuration depicted in the bottom schematic, also features an axisymmetric cavity region located between the nozzle and the nacelle. The cavity has a depth of $l/D = 0.95$ and a thickness of $0.0875D$. The model geometry and the annular cavity region are designed to facilitate a canonical test case and examine the impact of the cavity on the fundamental aerodynamics. The two systems allow for a direct comparison and hence, identification of the cavity impact. The configurations shown in Fig. 2, are referred to as baseline and cavity cases, throughout the rest of this study.

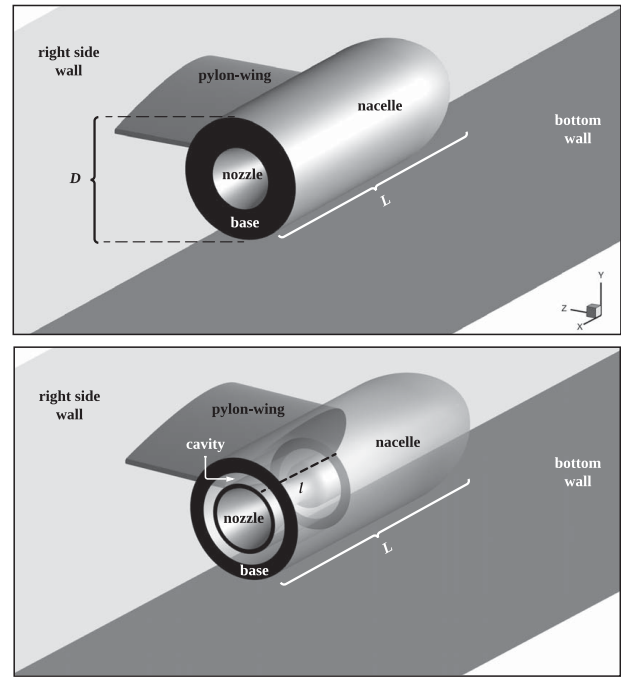


Fig. 2 Examined configurations: baseline case (top) and cavity case (bottom)

The wall contour of the nozzle in the divergent section is designed using the axisymmetric method of characteristics (MoC) procedure [40] to obtain a shock-less flow expansion. The near critical properties of the flow at the sonic line are approximated by employing Sauer's small perturbation technique [41] and used as an initial-value line for the MoC marching procedure in the downstream supersonic flow. The resulting ideal nozzle is truncated at an angle of 7.6 deg, thereby producing a design exit Mach number of approximately 3.4 and a design NPR_d of 59. The computations are performed for conditions of $TPR = 1.28$ and $NPR = 10$, which correspond to more realistic over-expansion behavior in terms of side loads.

2.3 Computational Strategy

2.3.1 Governing Equations. The model used to obtain the field variables is based on solving the Reynolds-averaged version of the three-dimensional Navier-Stokes equations describing the

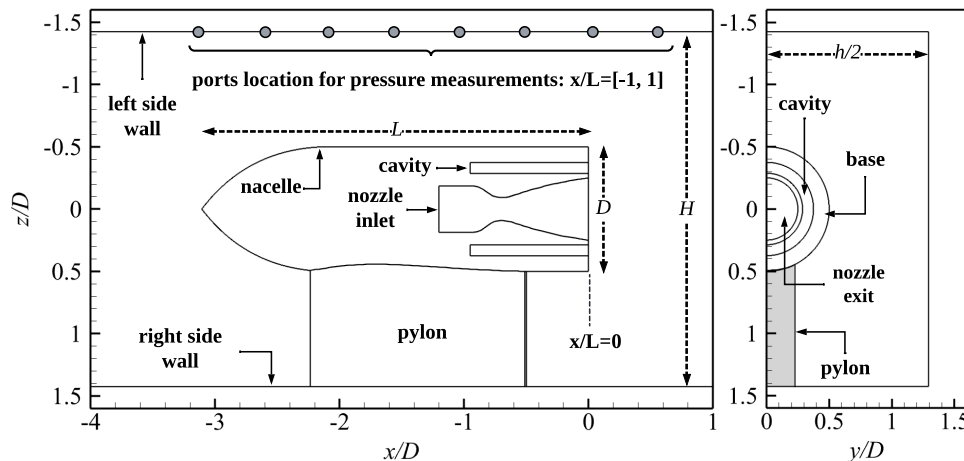


Fig. 1 Schematic of the experimental rig

conservation of mass, momentum, and energy (Eqs. (1)–(3))

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_i)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial(t_{ij} + \rho \tau_{ij})}{\partial x_j} = 0 \quad (2)$$

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho u_i E + p u_j)}{\partial x_j} - \frac{\partial(u_i(t_{ij} + \rho \tau_{ij}))}{\partial x_j} + \frac{\partial q_j}{\partial x_j} = 0 \quad (3)$$

where ρ is the density, u_i is the velocity component in the i th direction, and E is the total energy per unit mass. The total stress tensor, is the sum of the viscous stress tensor, denoted as t_{ij} , and the Reynolds stress tensor $\rho \tau_{ij}$, which is defined based on the Boussinesq approximation [42]

$$t_{ij} = 2\mu S_{ij} \quad (4)$$

$$\rho \tau_{ij} = 2\mu_t S_{ij}^* - \frac{2}{3}\rho k \delta_{ij} \quad (5)$$

$$S_{ij}^* = S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad (6)$$

where S_{ij} denotes the mean strain of rate tensor; μ is the dynamic viscosity and is calculated based on Sutherland's law [43]; μ_t is the eddy viscosity; δ_{ij} is the Kronecker delta, and k is the turbulent kinetic energy. Additionally, the heat-flux vector q_j is obtained from the molecular and turbulent components as shown in Eq. (7)

$$q_j = -c_p \left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial T}{\partial x_j} \quad (7)$$

with Pr and Pr_t denoting the laminar and turbulent Prandtl numbers, respectively. For compressible flows with density variations, the Favre-averaged mean conservation equations can be considered with variables representing mass-averaged values [42], not repeated here for brevity.

2.3.2 Modeling Approach. Due to the wall-bounded flow topology examined as well as the high Reynolds number in the surrounding flow which is $Re_D = 0.85 \cdot 10^6$ at the tunnel inlet and approximately equal to $Re_D = 1.03 \cdot 10^6$ in the vicinity of the base, the turbulence modeling approach adopted herein is the DDES [38]. This approach belongs to the hybrid Reynolds-averaged Navier–Stokes (RANS)/large eddy simulation (LES) methods [44] and its implementation in this work is based on the $k - \omega$ shear stress transport (SST) eddy viscosity model, introduced by Menter [45]. The model solves two transport equations for the turbulent kinetic energy k and specific dissipation rate ω (Eqs. (8) and (9))

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \tilde{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} [(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i}] \quad (8)$$

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} = & a \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} [(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i}] \\ & + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (9)$$

The detailed formulation is not repeated here for the sake of brevity. The interested reader is referred to Ref. [45].

The dissipation term of the turbulent kinetic energy in the formulation of the $k - \omega$ SST model is modified for the DDES model as described in Refs. [46,47]. The transport equation of k in this

formulation reads as

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} [(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i}] + \tilde{P}_k - \rho \sqrt{k^3} / l_{DDES} \quad (10)$$

The DDES length scale is defined as follows:

$$l_{DDES} = l_{RANS} - f_d \max(0, l_{RANS} - l_{LES}) \quad (11)$$

$$l_{LES} = C_{DES} \Delta \quad (12)$$

$$l_{RANS} = \frac{\sqrt{k}}{\beta^* \omega} \quad (13)$$

$$C_{DES} = C_{DES1} F_1 + C_{DES2} (1 - F_1) \quad (14)$$

where $C_{DES1} = 0.78$ and $C_{DES2} = 0.61$ are calibration constants, $\beta^* = 0.09$, and Δ is the maximum local grid spacing ($\Delta = \max(\Delta_x, \Delta_y, \Delta_z)$ in a Cartesian grid). Finally, the blending function that protects the boundary layer from the DES limiter in the DDES formulation is computed as shown in Eq. (15).

$$f_d = 1 - \tanh \left[\left(C_{d1} \frac{\nu + \nu_t}{\kappa^2 y^2 \sqrt{0.5(S^2 + \Omega^2)}} \right)^{C_{d2}} \right] \quad (15)$$

Here S and Ω are the magnitude of the strain rate and vorticity tensors, respectively; κ is the von Kármán constant and C_{d1} , C_{d2} , are constants equal to 20 and 3, respectively [47]. The introduction of this blending function serves as a modification of the original DES formulation (known as DES97) introduced by Spalart et al. [48]. It guarantees that the solver remains in the RANS mode in the attached boundary layers, even under severely refined grids, which would activate the DES limiter in the original formulation. This is necessary to prevent the excessive reduction of the eddy viscosity in the switch area (known as “gray area”) [38,49], leading to the phenomenon of modeled stress depletion (MSD), due to the reduced modeled Reynolds stresses. Severe cases of MSD can lead to grid-induced separation, as emphasized by Menter and Kuntz [50] and can inversely impact a rather good RANS solution in the boundary layer.

