



ASME Accepted Manuscript Repository

Institutional Repository Cover Sheet

Cranfield Collection of E-Research - CERES

ASME Paper

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

Authors: Spyros Tsentis, Ioannis Goulos, Simon Prince, Vassilios Pachidis, Vladeta Zmijanovic

ASME Conf Title: ASME Turbo Expo 2024: Turbomachinery Technical Conference and Exposition

Volume/Issue: _ Volume 1: Aircraft Engine _ Date of Publication (VOR* Online): _28 August 2024

ASME Digital Collection

URL: <https://asmedigitalcollection.asme.org/GT/proceedings/GT2024/87929/V001T01A040/1203748>

DOI: <https://doi.org/10.1115/GT2024-126486>

*VOR (version of record)

CAVITY IMPACT ON THE BASE FLOW UNSTEADINESS FOR A HIGH-SPEED EXHAUST SYSTEM

Spyros Tsentis^{1,*}, Ioannis Goulos¹, Simon Prince¹, Vassilios Pachidis¹, Vladeta Zmijanovic²

¹Cranfield University, Bedford, United Kingdom

²Reaction Engines Ltd., Oxford, United Kingdom

ABSTRACT

Future propulsion systems will be essential to enable sustainable high-speed flight and routine space access. Such concepts usually employ base-embedded, convergent-divergent nozzles and cavity regions to facilitate their mission, thus altering the flow dynamics at the base notably in comparison to contemporary launch vehicles. This paper presents a numerical investigation on the impact of a cavity region on the base flow unsteadiness for a sub-scale, high-speed exhaust system at over-expanded mode. The fully-installed model in the test section of a wind tunnel is employed to facilitate an ongoing experimental campaign. The Delayed Detached Eddy Simulation turbulence modeling approach is utilized to investigate the flow at the base. The configuration featuring the cavity is directly compared to a baseline apparatus, where the cavity has been removed, thus allowing for the impact of the cavity to be identified. Results show that the cavity region can reduce the base pressure fluctuations up to 20% and acts in a damping-like manner for the base flow unsteadiness. The total energy of the pressure fluctuations spectrum at the base can be reduced by as much as 38% compared to the baseline configuration. However, the impact of the cavity on the time-averaged pressure distribution at the base is negligible. Finally, the cavity is found to have a notable effect on the nozzle side loads, which are reduced by an order of magnitude compared to the baseline case, and behave in an axisymmetric manner. This indicates that the cavity could act as a passive flow control mechanism for side loads reduction.

Keywords: Base Cavity, Hybrid RANS/LES, DDES, Over-expansion, Side Loads, Base Drag, Passive Flow Control

NOMENCLATURE

Roman letters

C_p	pressure coefficient
$C_{V,s}$	nozzle standard velocity coefficient
D	model diameter [m]

f	frequency [Hz]
F_G	gauge stream force [N]
L	model length [m]
p	static pressure [Pa]
R	model radius [m]
T	static temperature [K]
y^+	non-dimensional wall distance

Greek letters

Δ	grid spacing
θ, ϕ	fluid force in drag and thrust domains [N]
σ	standard deviation
τ	viscous shear stress [Pa]

Dimensionless groups

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

Subscripts

bs	referring to baseline case
rms	root mean square
s	standard
w	referring to the wall
$\infty, 0$	indicating flow conditions at the base and 0° angle
0	indicating total conditions

Acronyms

DDES	Delayed Detached Eddy Simulation
FSS	Free Shock Separation
GPF	Gross Propulsive Force
LES	Large Eddy Simulation
NPR	Nozzle Pressure Ratio
PDF	Probability Density Function
PSD	Power Spectral Density
RANS	Reynolds-averaged Navier-Stokes
RSS	Restricted Shock Separation
TPR	Tunnel Pressure Ratio

*Corresponding author: spyros.tsentis@cranfield.ac.uk

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, 2]. In this context, the most commonly found approaches include systems which feature no rotating components, namely ramjets [3] and supersonic combustion ramjets (scramjets) [4] for the supersonic and hypersonic regimes, respectively. These systems however, do not feature the capability of generating static thrust, since their operating principle is based on ram air compression. Combined-cycle concepts have been developed to address this issue, where two (or more) propulsion systems are usually combined to cover a bigger Mach number range from static conditions to very high-speeds. These include turbine-based [5] as well as rocket-based [6] combined-cycle concepts, among others.

Additionally, routine space travel has been the topic of intense research and engineering efforts, mainly focusing on the development of re-usable systems [7]. A technology that appears promising and can be employed to establish both improved space access as well as high-speed, intercontinental travel capabilities is based on the concept of pre-cooled combined-cycle engines [8]. This new class of engines, deeply cools down the incoming air prior to compression, therefore substantially 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 [9–11]. Derivatives of this concept, tailored for high-speed flight instead of space access can also be found in the open literature [12].

While the advantage of these systems is well-established from a thermodynamic perspective, their overall aero-propulsive behavior is not yet fully understood. In specific, these concepts usually feature high-speed nozzles at the base, similar to contemporary rockets. However, the propulsive nozzle in most rocket-based systems usually extends downstream of the base, while in the aforementioned combined-cycle 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, owing to the embedded-nozzle design, these systems often 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 base drag at specific flow conditions by as much as 12% [13]. However, the potential impact of such cavities on the unsteady base flow characteristics and nozzle side loads, has not been reported in previous studies.

1.2 Unsteady base flow features

Base flow dynamics are of major importance for the design and control of launch vehicles, owing to the massive separation that occurs at the base and near-wake regions, which is typically characterized by highly unstable features [14]. This region is

also characterized by very low pressure levels, therefore, substantially increasing base drag [15]. 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 [16] and axisymmetric configurations. In the context of axisymmetric topologies, many studies focus on blunt-based geometries [17–19] and backward facing steps (BFS) for both power-off [18, 19] and power-on [20] conditions.

Deprès and Reijasse [21] investigated experimentally the unsteady afterbody flow characteristics at transonic conditions for both blunt-based and BFS axisymmetric configurations, with and without a propulsive jet. They reported the length extension of the nozzle in the BFS case 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 [14] and Weiss et al. [22] examined the dynamics of such separating/reattaching flows for an axisymmetric BFS geometry, with and without a jet respectively, employing the Zonal Detached Eddy Simulation (ZDES) approach [23] and identified dominant large-scale motions at $St_D \sim 0.2$, corroborating the experimental observations. Saile et al. [24] investigated experimentally a similar BFS topology, focusing on the impact of nozzle length extension from the base 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 [25]. They concluded that the nozzle extension length is important in the base flow unsteadiness, with the worst scenario being the length at which the mean shear layer reattaches just on the tip of the nozzle [25]. Similar configurations have been studied by Statnikov et al. [26] at the hypersonic regime ($M_\infty=6$) and at the presence of a highly under-expanded jet with a nozzle pressure ratio (NPR) of 100. A subsonic cavity was formed around the step extension due to the displacement effect of the jet plume and the shear layer shedding, which amplified the shear layer instability.

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 [27] and computationally [28] 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, from numerical calculations as well as experimental activities, can be found in the study of Schwane [29]. The studies mentioned previously, demonstrate the significant impact of base geometry on the mean and unsteady base flow characteristics. Additionally, the widely selected topology found in most of base flow studies, is inherently connected with the design aspects of contemporary launch systems, where the propulsive nozzle extends downstream of the base. This configuration can be simplified and examined from a canonical flow physics perspective, with a cylindrical BFS configuration. However, for most high-speed, combined-cycle concepts

as those discussed earlier, the nozzle is typically embedded at the base [30]. Consequently, separate attention must be given to those configurations.

A number of passive flow control techniques have been studied, mainly with the focus of reducing base drag and controlling the base flow of bodies of revolution [31] at the absence of a jet. They are mainly associated with base geometry modifications, including base cavities, locked vortex afterbodies as well as boat tailing [17, 31]. A comprehensive review on flow control techniques for base drag reduction can be found in the study of Tanner [32]. Base cavities in specific, have been traditionally studied for base drag reduction capabilities and base pressure control [31, 32]. 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 [33]. Tripathi et al. [34], examined experimentally the effectiveness of different base geometries for base pressure control, including a base cavity case of $l/D=0.12$, with l and D being the cavity length and model diameter, respectively. 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 [32, 34]. However, advanced propulsion concepts with embedded nozzles, could feature cavities as an integrated design aspect to allow space for nozzle gimbaling, with relatively high l/D ratios. Therefore, separate attention must be given to study the impact of such cavities on the unsteady base flow and nozzle characteristics of future, high-speed exhaust systems.

1.3 Nozzle side loads

The performance of launch vehicles is strongly connected to the characteristics and operation of the propulsive nozzle(s). The operation of the nozzle depends on the NPR and could be in over, under or ideal expansion. Over-expansion is usually more important compared to under-expansion, in terms of nozzle performance. Over-expansion occurs at low altitudes and during engine start-up and shut-down, and involves flow separation at the nozzle walls owing to a large adverse pressure gradient, therefore creating undesirable side loads [35, 36]. Two states of flow separation have been observed in convergent-divergent (C-D) nozzles, namely Free Shock Separation (FSS) [35] and Restricted Shock Separation (RSS) [35, 37]. 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 [38]. In truncated, ideal-contoured (TIC) nozzles [39], compression waves do not coalesce to form an internal shock, and therefore only FSS state is observed for this type of nozzle contour [35].

A number of studies can be found regarding side loads in rocket nozzles at over-expanded conditions. Hageman et al. [40] compared a thrust-optimized parabolic (TOP) and a TIC nozzle contour in a sub-scale, cold-flow test campaign, to examine the generated side loads at over-expanded conditions. They reported that the side loads generated in the TIC nozzle were approximately one-third with respect to the TOP nozzle, owing to the transition from FSS to RSS in the parabolic nozzle contour. Deck and

Nguyen [41] conducted numerical studies (employing Unsteady Reynolds-averaged Navier-Stokes or URANS computations) 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 FSS state is dominated by low-frequency energy (less than 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 [42], and found to have reduced side loads compared to TIC nozzles. More recently, Cimini et al. [43] carried out numerical simulations for an over-expanded, dual bell nozzle, using the Delayed Detached Eddy Simulation (DDES) approach [44]. 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 [36]. While the available studies on side loads generation during the over-expanded state of several types of C-D nozzles are numerous in the open literature, the impact of a base cavity region 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 [13].

1.4 Scope of 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 sub-scale, high-speed exhaust system at over-expanded conditions. The examined apparatus is representative of future, advanced propulsion concepts. The analysis involves a fully-installed model in the test section of a wind-tunnel to facilitate an ongoing experimental campaign. The Delayed Detached Eddy Simulation (DDES) [44] turbulence modeling approach is selected among hybrid Reynolds-averaged Navier-Stokes/Large Eddy Simulation (RANS/LES) methods [45] with scale-resolving capabilities, to reduce the computational cost. A direct comparison of the exhaust system against an identical, baseline case without the cavity region is performed. The instantaneous and time-averaged flow organization at the base is identified and discussed. The effect of the cavity on the mean and unsteady base pressure is investigated, in terms of pressure coefficient and power spectrum energy of the pressure fluctuations. Additionally, the impact of the cavity on the unsteady side loads of the over-expanded TIC nozzle is quantified and reported. To the authors' best knowledge, the impact of a base-embedded cavity region on the unsteady flow characteristics at the base and side loads for a high-speed exhaust system, has not been examined in previous studies.

