



Experimental and computational assessment of powered-on base flows in a transonic wind tunnel under large blockage conditions

Spyros Tsentis¹ · Marco Debiasi² · Ioannis Goulos¹ · Simon Prince¹ · Vladeta Zmijanovic³

Received: 13 March 2026 / Revised: 11 June 2026 / Accepted: 30 June 2026
© The Author(s) 2026

Abstract

Future high-speed propulsion concepts hold great potential for routine space access and sustainable high-speed flight. However, their development still requires extensive research, particularly in understanding the complex base flow aerodynamics during transonic flight, thus requiring detailed experimental testing. Transonic testing and numerical simulations of tunnel-installed, powered-on models still present many challenges and limitations. In this work, a combined experimental and numerical study is carried out, in an attempt to address some of these challenges. The aim is to identify effective approaches based on Reynolds-averaged Navier–Stokes simulations and to assess their accuracy in predicting the mean base flow for a high-speed, powered-on model installed in a transonic tunnel under large blockage conditions. The motivation stems from the high blockage ratios typically encountered in small-scale transonic facilities when testing powered-on models. This environment is challenging to model computationally. Important details related to computational domain, boundary conditions and proper matching of the experimental flow state are discussed. The impact of the blockage and model support on the base flow inside the tunnel, as well as its similarity relative to free-air conditions, are also highlighted. The results show that the proposed modelling approach can capture the flow field with a maximum error in the base pressure which is below 1.6% relative to the experimental measurements. Based on the findings, suitable practices are provided for the combined experimental and numerical analyses of high-speed, powered-on models.

Keywords RANS · Transonic wind tunnel · Blockage · Pylon interference · High-speed jet · Base flow

1 Introduction

The development of efficient high-speed propulsion systems has been a long-standing challenge despite significant efforts. Conventional air-breathing systems, such as turbojets, have well-known limitations in terms of their operating Mach range [1]. Ramjets and scramjets can address the limitations of gas turbines, but they require an alternate means to propel them to their takeover speeds. Additionally, they face considerable challenges in other areas, such as in propulsion-airframe integration for example [2]. The requirement of broad Mach range operation has led to the development of more advanced concepts, which combine the strengths of different propulsion architectures across different flight regimes, such as the turbine-based combined cycles or TBCC [2, 3], among others.

Space access on the contrary, relies on rocket-based systems which are approaching their efficiency and performance limits [4], with high costs per kilogram of payload delivered to orbit. Although some great strides have been

✉ Spyros Tsentis
spyros.tsentis@cranfield.ac.uk

Marco Debiasi
marco.debiasi@cranfield.ac.uk

Ioannis Goulos
i.goulos@cranfield.ac.uk

Simon Prince
simon.prince@cranfield.ac.uk

Vladeta Zmijanovic
vladeta.zmijanovic@alumni.polytechnique.org

¹ Centre for Aeronautics, Propulsion and Power, Cranfield University, Bedfordshire MK43 0AL, UK

² Centre for Defence Engineering, Cranfield University, Oxfordshire SN6 8LA, UK

³ Reaction Engines Ltd., Oxford OX14 3DB, UK

made in reusable rocket technology through retro-propulsive vertical landing [5], achieving routine space access continues to be an elusive goal. A concept that has received attention and could provide a step-change technology is based on pre-cooled air-breathing combined cycles [6]. Figure 1 illustrates a simplified schematic of a typical pre-cooled, air-breathing architecture. Although the thermodynamic efficiency benefits of such concepts are well-established [7, 8], their aero-propulsive behavior is not yet well understood, requiring extensive experimental testing.

The base flow is related to undesirable effects such as increased base drag as well as dynamic loads which can put the mission at risk. Such effects are particularly important during transonic flow. Indeed, as reported in [9], the most critical phase in terms of unsteady loads during the ascent of the Ariane 5 launcher occurs at high-subsonic to transonic speeds. Understanding and accurately predicting the base flow aerodynamics of high-speed systems is crucial. This was highlighted during the maiden flight of Ariane 5, in 1996, where severe pressure fluctuations were recorded on the main-stage nozzle Vulcain 2, during the first 37 s of the launch [9]. Additionally, after the failure of the Ariane 5 Evolution Cryotechnique type A (ECA) first flight, the investigation board identified the "non-exhaustive definition of the loads to which the Vulcain 2 engine is subjected during flight" as one of the causes [10]. An extensive campaign was initiated as a result, with the aim to improve the understanding of the complex flow interactions at the base of launchers, primarily at transonic conditions [9].