2.3.3 Numerical Framework. Computations are carried out by means of a compressible, density-based, and implicit solver [51]. A second-order upwind scheme is selected for the spatial discretization of flow primitive variables as well as turbulent kinetic energy k and specific dissipation rate ω . It is noted that schemes with relatively high numerical dissipation might damp out the smallest scales present at high frequencies. However, less dissipative schemes, often desired in classical LES approaches, such as central-differencing are prone to solution oscillations in the velocity field [52,53]. In this work, the use of the bounded-central-differencing scheme, led to well-known numerical stability issues [52].

A flux-vector splitting scheme—advection upstream splitting method (AUSM+)—is employed for the calculation of the inviscid fluxes. AUSM+ was introduced by Liou [54] and provides accurate resolution of shock and contact discontinuities, and is also free of the instability often involved in multidimensional shock waves, known as the carbuncle phenomenon. A least-squares cell-based method is used for calculation of the flow-field gradients. Air is modeled as an ideal gas. An eighth-order piece-wise polynomial is employed for the calculation of specific heat capacity, while the thermal conductivity is calculated based on the kinetic theory [51]. Time integration is carried out by means of a second-order accurate, backward scheme. Details on the accuracy of the solver in scale-resolving simulations for a number of severely separated flows can be found in Ref. [52].

The computational domain used for the simulations is shown in Fig. 3. The top and left side tunnel walls are hidden for visibility reasons. It consists of the full experimental apparatus, with the TSWT test section extended ten model lengths downstream of the

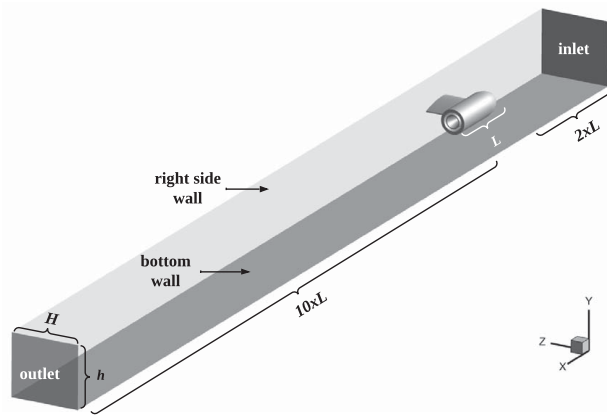


Fig. 3 Schematic of the computational domain

base (corresponding to 125 nozzle radii) and two lengths upstream. Tunnel inlet flow conditions are modeled by applying a pressure inlet boundary condition (BC) at the inlet surface, with total pressure and temperature as the prescribed values. The total pressure $P_{0,t}$ is set to atmospheric conditions. At the outlet of the domain, a pressure outlet BC is applied, imposing the static pressure p_{out} . This pressure value is used to control the TPR in the computations. The nozzle inlet conditions are provided through a pressure inlet BC, prescribing the total pressure and total temperature. The total temperature at the nozzle inlet is set to 290 K. Finally, all surfaces of the model, pylon, and wind tunnel are treated as viscous, no-slip, and adiabatic walls.

The domain is discretized using a hybrid meshing approach [55]. It is noted that grid convergence in scale-resolving simulations is subtle compared to direct numerical simulation (DNS) or RANS, owing to the filtering operation on the Navier–Stokes equations. This issue along with meshing guidelines for DES is discussed in detail by Spalart [56]. In this study, a precursor, converged RANS solution is used to initialize the DDES computation. The meshing strategy is initially informed from previous work [39], where a second-order grid convergence index (GCI) [57] study is performed, based on the generalized Richardson extrapolation procedure [58]. The GCI based on the RANS, corresponding to the medium mesh, was found to be 0.42% for the nozzle standard velocity coefficient ($C_{V,s}$) and 0.68% for the gross propulsive force (GPF). A detailed definition of the performance accounting metrics for this system has been reported in previous work [39]. Additionally, the apparent order p , was close to the formal order of the employed discretization scheme (approximately 1.8 and 2.1 for $C_{V,s}$ and GPF respectively), which can be taken as an indication of the grids being in the asymptotic range [57]. This meshing approach is employed as a baseline, since the part of the domain where the model acts in the RANS mode needs to be covered by a suitable RANS grid, where convergence has been demonstrated. It is specifically important that all relevant boundary layers are sufficiently resolved [52].

The baseline grid resolution is then further improved locally, through a mesh refinement region at the base and near-wake regions. The maximum cell size in that region is approximately equal to $\Delta_{max} = 0.23 \cdot \delta$, where δ is the boundary layer thickness just upstream of the base edge, where separation occurs. This cell size ensures that the main quantity of relevance, which is the ratio of the RANS length scale l_{RANS} (Eq. (13)) to the local grid spacing Δ_{max} , is sufficiently large to allow resolution of the initial flow instability of the separating shear layer from the base [52]. The mesh features values of $(l_{RANS}/\Delta_{max}) > 5$ as recommended in Ref. [52] at the base, cavity, and wake regions. This is depicted in Fig. 4 at the part of the domain which is of interest. The majority of this region has values that exceed 20, including the separated region inside the nozzle's divergence due to FSS state. A thin

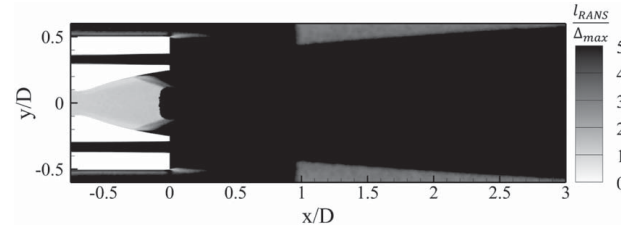


Fig. 4 Ratio of the RANS to the grid length scale at the base and near-wake regions for the cavity case

part of the initial shear layer has slightly smaller resolution. However, the shear layer thickness typically increases largely downstream of the separation edge and thus (l_{RANS}/Δ_{max}) increases quickly to sufficient values [52]. Given the desired cell isotropy in the LES part of the domain [56], reducing the resolution even marginally at that region would severely increase computational cost. It is noted however, that in simulations where small scales are critical (such as in acoustics), delaying the initial instability could result in loss of spectral information.

The final computational mesh is shown in Fig. 5 in a close-up view at the base region. The refined region at the wake extends one model diameter D downstream of the base and covers the upstream cavity region. A total of 35 prism layers are employed for improved boundary layer resolution at all wall surfaces, ensuring a non-dimensional wall distance of y^+ below 1 for all wall-adjacent nodes. The remainder of the domain is discretized in an unstructured manner with polyhedral elements which are typically more efficient in terms of cell-count compared to tetrahedrons. A growth ratio of 1.2 has been maintained throughout the domain. The grid corresponding to the cavity case, comprises a total of 25 million cells, while the one for the baseline, features approximately 23 million cells, owing to the absence of the cavity. A fully converged RANS solution on these grids is used to initialize the unsteady and scale-resolving part of the simulation.

The time-step of the computations is fixed to $\Delta t = 2 \cdot 10^{-5}$ s, which corresponds to a non-dimensional time-step size of $\Delta t^* = \frac{\Delta t U_{\infty,0}}{D} = 0.0535$, where $U_{\infty,0}$ denotes the axial velocity in the vicinity of the base at an azimuthal angle of 0 deg (Fig. 8). The use of this velocity facilitates the generalization of the results, which are usually reported on the basis of the freestream velocity at separation in the case of axisymmetric bodies [11,17,29]. The employed time-step size is selected as such, to ensure that the local Courant–Friedrichs–Lewy number is below 1 in most of the scale-resolving part of the domain. This is facilitated through the evaluation of the local time scales of the precursor RANS simulation [52], as $\Delta/|U|$, where $|U|$ is the magnitude of local velocity vector and Δ is approximated as $(\Delta_x \Delta_y \Delta_z)^{1/3}$.

Additionally, an analysis based on global unsteady dynamics aimed at resolving the shedding frequency of large-scale structures at $St_D = 0.2$, showed that the employed time-step size is sufficient.

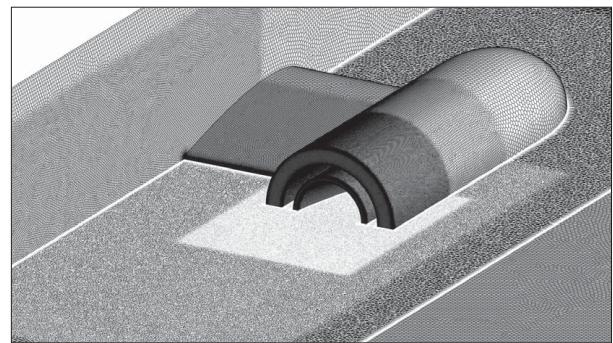


Fig. 5 Close-up view of the surface mesh with volume mesh shown along the symmetry plane

It is noted that smaller time scales are also present in the high-speed jet. However, aiming the simulation to accurate resolution of these scales would increase the computational cost by at least one order of magnitude [59], which was not feasible. After a transient phase, the unsteady calculation initiates, allowing for collection of statistics. The useful unsteady calculation is performed over a period of $TU_{\infty,0}/D = 670$.