2. METHODOLOGY

2.1 Experimental details

This study facilitates an experimental campaign which is currently ongoing. In this section a brief description of the experimental rig is presented. The activities are designed to be performed at the closed-circuit, ejector-driven, transonic wind tunnel (TSWT) of the Defence Academy of the United Kingdom. The facility has a working test section of $H \times h = 0.228 \times 0.206$ m

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 [46]. 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 (BR) of approximately 16.5%, the approaching flow conditions are dependent on the flow conditions at 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 inlet total pressure of the nozzle $P_{0,n}$ over the outlet static pressure p_{out} .

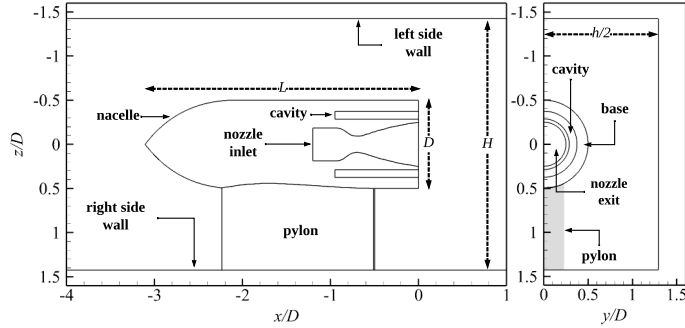


FIGURE 1: SCHEMATIC OF THE TEST SECTION AND MOUNTED MODEL [46]

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 , as indicated in the top schematic of Fig. 2. 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 nacelle. The cavity region has a depth of $l/D=0.95$ and a corresponding thickness of $0.0875D$. The two systems allow for a direct comparison and therefore, identification of the impact of cavity on the base flow unsteady characteristics. The two configurations shown in Fig. 2, are referred to as baseline and cavity case, throughout the rest of this study.

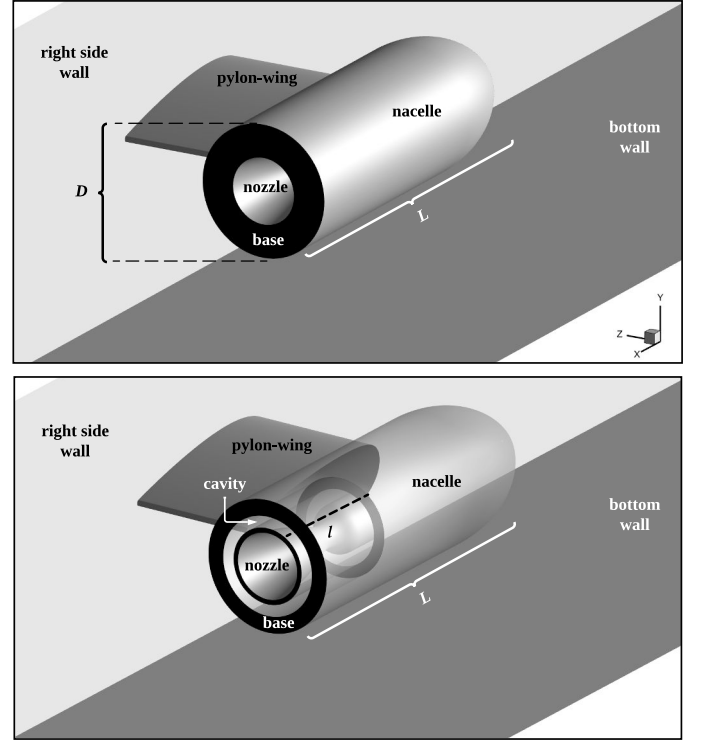


FIGURE 2: INVESTIGATED SYSTEMS; BASELINE CASE (TOP) AND CAVITY CASE (BOTTOM)

The wall contour of the nozzle in the divergent section is generated using the axisymmetric Method of Characteristics (Moc) procedure [39] 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 [47] 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 degrees, and therefore, producing a design exit Mach number of approximately 3.4 and design NPR_d of 59. In this work, computations are performed for conditions of $TPR=1.28$ and $NPR=10$, for both baseline and cavity cases.

2.3 Performance accounting

Figure 3 depicts the methodology employed for the performance accounting in this study. It is based on a well-established thrust and drag book-keeping system [48]. A near-field approach is selected to allow direct access to the constituent elements of total forces, and therefore, evaluation of cavity impact on mean and unsteady side loads [46]. It is of specific interest to quantify the impact of the cavity region on the side loads generated in the nozzle walls at over-expansion state. The aerodynamic forces exerted by the fluid on the wall surfaces, that reside within the thrust and drag domains are denoted as θ and ϕ , respectively. These are obtained through numerical integration of the gauge static pressure and shear stress terms along the viscous surfaces, as described in Eq. (1). The gauge stream force at the nozzle inlet and outlet is calculated through numerical integration of the momentum flux and pressure terms at the corresponding boundary

(Eq. (2)).

$$\theta, \phi = \iint_S (p - p_\infty) \sin a \, dS + \iint_S \tau_w \cos a \, dS \quad (1)$$

$$F_G^x = \iint_{Area} \rho u_x^2 \, dA + \iint_{Area} (p - p_\infty) \, dA \quad (2)$$

Here, p is the local static pressure; p_∞ is the ambient pressure taken at the tunnel outlet (p_{out}); τ_w denotes the local shear stress; a is the local surface angle measured from the axial direction and dS denotes the elemental surface area. In Eq. (2), ρ is the fluid density, u_x is the velocity component in the axial direction and dA is the elemental surface area of the corresponding boundary.

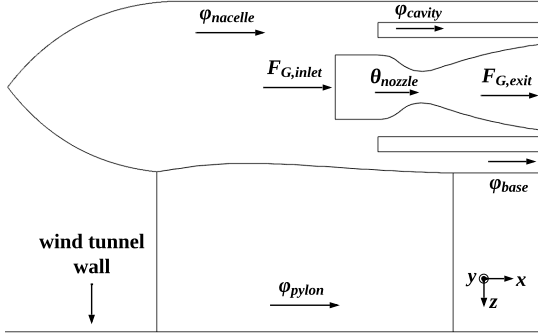


FIGURE 3: THRUST AND DRAG ACCOUNTING SYSTEM

The standard velocity coefficient $C_{V,s}$ is used for the assessment of the aerodynamic performance of the TIC nozzle. It is defined as the ratio of the standard gross thrust $F_{G,s}$, over the ideal thrust (Eq. (3)), that is the thrust that would be produced under isentropic, fully-expanded flow to the ambient pressure. The standard gross thrust is obtained based on control volume theory within the nozzle flow as $F_{G,s} = F_{G,inlet} - \theta_{nozzle}$. The ideal exit velocity V_{ideal} represents the jet exit velocity that would be achieved under fully isentropic expansion to the ambient static pressure (Eq. (4)).

$$C_{V,s} = \frac{F_{G,s}}{\dot{m}_{actual} V_{ideal}} \quad (3)$$

$$V_{ideal} = \sqrt{\frac{2\gamma RT_0}{(\gamma - 1)} \left(1 - \left(\frac{1}{NPR} \right)^{\frac{\gamma-1}{\gamma}} \right)} \quad (4)$$

In Eq. (4), R is the gas constant, γ is the ratio of specific heats and T_0 is the total temperature at the nozzle inlet. The summation of the aerodynamic forces that reside within the drag domain constitute the total modified drag force [48] as:

$$D^* = \phi_{nacelle} + \phi_{base} + \phi_{cavity} + \phi_{pylon} \quad (5)$$

Finally, the gross propulsive force, defined as the overall aerodynamic force produced by the system in the free-stream flow direction, can be obtained through Eq. (6).

$$GPF = F_{G,s} - D^* \quad (6)$$

2.4 Computational strategy

The model used to obtain the field variables is based on solving the Reynolds-averaged (and filtered for scale resolving simulation) version of the three-dimensional Navier-Stokes equations describing the conservation of mass, momentum and energy (Eqs. (7)-(9)):

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (7)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial(t_{ij} + \rho \tau_{ij})}{\partial x_j} = 0 \quad (8)$$

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho u_j E + p u_j)}{\partial x_j} - \frac{\partial(u_i(t_{ij} + \rho \tau_{ij}))}{\partial x_j} + \frac{\partial q_j}{\partial x_j} = 0 \quad (9)$$

In the equations above, 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:

$$t_{ij} = 2\mu S_{ij} \quad (10)$$

$$\rho \tau_{ij} = 2\mu_t S_{ij}^* - \frac{2}{3}\rho k \delta_{ij} \quad (11)$$

$$S_{ij}^* = S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad (12)$$

where S_{ij} denotes the mean strain of rate tensor; μ is the dynamic viscosity and is calculated based on Sutherland's law [49]; 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. (13):

$$q_j = -c_p \left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial T}{\partial x_j} \quad (13)$$

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 [50], not repeated here for brevity.

Owing to wall-bounded flow topology examined, as well as to 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 $Re_D = 1.03 \cdot 10^6$ at the vicinity of the base, the turbulence modeling approach adopted herein is the DDES [44]. This approach belongs to the hybrid RANS/LES methods [45] and its implementation in this work is based on the $k - \omega$ shear stress transport (SST) eddy viscosity model, introduced by Menter [51]. The model solves two transport equations for turbulent kinetic energy k and specific dissipation rate ω (Eqs. (14), (15)). Its main formulation is briefly reported here for convenience (Eqs. (16)-(25)) and to facilitate the implementation of the DDES approach based on this turbulence model.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \widetilde{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] \quad (14)$$

$$\begin{aligned} \frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} = & a\rho S^2 - \beta\rho\omega^2 + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] \\ & + 2(1 - F_1)\rho\sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (15)$$

In Eq. (14) the production term of turbulent kinetic energy P_k is defined as:

$$P_k = \mu_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (16)$$

A production limiter is used in SST formulation of the $k-\omega$ model to prevent the build-up of turbulence in stagnation regions [52]:

$$\widetilde{P}_k = \min(P_k, 10\beta^* \rho k \omega) \quad (17)$$

The turbulent eddy viscosity is defined as:

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (18)$$

In the equation above, S is the magnitude of the strain rate tensor and its use instead of vorticity Ω in the eddy viscosity definition is a slight modification over the original formulation of the $k-\omega$ SST model [52]. The blending function F_1 is given by:

$$F_1 = \tanh(\arg_1^4) \quad (19)$$

$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho\sigma_\omega k}{CD_{k\omega} y^2} \right] \quad (20)$$

with y being the distance to the nearest wall surface and $CD_{k\omega}$ is the positive portion of the cross-diffusion term appearing in the transport equation of ω (i.e., the last term in RHS of Eq. (15)), defined as:

$$CD_{k\omega} = \max \left(2\rho\sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right) \quad (21)$$