3 Results and Discussion

3.1 Validation of Computational Approach. The numerical approach is compared against experimental data for representative conditions of nozzle over-expansion to evaluate the soundness of the computations. Results for both the baseline and the cavity cases are included in the comparison for conditions of $TPR = 1.35$ and $NPR = 6.6$ which was the maximum achievable NPR under the tunnel conditions. Figures 6(a) and 6(b) present the time-averaged pressure distribution along the left side tunnel wall of the TSWT, and the azimuthal pressure distribution at the base, respectively, non-dimensionalized with the ambient pressure p_{∞} of 101,325 Pa.

The experimental static ports are equispaced and span a distance from the nose of the model up to one model length downstream of the base ($x/L = 1$) along the middle plane of the left tunnel side wall (Fig. 1). The pressure values are obtained by a 32-channel Scanivalve unit (model ZOC 22B/32 Px), with a range of 0.31–1.69 bar and accuracy of $\pm 5.5 \cdot 10^{-4}$ bar, recorder at 500 Hz. Three Kulite XCQ-062-1.7 barA dynamic pressure transducers with range of 0–1.7 bar and accuracy of $\pm 1.7 \cdot 10^{-3}$ bar, are used to obtain the pressure at the base of the model. These are located at counterclockwise angle locations of approximately 90 deg, 170 deg, and 270 deg, with 0 deg being opposite to the pylon and 180 deg directly behind it (Fig. 8). Each Kulite signal is low-pass filtered at 10 kHz by a Dewetron signal-conditioner unit before being acquired at a sampling frequency of 25 kHz. The duration of the useful signal during which the jet is active is 3 s.

The error bars correspond to the 95% confidence band based on six independent runs conducted for each case, namely, baseline and cavity. Their larger value close to the maximum cross section of the model and pylon around $x/L = -0.5$, is a result of the high blockage ratio [60], which amplifies the repeatability uncertainty. The latter is below 1% at the tunnel inlet. In that sense, the flow expansion around the model is reflected on the curves shown in Fig. 6(a), and hence, these are also used to ensure that aerodynamic similarity between the computational and experimental flow conditions in the

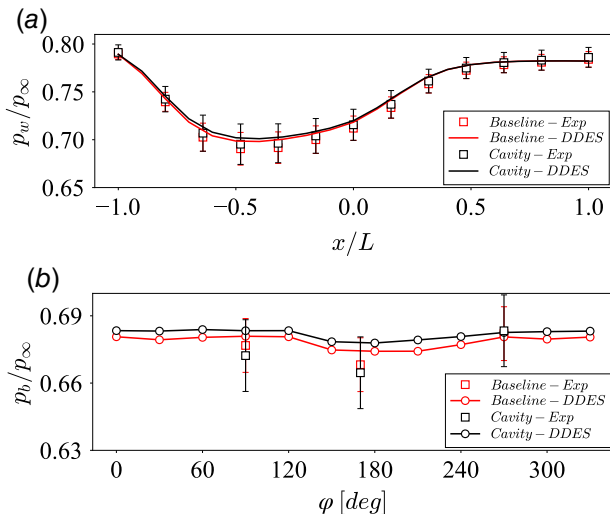


Fig. 6 Time-averaged pressure distribution along (a) the TSWT left side wall and (b) the base

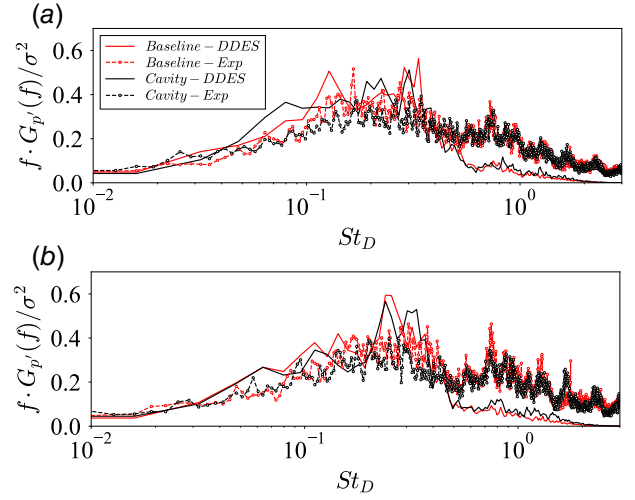


Fig. 7 Pre-multiplied base pressure spectra at (a) $\phi = 90$ deg and (b) $\phi = 270$ deg

vicinity of the base is established. For more details on the wind tunnel installation and blockage effects of this experimental apparatus, the reader is referred to Ref. [39].

As shown in Fig. 6(a), the numerical results are in good agreement with the experimental values, for both the baseline and cavity cases, with the maximum observed error (Eq. (16)) being below 0.9%. Similarly, for the time-averaged base pressure distribution in Fig. 6(b), DDES is in good agreement with the experiment, and properly captures the trend in the azimuthal direction with reduced levels of pressure behind the pylon. The maximum error in base pressure with respect to experiment is observed for the cavity case, at the location behind the pylon, and is approximately 2%. The impact of the cavity identified in the experimental values is small. The DDES results also show small difference between the cavity and baseline cases in terms of base pressure, which is below 0.4%. This is in line with the previous work based on axisymmetric, RANS computations for subsonic flow ($M_{\infty} = 0.7$) and highly over-expanded nozzle conditions [10].

$$\Delta\Phi(\%) = \left(\frac{|\Phi^{Exp} - \Phi^{CFD}|}{\Phi^{Exp}} \right) \cdot 100 \quad (16)$$

Figures 7(a) and 7(b) depict the pre-multiplied power spectral density (PSD) of the base pressure fluctuations, denoted as $G_p(f)$, at azimuthal angles of 90 deg and 270 deg, respectively. The signal of the DDES corresponds to a total of 0.25 s of physical time. The PSD functions of both experimental and DDES time series have been computed using Welch's method [61], with 50% overlap and a Hanning window. A modified periodogram is computed for each segment and then averaging of all periodograms is performed to obtain the PSD estimate. The non-dimensional Strouhal number based on the diameter of the model and defined as $St_D = fD/U_{\infty,0}$, where f denotes the frequency, is used to enable quantitative comparison of different base geometries examined in the previous studies. One can notice that the spectrum at both azimuthal locations is dominated by a broadband peak centered around St_D values of approximately 0.2, which is well captured by the DDES computations. This is in line with previous studies and corresponds to the shedding of the larger-scale structures of the shear layer. Similar content has been identified for BFSs [11,17,18] and blunt-based bodies [14,17]. Less dominant, higher-frequency phenomena are also observed in the experiment at non-dimensional frequencies of $St_D = 0.79 - 0.8$, which are also identified in the DDES computation, although at a weaker intensity probably as a result of numerical dissipation and grid resolution.

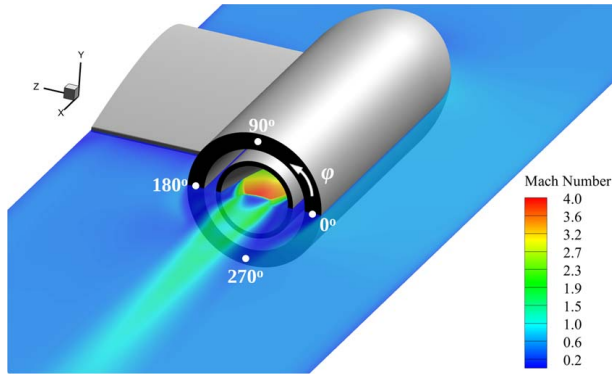


Fig. 8 Reynolds-averaged flow topology

In the context of this work, it is important to examine the capability of the computational approach to assess the impact of the cavity. As shown in Figs. 6 and 7, both the experimental and computational results agree that there is no notable impact of the cavity region at such highly over-expanded flow conditions both on the time-averaged base pressure levels as well as on the unsteady content of the base flow. Overall, the computational results agree with the experimental data within less than 2% for the time-averaged values. Additionally, DDES computations reproduce well the spectrum of the base pressure fluctuations found experimentally, namely a broadband contribution centered at $St_D \approx 0.2$, in line with the literature data.