F_1 is equal to one inside the boundary layer ($k-\omega$ model) and switches to zero far away from solid surfaces ($k-\epsilon$ model) because of the dependency to $1/y$ or $1/y^2$ in all three terms of Eq. (20). The second blending function F_2 appearing in Eq. (18) is defined by:

$$F_2 = \tanh(\arg_2^2) \quad (22)$$

$$\arg_2 = \max \left[2 \frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right] \quad (23)$$

All model constants are computed by a blend between the corresponding constants of the $k-\epsilon$ and $k-\omega$ model. For instance, if ϕ_1 represents a generic constant of the $k-\omega$ model and ϕ_2 represents the same for the equations of $k-\epsilon$, then the model constants are obtained as [45]:

$$\phi = \phi_1 F_1 + \phi_2 (1 - F_2) \quad (24)$$

The constants of this model are provided below.

$$\begin{aligned} \sigma_{k1} &= 0.85, \quad \sigma_{\omega1} = 0.5, \quad \beta_1 = 0.075, \quad a_1 = 0.31 \\ \sigma_{k2} &= 1.0, \quad \sigma_{\omega2} = 0.856, \quad \beta_2 = 0.0828, \quad \beta^* = 0.09 \\ \gamma &= \frac{\beta}{\beta^*} - \frac{\sigma_\omega \kappa^2}{\sqrt{\beta^*}}, \quad \kappa = 0.41 \end{aligned} \quad (25)$$

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 [52, 53]. The transport equation of k in this formulation reads as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] + \widetilde{P}_k - \rho \sqrt{k^3} / l_{DDES} \quad (26)$$

The DDES length scale is defined as follows:

$$l_{DDES} = l_{RANS} - f_d \max(0, l_{RANS} - l_{LES}) \quad (27)$$

$$l_{LES} = C_{DES} \Delta \quad (28)$$

$$l_{RANS} = \frac{\sqrt{k}}{\beta^* \omega} \quad (29)$$

$$C_{DES} = C_{DES1} F_1 + C_{DES2} (1 - F_1) \quad (30)$$

where $C_{DES1}=0.78$ and $C_{DES2}=0.61$ are calibration constants, 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. (31).

$$f_d = 1 - \tanh \left[\left(C_{d1} \frac{\nu + \nu_t}{\kappa^2 y^2 \sqrt{0.5(S^2 + \Omega^2)}} \right)^{C_{d2}} \right] \quad (31)$$

Here S and Ω is 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 [53]. The introduction of this blending function serves as a modification of the original DES formulation (known as DES97) introduced by Spalart [54]. It guarantees that the solver remains in 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 "grey area") [44, 55], 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 [56] and can inversely impact a rather good RANS solution in the boundary layer.

Computations are carried out by means of a compressible, density-based and implicit solver [57]. 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 (CD) are prone to solution oscillations in the velocity field [58, 59].

In this work, the use of the bounded-central-differencing (BCD) scheme, led to well-known numerical stability issues [58]. A flux-vector splitting scheme (Advection Upstream Splitting Method - AUSM+) is employed for the calculation of the inviscid fluxes. AUSM+ was introduced by Liou [60] and provides exact resolution of shock and contact discontinuities, and is also free of the instability often involved in multi-dimensional shock waves, known as the carbuncle phenomenon. A least-squares cell-based method is used for calculation of the flow-field gradients. Owing to the cold flow of the nozzle, air is modeled as an ideal gas. An 8th order piece-wise polynomial is employed for the calculation of specific heat capacity at constant pressure C_p as a function of static temperature, while thermal conductivity is obtained according to kinetic theory [57]. 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 [58].

The computational domain employed for the simulations is shown in Fig. 4. The top and right side tunnel walls are hidden for visibility reasons. It consists of the full experimental apparatus, where the TSWT test section is extended by a factor of 10 times the model's length downstream of the base (corresponding to 125 nozzle radii) and a factor of 2 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. 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 which is set equal to 1.28 in the computations. The nozzle inlet conditions are provided through a pressure inlet BC, prescribing the total pressure and total temperature. Owing to the cold flow tests, 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.

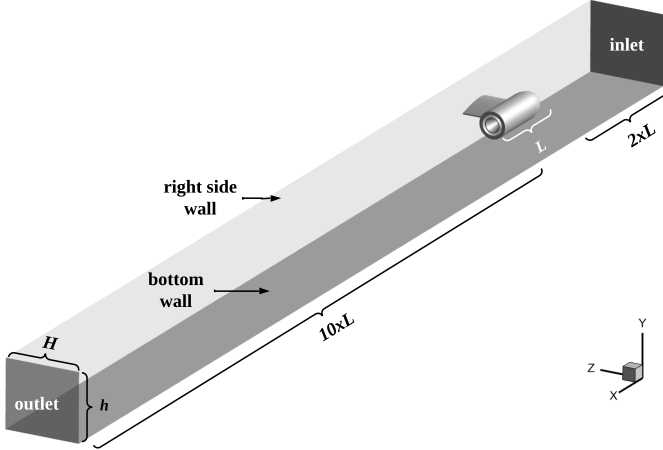


FIGURE 4: EMPLOYED COMPUTATIONAL DOMAIN

The domain is discretized using a hybrid meshing approach [61]. It is noted that grid convergence in scale resolving simulations is subtle compared to DNS (Direct Numerical Simulation) 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 in [62]. In this study, a precursor, converged RANS solution is used to initialize the DDES computation. The meshing strategy is initially informed from previous work [46], where a second order Grid Convergence Index (GCI) [63] study is performed, based on the generalized Richardson extrapolation procedure [64]. The GCI corresponding to the medium mesh for the nozzle standard velocity coefficient $C_{V,s}$ as well as for the GPF was found equal to 0.42% and 0.68%, respectively. Additionally, the apparent order p , is 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 [63]. This meshing approach is employed as a baseline, since the part of the domain where the model acts in 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 [58].

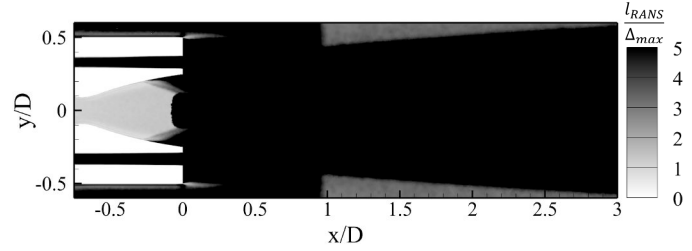


FIGURE 5: RATIO OF RANS TO GRID LENGTH SCALE AT THE BASE AND NEAR-WAKE REGIONS FOR THE CAVITY CASE

The baseline grid resolution is then further improved locally, through the creation of a mesh refinement region at the base and near-wake regions, which could be seen as a "focus region" of the simulation as recommended in [62]. The maximum cell size in that region is approximately equal to $\Delta_{max}=0.23 \cdot \delta$, where δ is the boundary layer thickness evaluated from a precursor RANS simulation 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 local grid Δ_{max} to the RANS length scale l_{RANS} (see Eq. (29)) is sufficiently small to allow resolution of the initial flow instability of the separating shear layer from the base [58]. The final mesh features values of $(\Delta_{max}/l_{RANS}) < 0.1$ as recommended in [58] at the base, cavity and wake regions. This is depicted in Fig. 5, where the ratio (l_{RANS}/Δ_{max}) is shown at the part of the domain which is of interest. The plotted values have been limited in the range 0-5, since 5 (or equivalently 0.2 for Δ_{max}/l_{RANS}) is considered a lower resolution limit. In practice, 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 part of the initial shear layer has slightly smaller resolution. However this is not necessarily a problem, since typically the shear layer thickness increases largely downstream of the separation edge and thus (l_{RANS}/Δ_{max}) increases quickly to sufficient values [58]. Given the desired cell isotropy in the LES part of the domain [62], 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

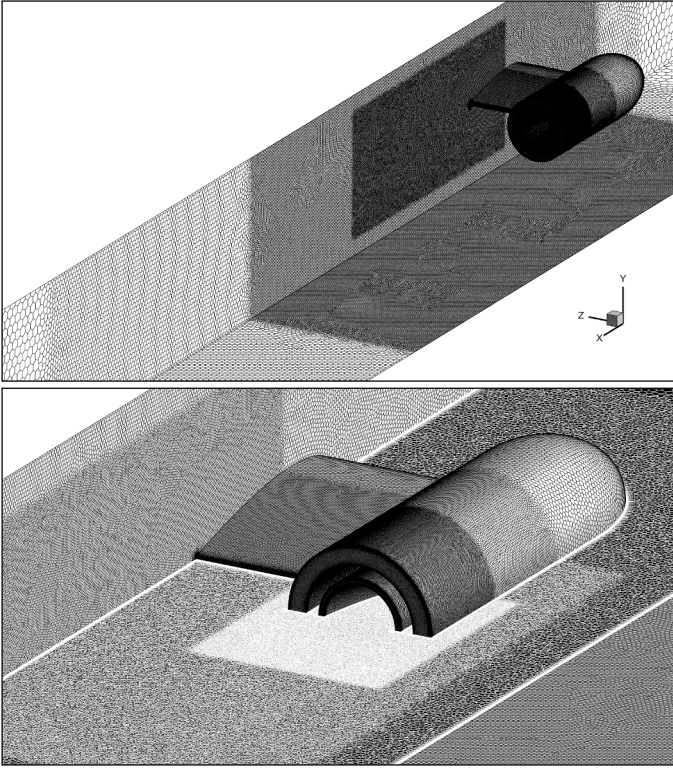


FIGURE 6: SURFACE MESH (TOP) AND CLOSE-UP VIEW ON THE BASE REGION WITH VOLUME MESH SHOWN ALONG THE SYMMETRY PLANE (BOTTOM)

(such as in acoustics), delaying the initial instability could result in loss of spectral information.

The final computational mesh is shown in Fig. 6 along with 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, employing polyhedral elements, which are typically more efficient in terms of cell-count compared to tetrahedrons. The grid corresponding to the cavity case, comprises a total 25 million cells, while the one for the baseline, features approximately $23 \cdot 10^6$ 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 at the vicinity of the base at an azimuthal angle of 0° (see Fig. 7). The use of this velocity is required due to the blockage in the TSWT test section which creates asymmetrical velocity distribution in the azimuthal direction. It also facilitates the generalization of the results, which are usually reported on the basis of the free-stream velocity at separation in the case of axisymmetric bodies [14, 21, 26, 34]. The employed time step size is selected as

such, to ensure that the local Courant–Friedrichs–Lewy (CFL) 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 [58], 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. 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 [65], which is unfeasible (~ 600 days of wall-time with our resources). Additionally, it is not of critical importance in assessing the impact of cavity on large-scale, base flow dynamics. After a transient phase, the unsteady calculation initiates, allowing for collection of statistics. The useful unsteady calculation is performed over a period of $TU_{\infty,0}/D = 670$. The computations for each configuration are carried out on 256 CPU cores of AMD EPYC-7543 processors of the high performance computing facility of Cranfield University and have a total computational cost of approximately 0.2 Mio CPU hours each.