3.2 Test Case Flow Topology. The conditions for which the base flow unsteadiness is examined in this work correspond to tunnel and nozzle settings of $TPR=1.28$ and $NPR=10$, respectively. The wind tunnel blockage affects the flow conditions at the base and this is amplified with the approaching Mach number. Figure 8 shows the Mach number distribution along the symmetry plane ($x-z$) of the RANS simulation for the model featuring the cavity. The azimuthal angle definition at the base, where instantaneous pressure values are monitored for statistics, is shown. The locations annotated in the figure of 0 deg, 90 deg, 180 deg, and 270 deg, are located along the middle of the base surface, and facilitate the characterization of the three-dimensional base flow unsteadiness, including the impact of the pylon. These points are at identical spatial locations in both examined configurations. As depicted in the figure, the flow conditions in the vicinity of the base and $\phi = 0$ deg correspond to a Mach number of approximately 0.65. The nozzle is over-expanded, exhibiting a Mach disk reflection and being in the FSS state [30]. A notable region of flow separation within the nozzle is also visible in the figure. This region is responsible for undesirable side loads generation within the nozzle [31] and therefore, the cavity impact on the side load generation must be identified.

3.3 Instantaneous and Time-Averaged Flow Structure. The salient characteristics of the instantaneous flow field at the base are shown in Fig. 9 for the baseline case. Coherent structures are identified through the visualization of positive iso-surfaces of the Q-criterion [62], defined as the second invariant of the velocity gradient tensor ∇u

$$Q = \frac{1}{2}(\Omega_{ij}\Omega_{ij} - S_{ij}S_{ij}) \quad (17)$$

where Ω_{ij} and S_{ij} are the anti-symmetric and symmetric components of the velocity gradient tensor, respectively. Observing Fig. 9, the roll-up of vortical structures in the azimuthal direction can be identified, corresponding to the annular shear layer instability. As they move downstream they grow by pairing and are replaced by three-dimensional structures. Similar features have been observed in generic, axisymmetric geometries [11] as well as in more

detailed configurations on the base flow of the Ariane 5 European launcher [63]. Unlike in many studies focused on axisymmetric launch vehicles and BFS geometries [11], where these azimuthal structures are initially toroidal in shape due to uniform freestream velocity, in the present work these structures quickly lose their azimuthal coherence. This is due to the non-square TSWT test section (Fig. 1), which under this blockage ratio creates non-uniform velocity distribution in the azimuthal direction. This results in an asymmetry in the shear layer development. Furthermore, the formation of small- and large-scale hairpin vortices can be seen, similar to what has been identified on the base flow of axisymmetric BFSs [11] and blunt-based bodies [15]. Additionally, it is noticed that the formation and development of turbulent structures takes place promptly after separation without any notable delay in the separating shear layer, which indicates sufficient resolution and proper activation of the scale-resolving branch [52]. This is further confirmed by examining of the blending function f_d (Eq. (15)) [38]. The two cases examined, exhibit similar base flow characteristics in terms of positive Q-criterion iso-surfaces.

The instantaneous aerodynamic field is time-averaged using data at discrete time instances of $10\Delta_t$ for the entire computation period. Figure 10 depicts the time-averaged Mach number distribution and streamlines, along the $x-z$ plane for both the baseline and cavity cases. Included are also the solutions obtained with the steady, Reynolds-averaged computations. The time-averaged flow at the base is organized in a similar manner, for both configurations examined. Both cases exhibit a similar jet vectoring effect due to the presence of the pylon [39]. A pair of low-speed, counter-rotating recirculation regions is established at the separated flow immediately downstream of the base, between the freestream flow and the supersonic jet boundary ($\sim$ Mach 2). This region is dominated by a primary recirculation, associated with the shear layer instability and interacts with the secondary vortex, which appears stretched, as a result of increased jet shear. Additionally, the asymmetrical organization of the shear layer separating from the base is evident due to the pylon, with the lower part of the shear layer being forced upwards. The time-averaged flow enters the cavity region in the upper part, while exiting in the lower part. This is probably a result of the asymmetric base pressure distribution caused by the presence of the pylon, as will be shown in Sec. 3.4, and highlights the three-dimensional nature of the flow inside the cavity. Furthermore, a pair of counter-rotating vortices appears downstream of the Mach disk in the cavity case. This phenomenon has been identified in previous studies for over-expanded TIC nozzles [10,59]. It is related to the curvature of the Mach disk reflection and a detailed description can be found in Ref. [30]. The baseline case periodically exhibited this effect as well, although it was less intense.

The base flow topology predicted using Reynolds-averaged computations is slightly different compared to DDES. One can notice that the length of the separated flow region at the base is reduced

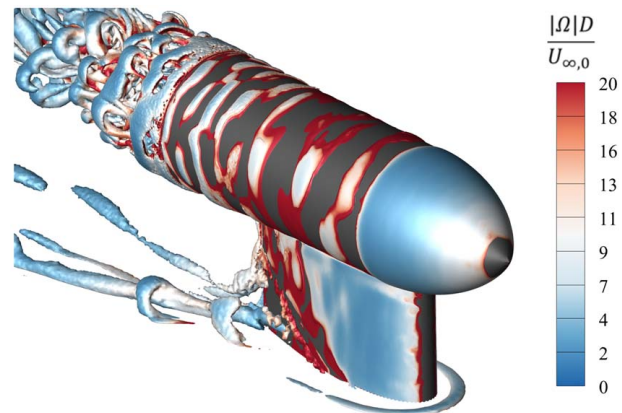


Fig. 9 Instantaneous Q-criterion iso-surfaces colored by non-dimensional vorticity magnitude (Color version online.)

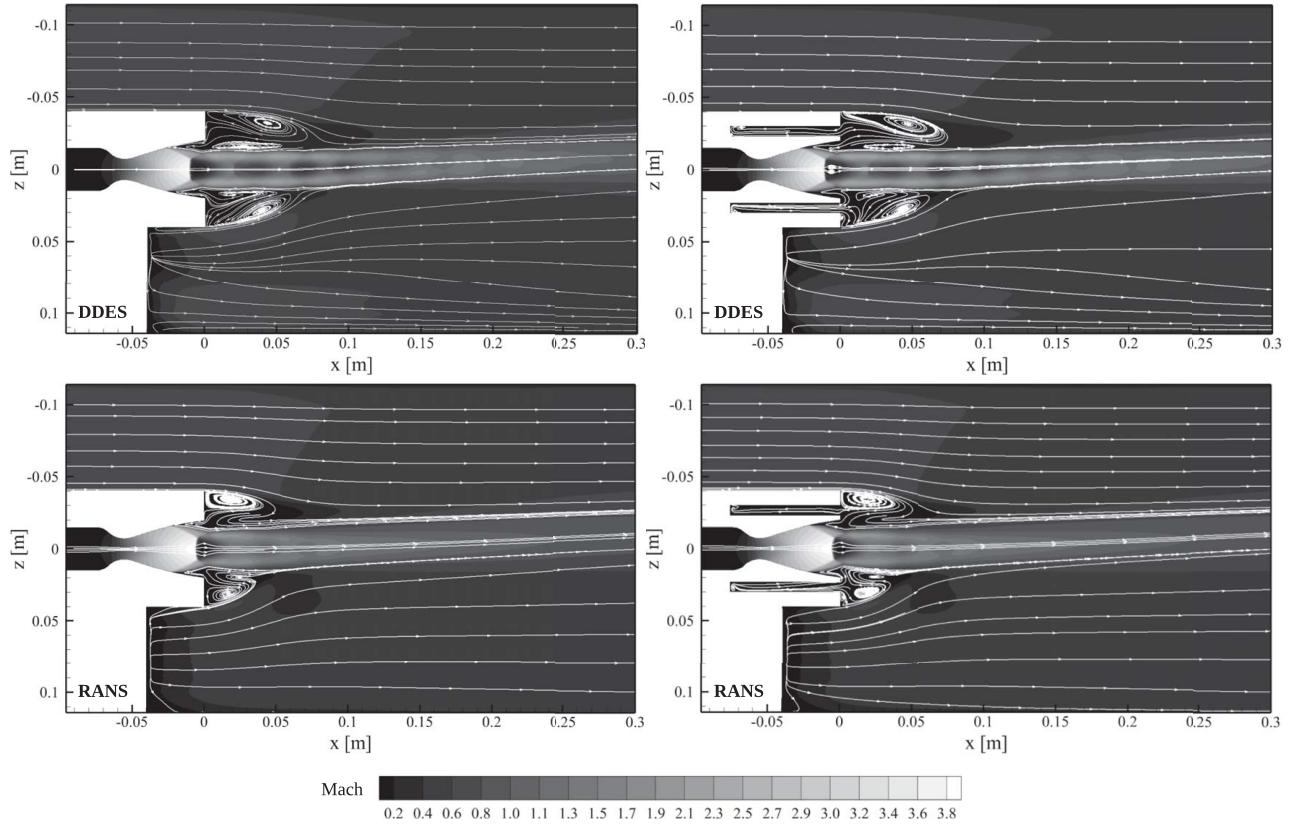


Fig. 10 Time-averaged Mach number distribution from DDES (top) and RANS (bottom)

with the RANS. This has been observed before in base flows studies [64,65]. The core of the wake recirculation region is closer to the base compared to DDES, for both baseline and cavity cases, therefore leading to reduced levels of base pressure. This effect can be seen in Fig. 10 through the location of the Mach disk reflection which is further downstream in the RANS computations compared to the DDES. This indicates increased flow expansion as a result of reduced base pressure.