3. RESULTS AND DISCUSSION

3.1 Test case flow topology

Owing to the blockage and therefore, varying flow field throughout the domain, it is necessary to establish the flow conditions in the vicinity of the model's base as well as the state of the nozzle at $TPR = 1.28$ and $NPR = 10$. Figure 7 shows the Mach

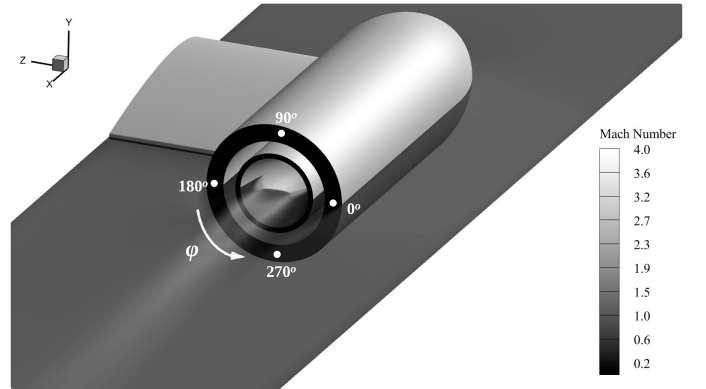


FIGURE 7: REYNOLDS-AVERAGED FLOW TOPOLOGY OF THE TEST CASE

number distribution along the symmetry plane (xz) 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° , 90° , 180° and 270° , correspond to locations where dynamic pressure measurements will be extracted at the experimental campaign through miniaturized dynamic sensors, and have a radial location of $r/R = 0.875$, with R being the radius of the model. Consequently, these are selected as monitoring points in the computations to establish consistency with the experimental data. It is important to note that these points are on

the base surface and at identical spatial locations in both examined configurations. Therefore, the impact of the cavity on the flow unsteadiness is evaluated at the same location on the base. As depicted in the figure, the flow conditions at the vicinity of the base correspond to a Mach number of approximately 0.65 (at 0°) and the nozzle is over-expanded, exhibiting a Mach disk reflection and being in FSS state [35]. A notable region of flow separation within the nozzle is also visible in the figure. This region is responsible for side loads generation within the nozzle and therefore, the cavity impact on the side load generation must be identified.

3.2 Instantaneous and time-averaged flow organization

The salient characteristics of the instantaneous flow field at the base are shown in Fig. 8 for the baseline case. Coherent structures are identified through the visualization of positive iso-surfaces of the Q-criterion [66], defined as the second invariant of the velocity gradient tensor ∇u :

$$Q = \frac{1}{2}(\Omega_{ij}\Omega_{ij} - S_{ij}S_{ij}) \quad (32)$$

where Ω_{ij} and S_{ij} are the anti-symmetric and symmetric components of the velocity gradient tensor, respectively. Observing Fig. 8, 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 [14] as well as in more detailed configurations on the base flow of the Ariane 5 European launcher [67]. It is worth noting that unlike in many studies focused on axisymmetric launch vehicles and BFS geometries [14], where these azimuthal structures are initially toroidal in shape due to uniform free-stream velocity, in the present work these structures quickly lose their azimuthal coherence. This is owing to the high blockage in the TSWT test section, essentially causing a non-uniform velocity distribution in the azimuthal direction and therefore, imposing an azimuthal 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 [14] and blunt-based bodies [19]. 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 [58]. This is further confirmed by examining of the blending function f_d (Eq. (31)) [44]. The two cases examined, exhibit similar base flow characteristics in terms of positive Q-criterion iso-surfaces.

The instantaneous aerodynamic flow field of the unsteady computations is time-averaged using data at discrete time instances of $10\Delta_t$ for the entire computation period. Figure 9 depicts the time-averaged Mach number distribution and streamlines, along the mid-plane (xz) 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 organised in a similar manner, for both configurations examined. Both cases exhibit a similar jet vectoring effect due

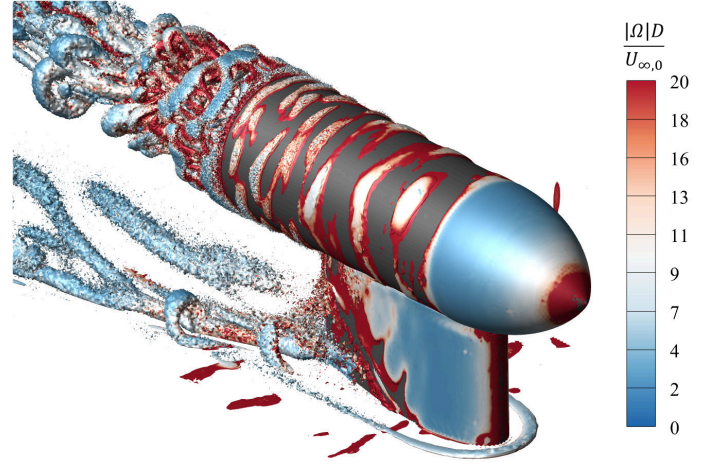


FIGURE 8: INSTANTANEOUS Q-CRITERION ISO-SURFACES COLOURED BY NON-DIMENSIONAL VORTICITY MAGNITUDE

to the presence of the pylon [46]. A pair of low-speed, counter-rotating recirculation regions is established at the separated flow immediately downstream of the base, between the free stream 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 shear imposed by the jet. 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. 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, both using Reynolds-averaged solutions [13] as well as scale-resolving simulations [65]. It is related to the curvature of the Mach disk reflection and a detailed description can be found in [35]. The baseline case periodically exhibited this effect as well, however it was less intense and in a time-averaged sense, the vortices do not develop.

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 with RANS in both cases examined. A strong recirculation region can be seen, with height approximately equal to two thirds of the base surface. The core of this 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. 9 through comparison of the nozzle flow expansion between RANS and DDES. The flow at the nozzle divergence in the RANS case achieves slightly higher expansion levels compared to DDES. This can be readily observed through the location of the Mach disk reflection which is further downstream in the RANS computations for both baseline and cavity cases, with respect to the DDES calculation.

3.3 Cavity impact on base pressure

The impact of the cavity region on the pressure distribution at the base is depicted in Fig. 10 through the time-averaged base

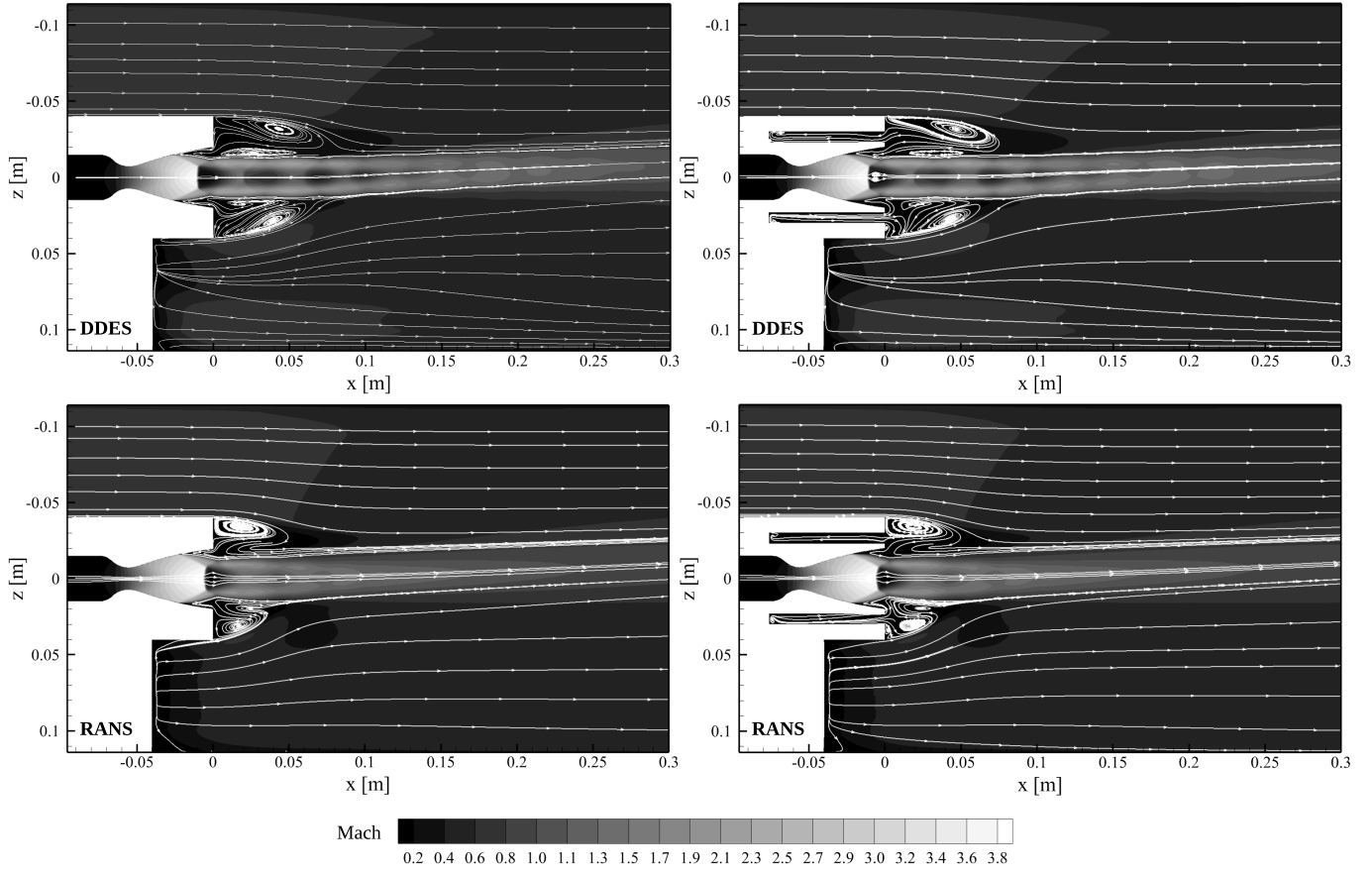


FIGURE 9: TIME-AVERAGED MACH NUMBER DISTRIBUTION FROM DDES (TOP) AND CORRESPONDING RANS COMPUTATION (BOTTOM)

pressure at different azimuthal angles, non-dimensionalized with the total pressure at the tunnel inlet $P_{0,t}$. Observing Fig. 10, it is evident that the cavity region has a negligible impact on the time-averaged base pressure distribution at the base. This is in line with previous two-dimensional and axisymmetric, RANS computations corresponding to similar NPR and flow conditions at the base, which demonstrated that there is no impact of the cavity on the mean base pressure at high over-expansion nozzle state [13]. This can be seen here again in Fig. 10, by looking at the results from the corresponding, three-dimensional RANS computations. Indeed, Reynolds-averaged solution yields almost identical results at the base for the cavity and baseline cases. The pressure levels obtained with RANS are slightly reduced, but are in close agreement and within 3% with respect to the DDES values. This decrease in pressure is what was previously identified in Fig. 9 through the time-averaged organization of the base flow, with the RANS producing a recirculation region closer to the base. Furthermore, one can notice the near-symmetrical behavior of the base flow in the xy plane, shown by the similar values of p_b at angles of 90° and 270° for both cases, as would be expected for an axially-symmetric geometry. In contrast, the strong asymmetry in the xz plane caused by the presence of the pylon is also evident (Fig. 10). Additionally, the trend captured from the RANS solution agrees with the DDES results. This, along with the relatively small differences in the mean pressure values, is a good indication towards using RANS at the concep-

tual design level for similar high-speed exhaust configurations, where high-fidelity and computationally-expensive calculations are usually prohibitive.

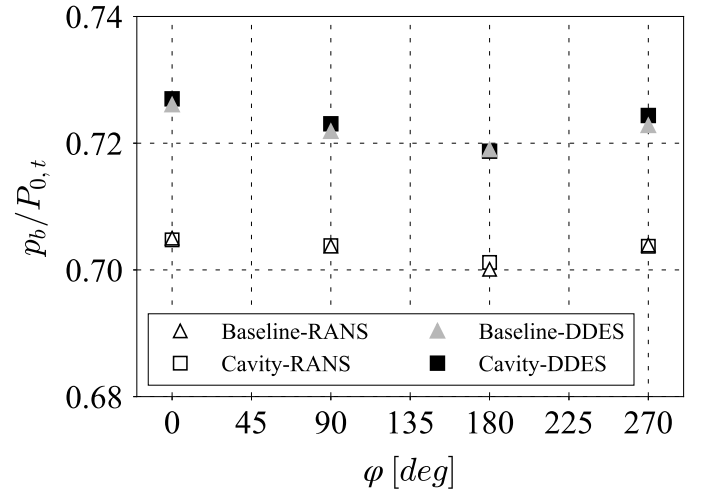


FIGURE 10: TIME-AVERAGED BASE PRESSURE AZIMUTHAL DISTRIBUTION

The preceding analysis shows that the cavity region has a negligible impact on the base flow in a time-average sense. Time-averaging is very useful from an engineering point of view, since

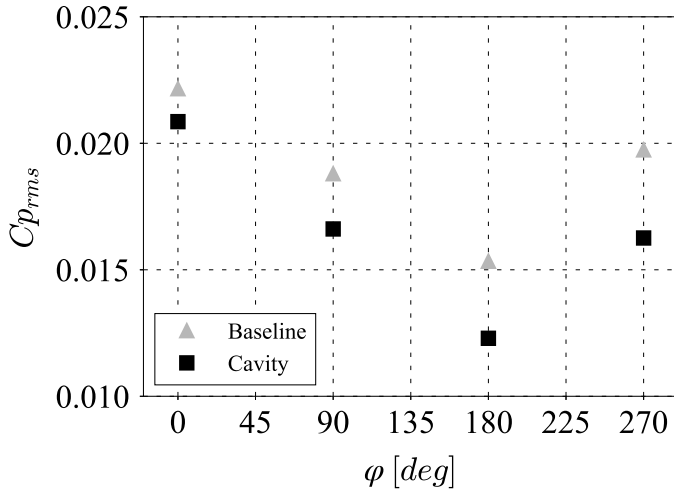


FIGURE 11: AZIMUTHAL DISTRIBUTION OF THE COEFFICIENT OF RMS BASE PRESSURE FLUCTUATIONS

it allows the mean values of interest to be obtained. However, it does not reveal the underlying unsteady dynamics which might exist and could be important. 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 [50], where p and $\langle p \rangle$ denote the instantaneous and time-averaged pressure values, respectively. Figure 11 depicts the coefficient of the rms value of the pressure fluctuation along the base wall $C_{p_{rms}}$, defined in Eq. (33):

$$C_{p_{rms}} = \frac{\sqrt{\langle p'^2 \rangle}}{\frac{1}{2} \rho_{\infty,0} U_{\infty,0}^2} \quad (33)$$

The level of base pressure fluctuations is similar with previous experimental studies on base flows of similar geometries and flow conditions [21, 68], and falls within the range of approximately 0.025-0.015, which can be taken as a good indication on the credibility of the numerical solution. Additionally, Fig. 11 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° 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, reducing the energy contained in the pressure fluctuations at the base, compared to the rigid wall in the baseline configuration. This effect of reduction in rms pressure values is in line with the experimental results of Tripathi et al. [34], conducted for a cylindrical, powered-on geometry with base cavity. However, we note that the two configurations are different (e.g., notably different cavity depth and boat-tailed base geometry among others). Additionally, it is noted that rms values carry higher uncertainty compared to the mean values and should be treated with caution. While the consistency in the reduced rms levels of the cavity case, as well as the direct nature of the comparison in this study, allows for confidence in the relative change of the results, experiments are still required to validate this effect.

3.4 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 power spectral density (PSD), denoted as $G_{p'}$, describes how the mean squared value of wall pressure fluctuations is distributed in the frequency domain [69], since:

$$\sigma^2 = \int_0^\infty G_{p'}(f) df \quad (34)$$

As suggested by Owen [70] 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 at the vicinity of the base at 0° angle. The corresponding, non-dimensional Strouhal number defined as $St_D = fD/U_{\infty,0}$ based on the diameter of the model can then be used, where f denotes the frequency. Consequently for a zero-mean variable (i.e., fluctuations) [21, 70, 71]:

$$C_{p_{rms}}^2 = \int_0^\infty G_{p'}(St_D) d(St_D) = \int_0^\infty St_D G_{p'}(St_D) d(\log St_D) \quad (35)$$

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 [70]. The PSD functions are computed using Welch's method [72]. The original time series is divided into overlapping segments using 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.

With these provisions, Fig. 12 depicts the spectra of the wall pressure fluctuations between the baseline and cavity cases, at different azimuthal locations, namely at 0°, 90°, 180° and 270°. The spectra corresponding to the baseline case, exhibits similar organization at all examined locations. In specific, a broad peak of the pressure fluctuations spectrum can be identified, centered around approximately $St_D=0.2$. This value of non-dimensional frequency has been identified in previous studies for axisymmetric bodies [18, 21] and BFSs[14, 21, 22, 27], as well as for more detailed launch vehicle configurations [67]. 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 non-dimensional frequencies of approximately $St_D=0.45$ and 0.81 . These are very close to the dominant peaks of non-dimensional frequency at $St_D=0.42$ and 0.82 , revealed by spectral analysis of the experimental base pressure fluctuations in [34] for conditions of $M_\infty=0.6$ and over-expanded jet for the sharp-base geometry which is similar to the baseline case of this study. These frequencies are associated with smaller-scale structures being convected downstream in the wake. The lower-frequency peak of $St_D=0.2$ was also identified. The good agreement with the experimental values allows for additional confidence in the validity of the computations. It is noted that the energy distribution in the frequency domain is ranked as level 4 (out of 6) in validation grade for unsteady simulation techniques

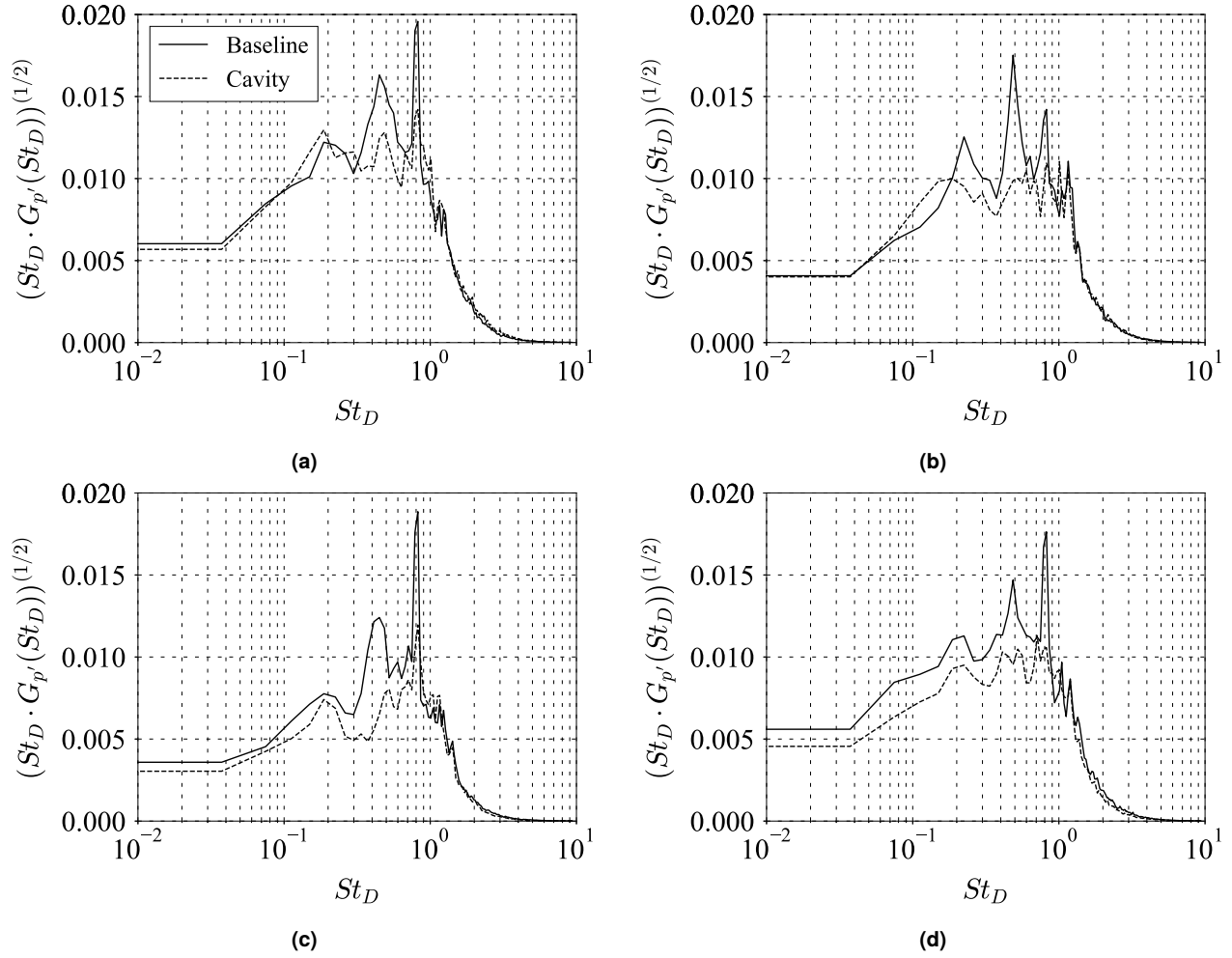


FIGURE 12: BASE PRESSURE SPECTRUM AT DIFFERENT AZIMUTHAL LOCATIONS; (a) $\phi=0^\circ$, (b) $\phi=90^\circ$, (c) $\phi=180^\circ$ and (d) $\phi=270^\circ$

as introduced by Sagaut and Deck [73]. Additionally, note that the resolution at very low frequencies of values below approximately $St_D=0.05$, is not very high. This is owing to the relatively short sampling duration due to increased CPU cost, and is a typical drawback of expensive computations in contrast to experiments which usually allow longer run times [73].