3.4 Cavity Impact on the Base Pressure. The impact of the cavity region on the pressure distribution at the base is depicted in Fig. 11 through the time-averaged base pressure at different azimuthal angles, non-dimensionalized with the total pressure at the tunnel inlet $P_{0,t}$. Observing Fig. 11, it is evident that the cavity region has a negligible impact on the time-averaged base pressure distribution at this NPR. This is in agreement with previous two-dimensional and axisymmetric, RANS computations at similar NPR values [10]. The Reynolds-averaged solution yields almost identical results at the base for the cavity and baseline cases (Fig. 11). The pressure levels obtained with RANS are slightly reduced and within 3% with respect to the DDES values. This decrease in the pressure is what was previously identified in Fig. 10 through the time-averaged organization of the base flow.

Furthermore, one can notice the near-symmetrical behavior of the base flow in the $x-y$ plane, shown by the similar values of p_b at angles of 90 deg and 270 deg for both cases, as would be expected for an axially-symmetric geometry. In contrast, the strong asymmetry in the $x-z$ plane caused by the presence of the pylon is also evident. Additionally, the trend captured in the RANS solution agrees with the DDES results. This is a good indication toward using RANS at the conceptual design level, where high-fidelity and computationally-expensive calculations are usually prohibitive.

The preceding analysis shows that the cavity region has a negligible impact on the base flow in a time-averaged sense. The unsteady behavior of the pressure at the base can be examined

through the root mean square (RMS) of the fluctuating component of pressure, defined as $p' = p - \langle p \rangle$ from Reynolds decomposition [42], where p and $\langle p \rangle$ denote the instantaneous and time-averaged pressure values, respectively. Figure 12 depicts the coefficient of the RMS value of the pressure fluctuations along the base wall $C_{p_{rms}}$, defined in Eq. (18)

$$C_{p_{rms}} = \frac{\sqrt{\langle p'^2 \rangle}}{\frac{1}{2} \rho_{\infty,0} U_{\infty,0}^2} \quad (18)$$

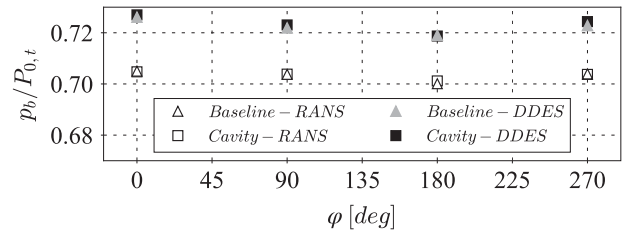


Fig. 11 Azimuthal distribution of the mean base pressure

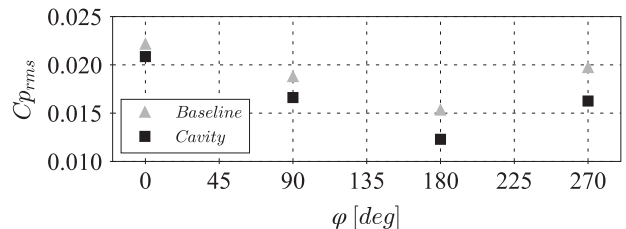


Fig. 12 Azimuthal distribution of the RMS coefficient of base pressure fluctuations

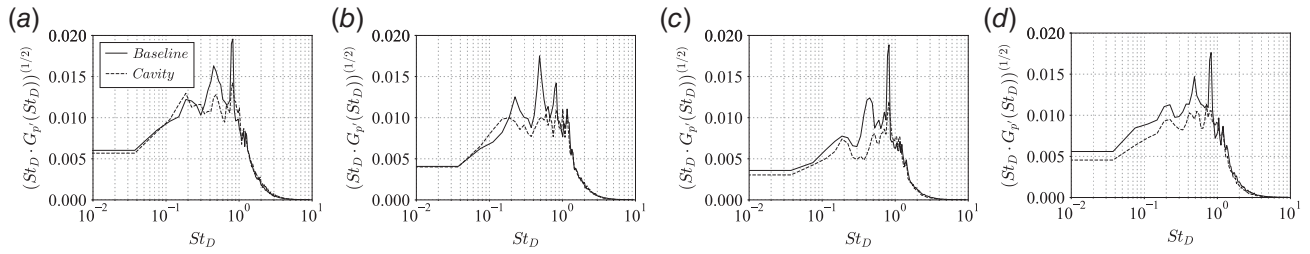


Fig. 13 Base pressure fluctuations spectra at (a) $\phi = 0$ deg, (b) $\phi = 90$ deg, (c) $\phi = 180$ deg, and (d) $\phi = 270$ deg

The level of base pressure fluctuations is similar with previous experimental studies conducted on similar canonical cases and flow conditions [17,66], and falls within the range of approximately 0.015–0.025. Additionally, Fig. 12 shows that the case with the cavity exhibits reduced levels of pressure fluctuations at all azimuthal locations compared to the baseline case. This reduction ranges from 6% at 0 deg angle to values of 20% behind the pylon. This shows that the cavity region has an important impact on the base flow unsteadiness of the system and potentially acts in a damping-like manner. The associated mechanism lies in the entropic effects due to the viscous recirculating flow inside the cavity region. This dissipates part of the mechanical energy of the flow that is transferred into the cavity in the form of upstream-propagating pressure waves [67], while part of that energy is reflected back to the base flow region. This mechanism does not manifest in the baseline case.

3.5 Base Flow Spectral Analysis. The impact of the cavity on the unsteady flow characteristics at the base can be further investigated through the power spectral density of the pressure fluctuations along the base surface. The PSD describes how the mean squared value of wall pressure fluctuations is distributed in the frequency domain [68] since

$$\sigma^2 = \int_0^\infty G_p(f) df \quad (19)$$

As suggested by Owen [69] for the investigation of buffet studies, the power spectral density of the pressure fluctuations obtained in (Pa^2/Hz), is non-dimensionalized over $(Dq_{\infty,0}^2/U_{\infty,0})$, where $q_{\infty,0} = 0.5\rho_{\infty,0}U_{\infty,0}^2$ is the dynamic pressure in the vicinity of the base at 0 deg angle. The corresponding, non-dimensional Strouhal number can then be used. Consequently for a zero-mean variable (i.e., fluctuations) [17,69,70]

$$\begin{aligned} C_{p_{rms}}^2 &= \int_0^\infty G_p(St_D) d(St_D) \\ &= \int_0^\infty St_D G_p(St_D) d(\log St_D) \end{aligned} \quad (20)$$

Therefore, by plotting the spectrum in the form $\sqrt{St_D G_p(St_D)}$ against St_D in linear/log axes, the mean intensity of the fluctuations at the considered frequency range can be obtained [69].

Figure 13 depicts the spectra of the wall pressure fluctuations between the baseline and cavity cases, at different azimuthal locations. The spectra corresponding to the baseline case, exhibits similar organization at all examined locations. A broad peak of the pressure fluctuations spectrum can be identified, centered at approximately $St_D = 0.2$, similar to what was observed for the lower NPR in Fig. 7. This value of non-dimensional frequency has been identified in previous studies for axisymmetric bodies [14,17] and BFSs [11,17,18,22], as well as for more detailed launch vehicle configurations [63]. It is typically associated with the shedding frequency of large-scale, coherent structures. Two additional peaks can be seen in all azimuthal locations centered around $\sim St_D = 0.45$ and 0.81 . Similar frequency content to that captured by the DDES computations has been observed experimentally in Ref. [29] for conditions of $M_\infty = 0.6$ and over-expanded jet of a blunt-based body, as well as in Ref. [22] for an axisymmetric BFS at flow conditions of $M_\infty = 0.7$, although the higher level frequency peaks were weaker.

In the case of BFSs and blunt-based bodies, flow separation is induced by the abrupt geometry change at the base edge. In that context, phenomena associated with the frequency content in the range of $St_D = 0.5 - 0.6$, are caused by smaller structures formed by the shear layer instability in the recirculation region, as described in Refs. [11,17,65]. Additionally, the recirculation region interacts with the jet shear layer and the corresponding flow entrainment effect. This interaction can impact the observed frequency range relative to the typical value of the instabilities associated with the development of free shear layers [11]. Furthermore, dominant frequency content at higher range of $St_D = 0.8$, is associated with the smaller-scale structures located close to the flow reattachment [17]. This is attributed to the interaction between the large-scale structures of the shear layer with the high-speed jet. This interaction causes the formation of smaller scales, with the associated instabilities propagating upstream in the subsonic region.

For the spectra corresponding to the cavity case, a similar low-frequency peak at $\sim St_D = 0.2$ is observed. The higher-frequency peaks at $St_D = 0.45$ and 0.81 can also be identified in the cavity case. However, the associated intensity level of the pressure fluctuations appears notably reduced compared to the baseline case. This is in line with the results shown in Fig. 12 for the rms values of base pressure fluctuations, and demonstrates that the cavity has a notable impact on base flow unsteadiness, especially in higher frequencies.