For the spectra corresponding to the cavity case, a similar low-frequency peak at $\sim St_D=0.2$ is observed, although with slightly smaller intensity at azimuthal angles of 90° and 270° . This again highlights the effects of the blockage. 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 with the baseline case. This is in line with the results shown in Fig. 11 for the rms values of pressure fluctuations at the base, and clearly demonstrates that the cavity has a notable impact on base flow unsteadiness, especially in higher frequencies. The effect of the cavity on the total associated energy in the pressure fluctuations spectra can be approximated through the numerical integration of the spectrum as described in Eq. (34) and can be seen in Fig. 13 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 total energy in the pressure fluctuations in all azimuthal locations. 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 alter the large-scale dynamics greatly. The cavity case exhibits total energy levels which are 15% (Fig. 13a) to 38% (Fig. 13c) lower with respect to the baseline case. The significant reduction at the 180° angle, indicates stronger effectiveness in reducing the intensity of pressure fluctuations caused by the wing-pylon. This could potentially be important for future, high-speed vehicles, usually featuring wingtip-mounted nacelles [30], and should be studied further in future studies.

3.5 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. 14. The time distribution of the side loads 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 in linear/log axes, since as described in Eq. (34), in this way the contribution of the side loads in the total energy is obtained for the corresponding frequency range [69]. One can notice that the

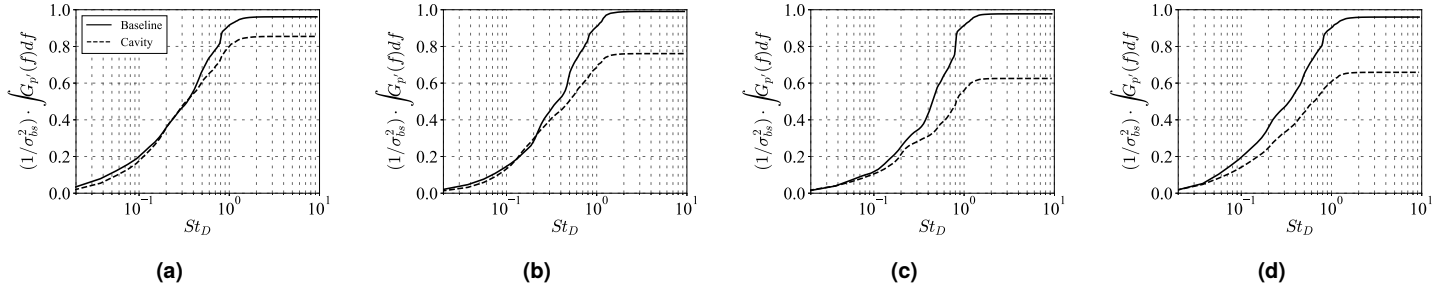


FIGURE 13: CUMULATIVE ENERGY OF FLUCTUATING PRESSURE SPECTRUM; (a) $\phi=0^\circ$, (b) $\phi=90^\circ$, (c) $\phi=180^\circ$ and (d) $\phi=270^\circ$

energy spectra of the side load in the y direction (perpendicular to the pylon plane), exhibits a broad peak centered at medium frequencies of around 600 Hz for both configurations. A 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 (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 lower frequency range, and increased levels at higher frequencies of ~ 1300 Hz. This suggests that the level of pressure fluctuations (i.e., the side loads) at higher frequencies, is dampened by the existence of the cavity, compared to the baseline case. This effect is more pronounced in the pylon-parallel plane (z direction) and is in line with previous observations in the base pressure spectra (see Fig. 13).

Figure 15 shows a polar plot of the side loads (i.e., $F_y(t)$ vs $F_z(t)$ component of the side loads) for both configurations, along with its $a=0.95$ confidence ellipse, which includes 95% of the observations. One can notice the random character of side loads fluctuations. Additionally, the confidence ellipse becomes a circle for both configurations (i.e., $\sigma_{F_z} = \sigma_{F_y}$), indicating the isotropic nature of the side loads over time. It is worth noting that, for the baseline case, the side loads envelope is not centered at zero, as would be the case for an axisymmetric configuration (since $\langle F_y \rangle = \langle F_z \rangle = 0$) [14]. In specific, because of the non-axisymmetric topology of the examined configuration, 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 xz plane. However, the existence of the cavity region shifts the side loads envelope back to zero-mean values, as it would appear for an axisymmetric geometry, suggesting that the impact of the cavity in the base flow unsteadiness is large enough to notably affect the time-averaged flow organization and the side loads of the nozzle. This means that the cavity, which would be essential for future, high-speed systems [13], could act as a passive flow control mechanism for side loads reduction, especially for wingtip-mounted nacelles. We note that the actual, mean side load reduction is an order of magnitude in the cavity case.

The behavior of the side loads as described previously can be further understood simply by looking at the probability density functions (PDFs) of F_y and F_z . This is depicted in Fig. 16. It

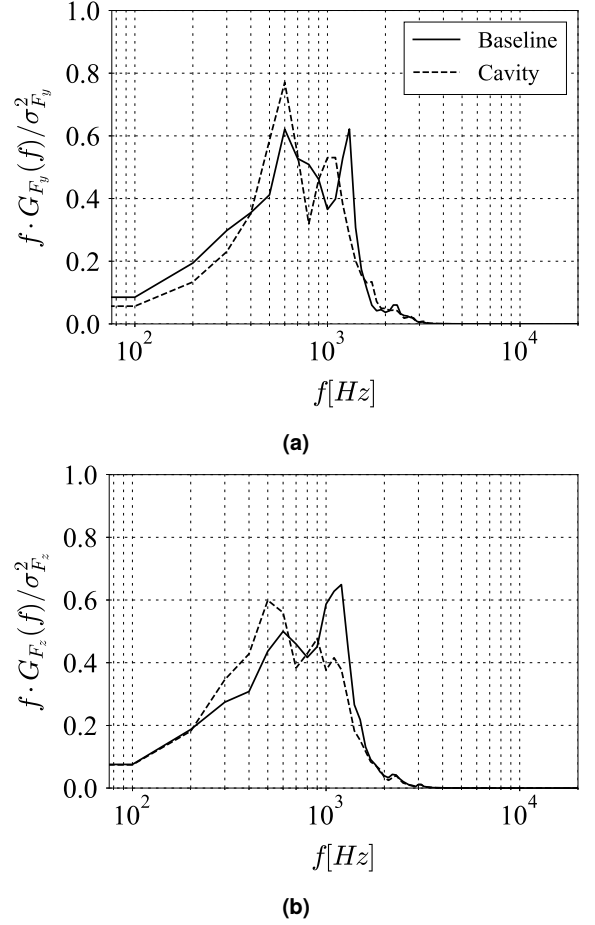


FIGURE 14: PRE-MULTIPLIED POWER SPECTRAL DENSITY OF THE SIDE LOADS COMPONENTS; (a) F_y , (b) F_z

is clearly evident that both components are normally distributed for both configurations. However, the baseline case exhibits non-zero mean values, and as a result the PDFs appear shifted with respect to the theoretical Gaussian distribution, which is also shown for convenience. This shift is increased in the F_z side load. However, the addition of the cavity region, shifts the PDFs back to zero-mean which now fit very well the theoretical Gaussian distribution. As a result the probability distribution of the magnitude of side loads ($|F| = (F_y^2 + F_z^2)^{1/2}$), follows a Rayleigh distribution, not shown here for brevity, as found by Dumnov [74].

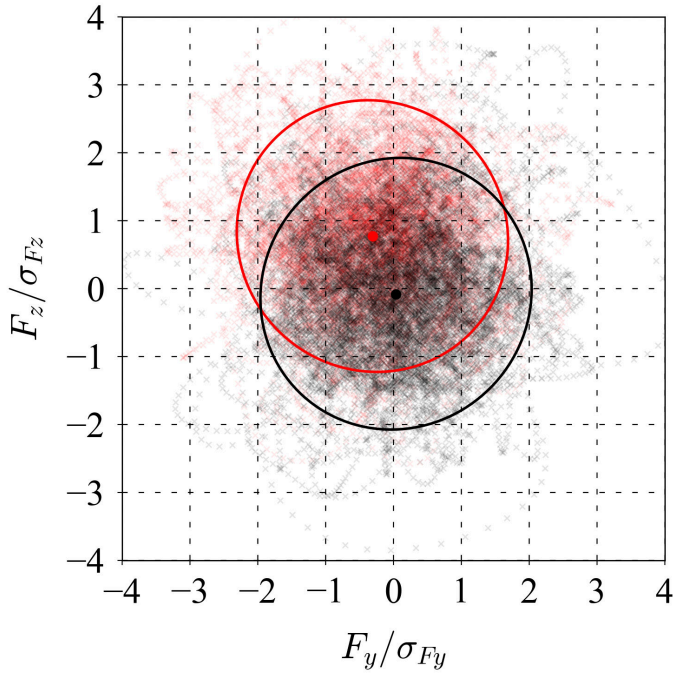


FIGURE 15: POLAR PLOT OF NOZZLE SIDE LOADS (SYMBOLS) AND 95% CONFIDENCE ELLIPSE (LINES). THE DOTS CORRESPOND TO THE MEAN VALUES. (RED: BASELINE, BLACK: CAVITY)

4. CONCLUSIONS AND FUTURE WORK

A numerical investigation on the impact of a base cavity on the base flow unsteadiness for a sub-scale, high-speed exhaust system is presented in this work. The geometry is considered representative of future, advanced propulsion concepts. It features a high-speed, over-expanded TIC nozzle embedded at the base. The model is mounted on the test section of a wind tunnel to facilitate ongoing experiments. The DDES turbulence modelling approach is employed to reduce the computational cost compared to LES for wall-bounded flows. A direct comparison to an identical, non-cavity geometry is carried out, allowing for the cavity impact to be exposed. The two investigated systems are referred to as baseline and cavity cases. The credibility of the numerical simulation is evaluated and found to be in good agreement with available experimental studies on geometries similar to the baseline case, in terms of non-dimensional frequency content.

It is found that the cavity region has a negligible effect on the time-averaged base pressure distribution compared to the baseline case for the conditions examined which correspond to over-expanded nozzle operation. However, the level of the base pressure fluctuations coefficient is notably 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 approximately $\sim St_D=0.2$. The cumulative energy of base pressure fluctuations is reduced by 15% to 38%, depending on the azimuthal location, compared to the baseline case. This indicates that the cavity region, acts in a damping-like manner for the intensity of unsteadiness at the

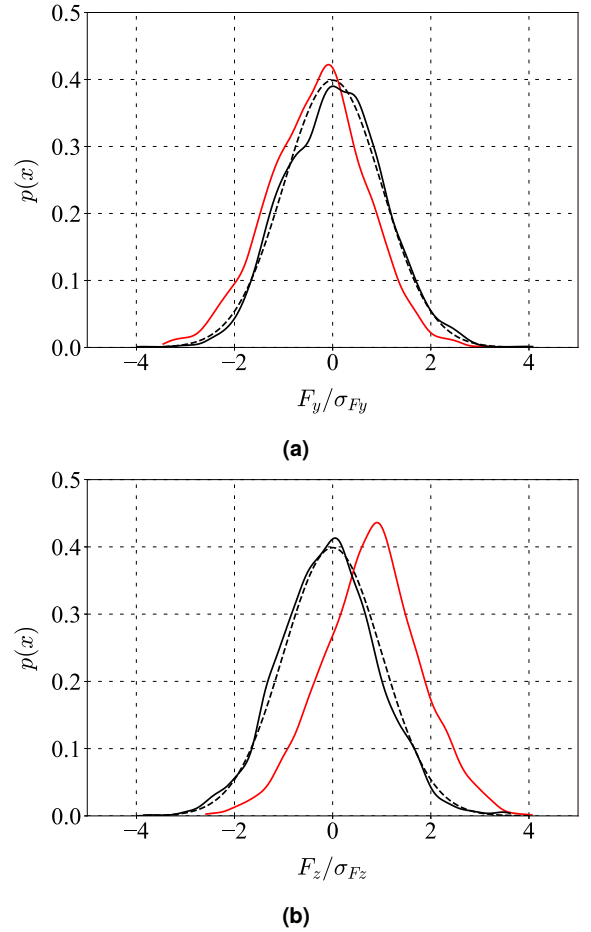


FIGURE 16: SIDE LOADS PROBABILITY DENSITY FUNCTION; (a) F_y , (b) F_z . RED LINE: BASELINE, BLACK LINE: CAVITY. DASHED LINE DENOTES THE THEORETICAL GAUSSIAN DISTRIBUTION

base flow.

Additionally, results show that the cavity region has a notable effect on the generated side loads of the TIC nozzle 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 existence of the cavity shifts the side loads behavior back to an axisymmetric-like geometry, and their absolute value is reduced by an order of magnitude compared to the baseline case. This means that the cavity region, which will be essential for future, high-speed exhaust systems, could also act as a passive flow control mechanism for side loads reduction, which are critical at over-expanded nozzle conditions.

The ongoing experimental campaign will facilitate a validation study for the computational results and thus, help to identify the strengths and weaknesses of the employed numerical approach for future, high-speed exhausts involving subtle details such as base cavity effects. Currently, the experimental rig is limited to lower NPR s owing to practical limitations which were unknown during the computational setup and preparation of this manuscript, and which may lead to perform additional computations at lower NPR s. These will be the focus of future work.

ACKNOWLEDGMENTS

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 for granting permission to publish this research. The first author would like to thank Dr Andrea Spinelli for fruitful discussion on multi-dimensional random variable statistical analyses.

REFERENCES

- [1] McClinton, C. R. "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 (2007): pp. 1–32.
- [2] D., Sziroczak and H., Smith. "A review of design issues specific to hypersonic flight vehicles." *Progress in Aerospace Sciences* Vol. 84 (2016): pp. 1–28. DOI [10.1016/j.paerosci.2016.04.001](https://doi.org/10.1016/j.paerosci.2016.04.001).
- [3] Ingenito, A. *Subsonic Combustion Ramjet Design*. Springer Nature, Switzerland (2021). DOI [10.1007/978-3-030-66881-5](https://doi.org/10.1007/978-3-030-66881-5).
- [4] Murthy, S.N.B. and Curran, E.T. *Scramjet Propulsion*. AIAA (2001). DOI [10.2514/4.866609](https://doi.org/10.2514/4.866609).
- [5] T., Bogar, E., Eiswirth, L., Couch, J., Hunt and C., McClinton. "Conceptual design of a Mach 10, global reach reconnaissance aircraft." *32nd Joint Propulsion Conference and Exhibit*. DOI [10.2514/6.1996-2894](https://doi.org/10.2514/6.1996-2894).
- [6] R., Daines and C., Segal. "Combined Rocket and Airbreathing Propulsion Systems for Space-Launch Applications." *Journal of Propulsion and Power* Vol. 14 No. 5 (2012): pp. 605–612. DOI [10.2514/2.5352](https://doi.org/10.2514/2.5352).
- [7] R., Varvill and A., Bond. "A comparison of propulsion concepts for SSTO reusable launchers." *Journal of the British Interplanetary Society* Vol. 56 (2003): pp. 108–117.
- [8] Dai, J. and Zuo, Q. "Key technologies for thermodynamic cycle of precooled engines: A review." *Acta Astronautica* Vol. 177 (2020): pp. 299–312. DOI [10.1016/j.actaastro.2020.07.039](https://doi.org/10.1016/j.actaastro.2020.07.039).
- [9] J., Zhang, Z., Wang and Q., Li. "Thermodynamic efficiency analysis and cycle optimization of deeply precooled combined cycle engine in the air-breathing mode." *Acta Astronautica* Vol. 138 (2017): pp. 394–406. DOI [10.1016/j.actaastro.2017.06.011](https://doi.org/10.1016/j.actaastro.2017.06.011).
- [10] F., Villace V. and G., Paniagua. "Simulation of a Combined Cycle for High Speed Propulsion." *48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*. AIAA 2010-1125. Orlando, Florida, 4–7 January, 2010.
- [11] S., Tsentis, V., Gkoutzamanis, A., Gaitanis and I., Kalfas. "Multi-platform App-embedded Model for Hybrid Air-Breathing Rocket-Cycle Engine in Hypersonic Atmospheric Ascent." *The Aeronautical Journal* Vol. 125 (1291) (2021): pp. 1631–1653. DOI [10.1017/aer.2021.3](https://doi.org/10.1017/aer.2021.3).
- [12] F., Jivraj, R., Varvill, A., Bond and G., Paniagua. "The Scimitar Pre-cooled Mach 5 Engine." *2nd European Conference for Aerospace Sciences*. EUCASS. Brussels, 2007.
- [13] Tsentis, S., Goulos, I., Prince, S., Pachidis, V. and Zmijanovic, V. "Propulsion Aerodynamics for a Novel High-Speed Exhaust System." *Journal of Engineering for Gas Turbines and Power* Vol. 145 No. 12 (2023): p. 121011. DOI [10.1115/1.4063416](https://doi.org/10.1115/1.4063416).
- [14] Deck, Sébastien and Thorigny, Pascal. "Unsteadiness of an axisymmetric separating-reattaching flow: Numerical investigation." *Physics of Fluids* Vol. 19 No. 6 (2007): p. 065103. DOI [10.1063/1.2734996](https://doi.org/10.1063/1.2734996).
- [15] Hammond, Walter E. *Design Methodologies for Space Transportation Systems*. AIAA, Virginia (2001). DOI [10.2514/4.861734](https://doi.org/10.2514/4.861734).
- [16] S., Scharnowski and C., Kähler. "Investigation of the base flow of a generic space launcher with dual-bell nozzle." *CEAS Space Journal* Vol. 13 (2021): pp. 197–216. DOI [10.1007/s12567-020-00333-5](https://doi.org/10.1007/s12567-020-00333-5).
- [17] T.H., Tran, H.Q., Dinh and et al., Chu H.Q. "Effect of boattail angle on near-wake flow and drag of axisymmetric models: a numerical approach." *J Mech Sci Technol* Vol. 35 (2021): pp. 563–573. DOI [10.1007/s12206-021-0115-1](https://doi.org/10.1007/s12206-021-0115-1).
- [18] A., Mariotti and G., Buresti. "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* Vol. 54 No. 1612 (2013). DOI [10.1007/s00348-013-1612-5](https://doi.org/10.1007/s00348-013-1612-5).
- [19] A., Mariotti, G., Buresti and V., Salvetti M. "Connection between base drag, separating boundary layer characteristics and wake mean recirculation length of an axisymmetric blunt-based body." *Journal of Fluids and Structures* Vol. 55 (2015): pp. 191–203. DOI [10.1016/j.jfluidstructs.2015.02.012](https://doi.org/10.1016/j.jfluidstructs.2015.02.012).
- [20] Paciorri, R., Sabetta, F., Valenza, F., Fauci, R., Passaro, A. and Baccarella, D. "Base-Pressure Experimental Investigation on a Space Launcher in Subsonic Regime." *Journal of Spacecraft and Rockets* Vol. 50 No. 3 (2013): pp. 572–578. DOI [10.2514/1.A32283](https://doi.org/10.2514/1.A32283).
- [21] Deprés, D., Reijasse, P. and Dussauge, J. P. "Analysis of Unsteadiness in Afterbody Transonic Flows." *AIAA Journal* Vol. 42 No. 12 (2004): pp. 2541–2550. DOI [10.2514/1.7000](https://doi.org/10.2514/1.7000).
- [22] P., Weiss E., S., Deck, C., Robinet J. and P., Sagaut. "On the dynamics of axisymmetric turbulent separating/reattaching flows." *Physics of Fluids* Vol. 21 No. 075103 (2009). DOI [10.1063/1.3177352](https://doi.org/10.1063/1.3177352).
- [23] Deck, S. "Zonal-Detached-Eddy Simulation of the Flow Around a High-Lift Configuration." *AIAA Journal* Vol. 43 No. 11 (2005): pp. 2372–2384. DOI [10.2514/1.16810](https://doi.org/10.2514/1.16810).
- [24] D., Saile, V., Kühl and A., Gülhan. "On subsonic near-wake flows of a space launcher configuration with various base geometries." *Experiments in Fluids* Vol. 62 No. 122 (2021). DOI [10.1007/s00348-021-03149-z](https://doi.org/10.1007/s00348-021-03149-z).
- [25] D., Saile, V., Kühl and A., Gülhan. "On the subsonic near-wake of a space launcher configuration with exhaust jet." *Experiments in Fluids* Vol. 60 No. 165 (2019). DOI [10.1007/s00348-019-2801-7](https://doi.org/10.1007/s00348-019-2801-7).
- [26] V., Statnikov, T., Sayadi, M., Meinke, P., Schmid and W., Schröder. "Analysis of pressure perturbation sources on a generic space launcher afterbody in supersonic flow using zonal turbulence modeling and dynamic mode decomposition." *Physics of Fluids* Vol. 27 No. 016103 (2015). DOI [10.1063/1.4906219](https://doi.org/10.1063/1.4906219).
- [27] Meliga, P. and Reijasse, P. "Unsteady Transonic Flow Behind An Axisymmetric Afterbody Equipped With Two Boosters." *25th AIAA Applied Aerodynamics Conference*. 2012. DOI [10.2514/6.2007-4564](https://doi.org/10.2514/6.2007-4564).
- [28] Pain, R., Weiss, P.-E. and Deck, S. "Zonal Detached Eddy Simulation of the Flow Around a Simplified Launcher Afterbody." *AIAA Journal* Vol. 52 No. 9 (2014): pp. 1967–1979. DOI [10.2514/1.J052743](https://doi.org/10.2514/1.J052743).
- [29] Schwane, Richard. "Numerical Prediction and Experimental Validation of Unsteady Loads on ARIANE5 and VEGA." *Journal of Spacecraft and Rockets* Vol. 52 No. 1 (2015): pp. 54–62. DOI [10.2514/1.A32793](https://doi.org/10.2514/1.A32793).
- [30] Mehta, U. B., Aftosmis, M. J., Bowles, J. V. and Pandya, S. A. "Skylon Aerodynamics and SABRE Plumes." *20th AIAA International Space Planes and Hypersonic Systems and Technologies Conference*. DOI [10.2514/6.2015-3605](https://doi.org/10.2514/6.2015-3605).
- [31] R., Viswanath P. "Flow Management Techniques for Base and Afterbody Drag Reduction." *Progress in Aerospace Sciences* Vol. 32 (1996): pp. 79–129.
- [32] M., Tanner. "Reduction of Base Drag." *Progress in Aerospace Sciences* Vol. 16 No. 4 (1975): pp. 369–384.
- [33] T., Morel. "Effect of Base Cavities on the Aerodynamic Drag of an Axisymmetric Cylinder." *Aeronautical Quarterly* Vol. 30 (1990): pp. 400–412. DOI [10.1017/s0001925900008611](https://doi.org/10.1017/s0001925900008611).
- [34] A., Tripathi, C., Manisankar and B., Verma S. "Control of base pressure for a boat-tailed axisymmetric afterbody via base geometry modifications." *Aerospace Science and Technology* Vol. 45 (2015): pp. 284–293. DOI [10.1016/j.ast.2015.05.021](https://doi.org/10.1016/j.ast.2015.05.021).
- [35] F., Nasuti and M., Onofri. "Shock structure in separated nozzle flows." *Shock Waves* Vol. 19 (2009): pp. 229–237. DOI [10.1007/s00193-008-0173-7](https://doi.org/10.1007/s00193-008-0173-7).
- [36] Östlund, J., Damgaard, T. and Frey, M. "Side-Load Phenomena in Highly Overexpanded Rocket Nozzles." *Journal of Propulsion and Power* Vol. 20 No. 4 (2004): pp. 695–704. DOI [10.2514/1.3059](https://doi.org/10.2514/1.3059).
- [37] Frey, M. and Hagemann, G. "Restricted Shock Separation in Rocket Nozzles." *Journal of Propulsion and Power* Vol. 16 No. 3 (2000): pp. 478–484. DOI [10.2514/2.5593](https://doi.org/10.2514/2.5593).
- [38] T., Shimizu, H., Miyajima and M., Kodera. "Numerical Study of Restricted Shock Separation in a Compressed Truncated Perfect Nozzle." *AIAA Journal* Vol. 44 No. 3 (2006): pp. 576–584. DOI [10.2514/1.14288](https://doi.org/10.2514/1.14288).
- [39] Zucrow, Maurice J. and Hoffman, Joe D. *Gas Dynamics*. John Wiley and Sons, New York (1976).
- [40] Hagemann, G., Frey, M. and Koschel, W. "Appearance of Restricted Shock Separation in Rocket Nozzles." *Journal of Propulsion and Power* Vol. 18 No. 3 (2002): pp. 577–584. DOI [10.2514/2.5971](https://doi.org/10.2514/2.5971).