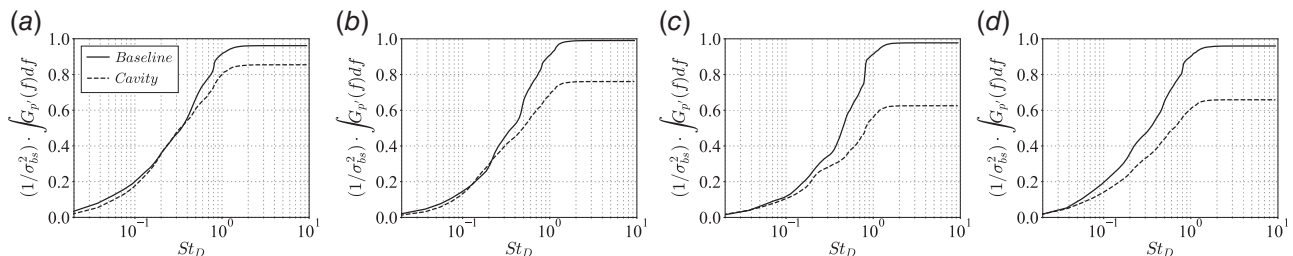


Fig. 14 Cumulative energy of the base pressure fluctuations spectra at (a) $\phi = 0$ deg, (b) $\phi = 90$ deg, (c) $\phi = 180$ deg, and (d) $\phi = 270$ deg

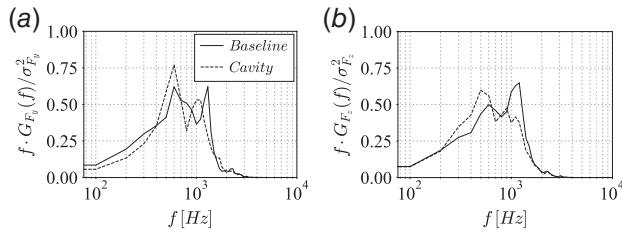


Fig. 15 Pre-multiplied spectra of the side load components: (a) F_y and (b) F_z

The base pressure spectra shown in Fig. 13, reflect the footprint of the propagation of disturbances through the aerodynamic field in the base flow region. The intensity of the pressure fluctuations associated with these disturbances is reduced in the presence of the cavity through the dissipative mechanism described previously [67].

The effect of the cavity on the associated energy in the pressure fluctuations spectra can be approximated through the numerical integration of the spectrum as described in Eq. (19). This is shown in Fig. 14 for the four azimuthal locations. It depicts the cumulative energy of the pressure spectra for both cases, non-dimensionalized with respect to the baseline case. It can be seen that the cavity region exhibits reduced energy in the pressure fluctuations. Additionally, the reduction appears to be notable at frequencies above $St_D = 0.2$, indicating that the cavity has an effect on the interaction of smaller scales and does not impact the large-scale dynamics. The cavity case exhibits energy levels which are 15% (Fig. 14(a)) to 38% (Fig. 14(c)) lower with respect to the baseline case, depending on the azimuthal location.

3.6 Cavity Impact on Nozzle Side Loads. The impact of the cavity on the power spectrum of the side loads F_y and F_z is depicted in Fig. 15. The time distribution of the side load components is obtained through numerical integration of the pressure and viscous stress terms along the nozzle walls. The normalized, pre-multiplied power spectral density is shown. The energy spectra of the side load in the y-direction exhibit a broad peak centered at medium frequencies of around 600 Hz for both configurations. A

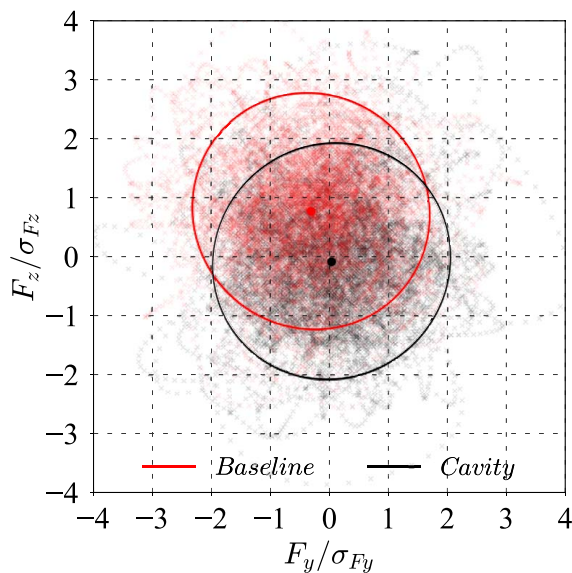


Fig. 16 Polar plot of nozzle side loads (symbols) with their 95% confidence ellipses (solid lines). The dots denote the mean values.

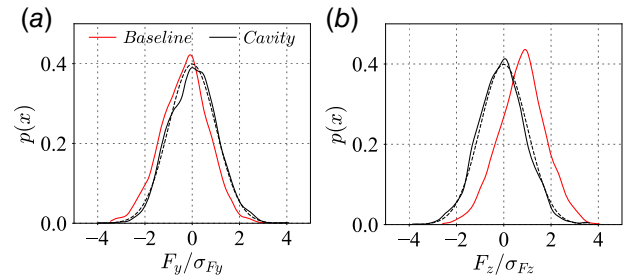


Fig. 17 PDFs of the side load components: (a) F_y and (b) F_z . The dashed line denotes the theoretical Gaussian distribution.

narrower peak is also evident at slightly higher frequencies of 1000 Hz for the cavity case and 1300 Hz for the baseline case.

For the side load in the z -direction, which is parallel to the pylon plane, it can be seen that the cavity case features a broad peak at approximately the same low frequency of 500–600 Hz, while higher frequencies contribute weakly in the energy of F_z . On the contrary, the baseline case exhibits reduced levels of energy at the low-frequency range, and increased levels at higher frequencies of ~ 1300 Hz. This suggests that the level of pressure fluctuations at higher frequencies is reduced in the presence of the cavity relative to the baseline case. This effect is more pronounced in the pylon-parallel plane, in line with the energy of the base pressure spectra (Fig. 14).

Figure 16 shows a polar plot of the side loads for both configurations, along with the corresponding $a = 0.95$ confidence ellipse, which includes 95% of the observations. One can notice the random character of the side loads fluctuations. Additionally, the confidence ellipse becomes a circle for both configurations (i.e., $\sigma_{F_z} = \sigma_{F_y}$), indicating their isotropic nature over time. For the baseline case, the side loads envelope is not centered at zero, as would be the case for an axisymmetric configuration, since $\langle F_y \rangle = \langle F_z \rangle = 0$. In specific, because of the non-axisymmetric topology, the baseline case exhibits positive and negative side loads in the z - and y -directions, respectively, in a time-averaged sense. The magnitude of F_z is almost three times higher compared to F_y due to the asymmetry in the $x-z$ plane. However, the presence of the cavity region shifts the side loads envelope back to zero-mean values, as it would appear for an axisymmetric geometry. This suggests that the impact of the cavity is large enough to affect the time-averaged flow organization and the nozzle side loads. The mean reduction is an order of magnitude in the cavity case. Consequently, the cavity region, which would be essential for future, high-speed systems [10], could limit the undesirable side loads.

The behavior of the side loads can be further understood by looking at the probability density functions (PDFs) of F_y and F_z . This is depicted in Fig. 17. It is clearly evident that both components are normally distributed for both geometries. The baseline case exhibits non-zero mean values and the PDFs appear shifted with respect to the theoretical Gaussian distribution. This shift is increased in the F_z side load. However, the presence of the cavity region, shifts the PDFs back to zero-mean which now fit well the theoretical distribution.

4 Conclusions and Future Work

The impact of a cavity region on the base flow unsteadiness for a sub-scale, high-speed exhaust is investigated numerically in this work. The DDES turbulence modeling approach is used. A direct comparison to an identical, non-cavity geometry is carried out, allowing for the cavity impact to be quantified. The computational approach is validated against experimental data at representative over-expanded nozzle flow conditions, for both configurations.

It is found that the coefficient of base pressure fluctuations is reduced in the cavity case by 6% to as much as 20% behind the pylon. Investigation of the power spectrum of the base pressure fluctuations, reveals that the cavity region reduces the intensity of unsteadiness contained at higher frequencies, above values of $St_D \approx 0.2$. The cumulative energy of base pressure fluctuations spectra is reduced by 15–38% relative to the baseline case, depending on the azimuthal location. This indicates that the cavity region, acts in a damping-like manner for the intensity of unsteadiness at the base flow.

Additionally, the results show that the cavity region has a notable effect on the generated nozzle side loads at over-expanded conditions. It is demonstrated that the side loads of the baseline case, deviate from the theoretical Gaussian distribution, due to three-dimensionality effects. However, the presence of the cavity shifts the side loads behavior back to zero-mean distribution, and their absolute value is reduced by an order of magnitude compared to the baseline case.

The novelty of this work lies in the quantification of the impact of base-embedded cavity regions on the unsteady propulsion aerodynamics of high-speed exhausts. It makes a valuable contribution in the field of unsteady base flow analyses and demonstrates that base-embedded cavities could reduce undesirable nozzle side loads, which are critical during over-expansion [35]. Such cavities will be essential in future systems to allow space for base-embedded nozzles [10]. Therefore, the results of this work can have a long-term impact on the design of future high-speed exhausts. The validated computational approach can be used in future research to further investigate the dominant mechanisms associated with the cavity and pylon as identified in this work, in a more detailed exhaust configuration, to evaluate airframe aerodynamic interference effects.

Acknowledgment

The authors would like to express their gratitude to Reaction Engines Ltd. and the Cranfield Air and Space Propulsion Institute (CASPI) for funding this project and granting permission to publish the findings. The support of Dr. Marco Debiassi and Dr. Mark Finnis during the experimental campaign is greatly acknowledged.

Conflict of Interest

There are no conflicts of interest.

Data Availability Statement

The datasets generated and supporting the findings of this article are obtainable from the corresponding author upon reasonable request.

Nomenclature

f = frequency (Hz)
 k = turbulent kinetic energy (m^2/s^2)
 l = length scale (m)
 u = velocity (m/s)
 D = model diameter (m)
 L = model length (m)
 f_d = DDES shielding function
 C_p = pressure coefficient
 y^+ = non-dimensional wall distance
 p, T = static pressure (Pa) and temperature (K)

Greek Symbols

Δ = grid spacing (m)
 μ = dynamic viscosity ($kg/(m \cdot s)$)

μ_t = eddy viscosity ($kg/(m \cdot s)$)
 ρ = density (kg/m^3)
 τ = viscous shear stress (Pa)
 ϕ = azimuthal angle (deg)
 ω = specific dissipation rate (s^{-1})

Subscripts

b = referring to the base
 bs = referring to the baseline geometry
 w = referring to the wall
 $(\infty, 0)$ = indicating flow conditions at the base and 0 deg angle
 0 = indicating total conditions

Dimensionless Groups

Re_D = Reynolds number based on the model diameter
 St_D = Strouhal number based on the model diameter

Acronyms

FSS = free shock separation
NPR = nozzle pressure ratio
RSS = restricted shock separation
TPR = tunnel pressure ratio

References

- [1] McClinton, C. R., 2007, "High Speed/Hypersonic Aircraft Propulsion Technology Development," *Advances on Propulsion Technology for High-Speed Aircraft, Educational Notes, RTO-EN-AVT-150*, Vol. 1, NATO RTO, Neuilly-sur-Seine, France, pp. 1–32.
- [2] Ingenito, A., 2021, *Subsonic Combustion Ramjet Design*, Springer Nature Switzerland, Cham, Switzerland.
- [3] Murthy, S., and Curran, E., 2001, *Scramjet Propulsion*, AIAA, Reston, VA.
- [4] Bogar, T., Eiswirth, E., Couch, L., Hunt, J., and McClinton, C., 1996, "Conceptual Design of a Mach 10, Global Reach Reconnaissance Aircraft," 32nd Joint Propulsion Conference and Exhibit, AIAA, Paper No. AIAA 1996-2894.
- [5] Daines, R., and Segal, C., 2012, "Combined Rocket and Airbreathing Propulsion Systems for Space-Launch Applications," *J. Propul. Power*, **14**(5), pp. 605–612.
- [6] Varvill, R., and Bond, A., 2003, "A Comparison of Propulsion Concepts for SSTO Reusable Launchers," *J. British Interplanet. Soc.*, **56**, pp. 108–117.
- [7] Dai, J., and Zuo, Q., 2020, "Key Technologies for Thermodynamic Cycle of Precooled Engines: A Review," *Acta Astronaut.*, **177**, pp. 299–312.
- [8] Zhang, J., Wang, Z., and Li, Q., 2017, "Thermodynamic Efficiency Analysis and Cycle Optimization of Deeply Precooled Combined Cycle Engine in the Air-Breathing Mode," *Acta Astronaut.*, **138**, pp. 394–406.
- [9] Tsentis, S., Gkoutzamanis, V., Gaitanis, A., and Kalfas, I., 2021, "Multi-platform App-Embedded Model for Hybrid Air-Breathing Rocket-Cycle Engine in Hypersonic Atmospheric Ascent," *Aeronaut. J.*, **125**(1291), pp. 1631–1653.
- [10] Tsentis, S., Goulos, I., Prince, S., Pachidis, V., and Zmijanovic, V., 2023, "Propulsion Aerodynamics for a Novel High-Speed Exhaust System," *ASME J. Eng. Gas Turbines Power*, **145**(12), p. 121011.
- [11] Deck, S., and Thorigny, P., 2007, "Unsteadiness of an Axisymmetric Separating-Reattaching Flow: Numerical Investigation," *Phys. Fluids*, **19**(6), p. 065103.
- [12] Hammond, W. E., 2001, *Design Methodologies for Space Transportation Systems*, AIAA, Reston, VA.
- [13] Scharnowski, S., and Kähler, C., 2021, "Investigation of the Base Flow of a Generic Space Launcher With Dual-Bell Nozzle," *CEAS Space J.*, **13**(2), pp. 197–216.
- [14] Mariotti, A., and Buresti, G., 2013, "Experimental Investigation on the Influence of Boundary Layer Thickness on the Base Pressure and Near-Wake Flow Features of an Axisymmetric Blunt-Based Body," *Exp. Fluids*, **54**(1612), pp. 1–16.
- [15] Mariotti, A., Buresti, G., and Salvetti, M. V., 2015, "Connection Between Base Drag, Separating Boundary Layer Characteristics and Wake Mean Recirculation Length of an Axisymmetric Blunt-Based Body," *J. Fluids Struct.*, **55**, pp. 191–203.
- [16] Paciorni, R., Sabetta, F., Valenza, F., Fauci, R., Passaro, A., and Baccarella, D., 2013, "Base-Pressure Experimental Investigation on a Space Launcher in Subsonic Regime," *J. Spacecr. Rocket.*, **50**(3), pp. 572–578.
- [17] Déprés, D., Reijasse, P., and Dussauge, J. P., 2004, "Analysis of Unsteadiness in Afterbody Transonic Flows," *AIAA J.*, **42**(12), pp. 2541–2550.
- [18] Weiss, E. P., Deck, S., Robinet, J. C., and Sagaut, P., 2009, "On the Dynamics of Axisymmetric Turbulent Separating/Reattaching Flows," *Phys. Fluids*, **21**(075103), pp. 1–8.
- [19] Deck, S., 2005, "Zonal-Detached-Eddy Simulation of the Flow Around a High-Lift Configuration," *AIAA J.*, **43**(11), pp. 2372–2384.
- [20] Saïle, D., Kühl, V., and Gülhan, A., 2021, "On Subsonic Near-Wake Flows of a Space Launcher Configuration With Various Base Geometries," *Exp. Fluids*, **62**(122), pp. 1–14.