- [41] Deck, S. and Nguyen, A. T. “Unsteady Side Loads in a Thrust-Optimized Contour Nozzle at Hysteresis Regime.” *AIAA Journal* Vol. 42 No. 9 (2004): pp. 1878–1888. DOI [10.2514/1.2425](https://doi.org/10.2514/1.2425).
- [42] Genin, C. and Stark, R. H. “Side Loads in Subscale Dual Bell Nozzles.” *Journal of Propulsion and Power* Vol. 27 No. 4 (2011): pp. 828–837. DOI [10.2514/1.B34170](https://doi.org/10.2514/1.B34170).
- [43] Cimini, M., Martelli, E. and Bernardini, M. “Numerical Analysis of Side-loads Reduction in a Sub-scale Dual-bell Rocket Nozzle.” *Flow, Turbulence and Combustion* Vol. 107 (2021): p. 551–574. DOI <https://doi.org/10.1007/s10494-021-00243-4>.
- [44] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K. and Travin, A. K. “A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities.” *Theoretical and Computational Fluid Dynamics* Vol. 20 (2006): pp. 181–195. DOI [10.1007/s00162-006-0015-0](https://doi.org/10.1007/s00162-006-0015-0).
- [45] Sagaut, P., Deck, S. and Terracol, M. *Multiscale and Multiresolution Approaches in Turbulence*, 2nd ed. Imperial College Press (2013). DOI [10.1142/p878](https://doi.org/10.1142/p878).
- [46] Tsentis, S., Goulos, I., Prince, S., Pachidis, V., Zmijanovic, V. and Saavedra, J. “Wind Tunnel Installation Effects on the Base Flow for a High-Speed Exhaust System.” *AIAA Scitech 2024 Forum*. Orlando, Florida, 8–13 January 2024. DOI [10.2514/6.2024-1774](https://doi.org/10.2514/6.2024-1774).
- [47] R., Sauer. “General Characteristics of the Flow through Nozzles at Near Critical Speeds.” Memorandum No. 1147. NACA, Washington, DC. 1947.
- [48] MIDAP, Study Group. “Guide to In-Flight Thrust Measurement of Turbojets and Fan Engines.” AGARDograph No. 237. Advisory Group for Aerospace Research and Development, 7 Rue Ancelle 92200 Neuilly, Sur Seine, France. 1979.
- [49] W., Sutherland. “The viscosity of gases and molecular forces.” *Philos. Mag.* Vol. 36 (1893): pp. 507–531.
- [50] D., Wilcox. *Turbulence Modeling for CFD*. DCW Industries, La Canada, California (2006).
- [51] Menter, F. R. “Two-equation eddy-viscosity turbulence models for engineering applications.” *AIAA Journal* Vol. 32 No. 8 (1994): pp. 1598–1605. DOI [10.2514/3.12149](https://doi.org/10.2514/3.12149).
- [52] F., Menter, M., Kuntz and R., Langtry. “Ten Years of Industrial Experience with the SST Turbulence Model.” *Heat and Mass Transfer* 4. Begell House Inc. (2003): pp. 625–632.
- [53] Gritskevich, M. S., Garbaruk, A. V., Schütze, J. and Menter, F. R. “Development of DDES and IDDES Formulations for the $k-\omega$ Shear Stress Transport Model.” *Flow Turbulence Combust* Vol. 88 (2012): pp. 431–449. DOI [10.1007/s10494-011-9378-4](https://doi.org/10.1007/s10494-011-9378-4).
- [54] Spalart, P.R., Jou, W. H., Strelets, M. and Allmaras, S. R. “Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach.” *In Advances in DNS/LES*: pp. 137–147. 1997.
- [55] Spalart, P. R. “Detached-Eddy Simulation.” *Annual Review of Fluid Mechanics* Vol. 41 No. 1 (2009): pp. 181–202. DOI [10.1146/annurev.fluid.010908.165130](https://doi.org/10.1146/annurev.fluid.010908.165130).
- [56] Menter, F. R. and Kuntz, M. “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 (2004): pp. 339–352. DOI [10.1007/978-3-540-44419-0_30](https://doi.org/10.1007/978-3-540-44419-0_30).
- [57] Ansys Inc., 275 Technology Drive, Canonsburg, PA 15317. *Ansys Fluent Theory Guide*, Release 15.0 (2013).
- [58] Ansys Inc., 275 Technology Drive, Canonsburg, PA 15317. *Best Practice: Scale-Resolving Simulations in Ansys CFD*, Technical paper 2.00 (2015).
- [59] Ansys Inc., 275 Technology Drive, Canonsburg, PA 15317. *Ansys Fluent User’s Guide*, Release 2021 R2 (2021).
- [60] Liou, M. S. “A Sequel to AUSM: AUSM+.” *Journal of Computational Physics* Vol. 129 (1996): pp. 364–382. DOI [10.1006/jcph.1996.0256](https://doi.org/10.1006/jcph.1996.0256).
- [61] Ansys Inc., 275 Technology Drive, Canonsburg, PA 15317. *Ansys FLUENT Meshing User Guide*, Release 14.5 (2012).
- [62] Spalart, P.R. “Young person’s guide to detached-eddy simulation grids.” Technical Report NASA CR-2001-211032. 2001.
- [63] B., Celik I., U., Ghia, J., Roache P., J., Freitas C., H., Coleman and E., Raad P. “Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications.” *Journal of Fluids Engineering* Vol. 130 No. 7 (2008). DOI [10.1115/1.2960953](https://doi.org/10.1115/1.2960953).
- [64] Richardson, L. F. and Gaunt, J. A. “The deferred approach to the limit.” *Philosophical Transactions of the Royal Society of London. Series A* Vol. 226 No. 636-646 (1927): pp. 299–361. DOI [10.1098/rsta.1927.0008](https://doi.org/10.1098/rsta.1927.0008).
- [65] Martelli, E., Saccoccio, L., Ciottoli, P. P., Tinney, C. E., Baars, W. J. and Bernardini, M. “Flow dynamics and wall-pressure signatures in a high-Reynolds-number overexpanded nozzle with free shock separation.” *Journal of Fluid Mechanics* Vol. 895 No. 1612 (2020): p. A29. DOI [10.1017/jfm.2020.280](https://doi.org/10.1017/jfm.2020.280).
- [66] Dubief, Y. and Delcayre, F. “On coherent-vortex identification in turbulence.” *Journal of Turbulence* Vol. 1 (2000): p. N11. DOI [10.1088/1468-5248/1/1/011](https://doi.org/10.1088/1468-5248/1/1/011).
- [67] Lüdeke, Heinrich, Mulot, Jean Daniel and Hannemann, Klaus. “Launch Vehicle Base Flow Analysis Using Improved Delayed Detached-Eddy Simulation.” *AIAA Journal* Vol. 53 No. 9 (2015): pp. 2454–2471. DOI [10.2514/1.J053297](https://doi.org/10.2514/1.J053297).
- [68] Eldred, K. McK. “Base Pressure Fluctuations.” *The Journal of the Acoustical Society of America* Vol. 33 No. 1 (2005): pp. 59–63. DOI [10.1121/1.1908404](https://doi.org/10.1121/1.1908404).
- [69] Shin, K. and Hammond, J. *Fundamentals of signal processing for sound and vibration engineers*. Wiley, Chchester, England (2008).
- [70] Owen, T.B. *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 (1958).
- [71] Mabey, D. G. “Analysis and Correlation of Data on Pressure Fluctuations in Separated Flow.” *Journal of Aircraft* Vol. 9 No. 9 (1972): pp. 642–645. DOI [10.2514/3.59053](https://doi.org/10.2514/3.59053).
- [72] Welch, P. “The use of fast Fourier transform for the estimation of power spectra: A method based on time averaging over short, modified periodograms.” *IEEE Transactions on Audio and Electroacoustics* Vol. 15 No. 2 (1967): pp. 70–73. DOI [10.1109/TAU.1967.1161901](https://doi.org/10.1109/TAU.1967.1161901).
- [73] Sagaut, P. and Deck, S. “Large eddy simulation for aerodynamics: status and perspectives.” *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences* Vol. 367 No. 1899 (2009): pp. 2849–2860. DOI [10.1098/rsta.2008.0269](https://doi.org/10.1098/rsta.2008.0269).
- [74] Dumnov, G. *Unsteady side-loads acting on the nozzle with developed separation zone*: DOI [10.2514/6.1996-3220](https://doi.org/10.2514/6.1996-3220).



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

2024-06-24

American Society of Mechanical Engineers (ASME)

<https://doi.org/10.1115/gt2024-126486>

Tsentis S, Goulos I, Prince S, et al., (2024) Cavity impact on the base flow unsteadiness for a high-speed exhaust system. In: ASME Turbo Expo 2024: Turbomachinery Technical Conference and Exposition, 24-28 June 2024, London, UK. Proceedings Volume 1: Aircraft Engine.