- [21] Saile, D., Kühl, V., and Gülhan, A., 2019, "On the Subsonic Near-Wake of a Space Launcher Configuration With Exhaust Jet," *Exp. Fluids*, **60**(165), pp. 1–17.
- [22] Meliga, P., and Reijasse, P., 2012, "Unsteady Transonic Flow Behind an Axisymmetric Afterbody Equipped With Two Boosters," 25th AIAA Applied Aerodynamics Conference, AIAA, Paper No. AIAA 2007-4564.
- [23] Pain, R., Weiss, P.-E., and Deck, S., 2014, "Zonal Detached Eddy Simulation of the Flow Around a Simplified Launcher Afterbody," *AIAA J.*, **52**(9), pp. 1967–1979.
- [24] Schwane, R., 2015, "Numerical Prediction and Experimental Validation of Unsteady Loads on Ariane5 and Vega," *J. Spacecr. Rockets*, **52**(1), pp. 54–62.
- [25] Mehta, U., Aftosmis, M., Bowles, J., and Pandya, S., 2016, "Skylon Aerospace Plane and Its Aerodynamics and Plumes," *J. Spacecr. Rockets*, **53**(2), pp. 340–353.
- [26] Viswanath, P. R., 1996, "Flow Management Techniques for Base and Afterbody Drag Reduction," *Prog. Aerosp. Sci.*, **32**(2–3), pp. 79–129.
- [27] Tanner, M., 1975, "Reduction of Base Drag," *Prog. Aerosp. Sci.*, **16**(4), pp. 369–384.
- [28] Morel, T., 1990, "Effect of Base Cavities on the Aerodynamic Drag of an Axisymmetric Cylinder," *Aeronaut. Q.*, **30**(2), pp. 400–412.
- [29] Tripathi, A., Manisankar, C., and Verma, S. B., 2015, "Control of Base Pressure for a Boat-Tailed Axisymmetric Afterbody Via Base Geometry Modifications," *Aeros. Sci. Technol.*, **45**, pp. 284–293.
- [30] Nasuti, F., and Onofri, M., 2009, "Shock Structure in Separated Nozzle Flows," *Shock Waves*, **19**, pp. 229–237.
- [31] Östlund, J., Damgaard, T., and Frey, M., 2004, "Side-Load Phenomena in Highly Overexpanded Rocket Nozzles," *J. Propul. Power*, **20**(4), pp. 695–704.
- [32] Frey, M., and Hagemann, G., 2000, "Restricted Shock Separation in Rocket Nozzles," *J. Propul. Power*, **16**(3), pp. 478–484.
- [33] Shimizu, T., Miyajima, H., and Kodera, M., 2006, "Numerical Study of Restricted Shock Separation in a Compressed Truncated Perfect Nozzle," *AIAA J.*, **44**(3), pp. 576–584.
- [34] Hagemann, G., Frey, M., and Koschel, W., 2002, "Appearance of Restricted Shock Separation in Rocket Nozzles," *J. Propul. Power*, **18**(3), pp. 577–584.
- [35] Deck, S., and Nguyen, A. T., 2004, "Unsteady Side Loads in a Thrust-Optimized Contour Nozzle at Hysteresis Regime," *AIAA J.*, **42**(9), pp. 1878–1888.
- [36] Genin, C., and Stark, R. H., 2011, "Side Loads in Subscale Dual Bell Nozzles," *J. Propul. Power*, **27**(4), pp. 828–837.
- [37] Cimini, M., Martelli, E., and Bernardini, M., 2021, "Numerical Analysis of Side-Loads Reduction in a Sub-Scale Dual-Bell Rocket Nozzle," *Flow Turbul. Combust.*, **107**(3), pp. 551–574.
- [38] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K., and Travin, A. K., 2006, "A New Version of Detached-Eddy Simulation, Resistant to Ambiguous Grid Densities," *Theor. Comput. Fluid Dyn.*, **20**(3), pp. 181–195.
- [39] Tsentis, S., Goulos, I., Debiassi, M., Prince, S., Pachidis, V., Zmijanovic, V., and Saavedra, J., 2025, "Wind Tunnel Installation Effects on a High-Speed Exhaust Flow Under Large Blockage," *J. Spacecr. Rockets*, **0**(0), pp. 1–15.
- [40] Zucrow, M. J., and Hoffman, J. D., 1976, *Gas Dynamics*, John Wiley and Sons, New York.
- [41] Sauer, R., 1947, "General Characteristics of the Flow Through Nozzles at Near Critical Speeds," *Memorandum No. 1147*, NACA, Washington, DC.
- [42] Wilcox, D., 2006, *Turbulence Modeling for CFD*, DCW Industries, La Canada, CA.
- [43] Sutherland, W., 1893, "The Viscosity of Gases and Molecular Forces," *Philos. Mag.*, **36**(223), pp. 507–531.
- [44] Sagaut, P., Deck, S., and Terracol, M., 2013, *Multiscale and Multiresolution Approaches in Turbulence*, 2nd ed., Imperial College Press, London, UK.
- [45] Menter, F. R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA J.*, **32**(8), pp. 1598–1605.
- [46] Menter, F., Kuntz, M., and Langtry, R., 2003, "Ten Years of Industrial Experience With the SST Turbulence Model," *Heat and Mass Transfer* **4**, K. Hanjalić, Y. Nagano, and M. Tummers, eds., Begell House Inc., Danbury, CT, pp. 625–632.
- [47] Gritskevich, M. S., Garbaruk, A. V., Schütze, J., and Menter, F. R., 2012, "Development of DDES and IDDES Formulations for the $K-\omega$ Shear Stress Transport Model," *Flow Turbul. Combust.*, **88**(3), pp. 431–449.
- [48] Spalart, P., Jou, W. H., Strelets, M., and Allmaras, S. R., 1997, "Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach," *Advances in DNS/LES*, pp. 137–147.
- [49] Spalart, P. R., 2009, "Detached-Eddy Simulation," *Annu. Rev. Fluid Mech.*, **41**(1), pp. 181–202.
- [50] Menter, F. R., and Kuntz, M., 2004, "Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flow Behind Vehicles," *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains. Lecture Notes in Applied and Computational Mechanics*, Vol. 19, Springer Berlin Heidelberg, Berlin, Heidelberg, pp. 339–352.
- [51] Ansys Inc, 2013, *Ansys Fluent Theory Guide*, Canonsburg, PA.
- [52] Ansys Inc, 2015, *Best Practice: Scale-Resolving Simulations in Ansys CFD*, Canonsburg, PA.
- [53] Ansys Inc, 2021, *Ansys Fluent User's Guide*, Canonsburg, PA.
- [54] Liou, M. S., 1996, "A Sequel to AUSM: AUSM+," *J. Comput. Phys.*, **129**(2), pp. 364–382.
- [55] Ansys Inc, 2012, *Ansys Fluent Meshing User Guide*, Canonsburg, PA.
- [56] Spalart, P., 2001, Young Person's Guide to Detached-Eddy Simulation Grids, Technical Report NASA CR-2001-211032.
- [57] Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., and Raad, P. E., 2008, "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications," *ASME J. Fluids Eng.*, **130**(7), p. 078001.
- [58] Richardson, L. F., and Gaunt, J. A., 1927, "The Deferred Approach to the Limit," *Philos. Trans. R. Soc. London. Series A*, **226**(636–646), pp. 299–361.
- [59] Martelli, E., Saccoccio, L., Ciottoli, P. P., Tinney, C. E., Baars, W. J., and Bernardini, M., 2020, "Flow Dynamics and Wall-Pressure Signatures in a High-Reynolds-Number Overexpanded Nozzle With Free Shock Separation," *J. Fluid Mech.*, **895**(1612), p. A29.
- [60] Chanetz, B., Délyry, J., Gilliéron, P., Gnemmi, P., Gowree, E. R., and Perrier, P., 2020, *Experimental Aerodynamics: An Introductory Guide*, Springer Nature, Switzerland.
- [61] Welch, P., 1967, "The Use of Fast Fourier Transform for the Estimation of Power Spectra: A Method Based on Time Averaging Over Short, Modified Periodograms," *IEEE Trans. Audio Electroacoust.*, **15**(2), pp. 70–73.
- [62] Dubief, Y., and Delcayre, F., 2000, "On Coherent-Vortex Identification in Turbulence," *J. Turbul.*, **1**, p. N11.
- [63] Lüdeke, H., Mulot, J. D., and Hannemann, K., 2015, "Launch Vehicle Base Flow Analysis Using Improved Delayed Detached-Eddy Simulation," *AIAA J.*, **53**(9), pp. 2454–2471.
- [64] Jeong, M., Yun, Y., Heo, S., and Jee, S., 2023, "Delayed Detached-Eddy Simulation of Subsonic Axisymmetric Base Flow," *Inter. J. Aeronaut. Space Sci.*, **24**(4), pp. 1160–1170.
- [65] Reynaud, J., Weiss, P.-E., and Deck, S., 2021, "Numerical Workflow for Scale-Resolving Computations of Space Launcher Afterbody Flows With and Without Jets," *Comput. Fluids*, **226**, p. 104994.
- [66] Eldred, K. M., 2005, "Base Pressure Fluctuations," *J. Acoust. Soc. Am.*, **33**(1), pp. 59–63.
- [67] Searby, G., Habiballah, M., Nicole, A., and Laroche, E., 2008, "Prediction of the Efficiency of Acoustic Damping Cavities," *J. Propul. Power*, **24**(3), pp. 516–523.
- [68] Shin, K., and Hammond, J., 2008, *Fundamentals of Signal Processing for Sound and Vibration Engineers*, Wiley, Chichester, UK.
- [69] Owen, T., 1958, "Techniques of Pressure-Fluctuation Measurements Employed in the RAE Low-Speed Wind-Tunnels," Report 172, North Atlantic Treaty Organization, Advisory Group for Aeronautical Research and Development.
- [70] Mabey, D. G., 1972, "Analysis and Correlation of Data on Pressure Fluctuations in Separated Flow," *J. Aircraft*, **9**(9), pp. 642–645.



DSSPACE

<https://dspace.org/>

Cavity impact on the base flow unsteadiness for a high-speed exhaust system

Tsentis, Spyros; Goulos, Ioannis; Prince, Simon A.; Pachidis, Vassilios; Zmijanovic, Vladeta

2025-12-01

American Society of Mechanical Engineers (ASME)

<https://doi.org/10.1115/1.4069222>

Tsentis S, Goulos I, Prince S, et al., (2025) Cavity impact on the base flow unsteadiness for a high-speed exhaust system. Journal of Turbomachinery, Volume 147, Issue 12, December 2025, Article number 121015