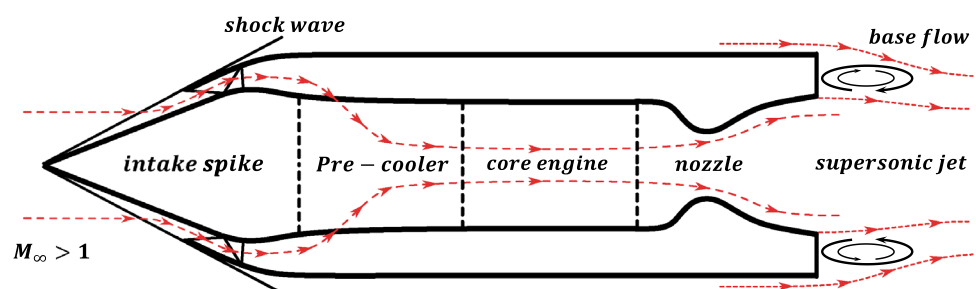
The base flow features are strongly related to the base geometry of the examined system. Therefore, prior knowledge on axisymmetric bodies and backward facing steps (BFS), obtained during extensive research into conventional space launchers, cannot be directly applied to new geometries. This is true even for a simple axisymmetric body after introducing side boosters [11]. Consequently, separate attention must be given to future, high-speed exhaust systems to properly characterize the underlying flow physics. Wind tunnel data are currently limited for such powered-on systems but will be essential for their technological advancement.

Traditionally, experimental tests aim to obtain data that can characterize the model under equivalent, free-air

conditions. This often requires the application of correction methodologies on the obtained dataset to account for blockage effects and model support interference. Such corrections can be neglected at low speeds for simple bodies under blockage ratios (BRs) below 1% according to the Engineering Sciences Data Unit (ESDU), but must be used at higher values above 5% [12]. Several studies have examined the influence of blockage ratio on the aerodynamic characteristics across a range of model configurations. These include axisymmetric and square-section bluff bodies [13, 14], rectangular prisms [15] and wind tunnel models for the automotive industry [16], to name a few. These studies demonstrate that the influence of blockage ratio is case-dependent, with some reporting noticeable effects at BR values of 5% [13], while others reported limited sensitivity up to BRs of 10% [14]. In general, there is no universally-acceptable blockage ratio limit. This typically depends on the model geometry, the wind tunnel facility, the operating conditions and the aim of the experiments, among others. There are numerous correction techniques to account for blockage for different wind tunnel types. The reader is referred to [17, 18] for a detailed overview of such methods as reported by the Advisory Group for Aerospace Research and Development (AGARD).

Although essential, application of corrections becomes more challenging as the geometry of the model gets more complex. For powered-on models, additional challenges arise due to the interaction of the propulsion system with the surrounding flow, including wake blockage as well as mass injection and entrainment effects for high-speed jets. Previous studies have developed dedicated correction methodologies for propeller-powered configurations, including blockage corrections for wind turbine tests [19] as well as support and wall interference corrections for propeller-powered aircraft models in low-speed wind tunnels [20]. For jet-powered configurations, isolated jet experiments in low-speed tunnels have been used to develop wall-signature correction methods to account for jet-induced effects [21]. Furthermore, prior experimental studies have highlighted the importance of flow entrainment for high-energy jets and suggested the use of wall-signature methods to account for the associated sink effect of the entrainment process [18].

Fig. 1 Simplified schematic of a typical pre-cooled air-breathing propulsion architecture at supersonic conditions and illustration of the base flow topology



In some cases, however, the primary objective of an experimental campaign is not to obtain free-air equivalent data, but rather to generate datasets for the validation and tuning of computational techniques. In such cases, the similarity of the measured flow with respect to free-air conditions is irrelevant. The domain used in the computational fluid dynamics (CFD) analysis should ideally consider the entire experimental rig, including the test section walls and model support. Although this approach adds considerable computational effort, particularly during meshing, it allows the wind tunnel limitations, such as blockage and model mount, to be considered as part of the examined system instead of interference effects. This allows any discrepancies in the CFD to be attributed to deficiencies of the computational model or experimental measurements rather than incomplete geometry modelling.

The use of CFD to simulate the entire experimental rig has been increasingly adopted. For example, Lessard [22] carried out Reynolds-averaged Navier–Stokes (RANS) calculations for the flow around the Technology Concept Airplane (TCA) wing-body high-lift model, installed in NASA Langley 14 x 22 ft Subsonic Tunnel at Mach 0.24. The simulations considered three configurations: the model in free-air conditions, the model with post support, and the model with both post support and tunnel walls. The study compared the results against experimental data and concluded that the support must be included in the modelling to capture the local flow physics properly. However, when overall performance metrics are of interest, such as forces and moments, free-air modelling seemed to be sufficient. It is noted that this might not be the case under large blockage conditions, requiring either the application of appropriate corrections or explicit modelling of the test section walls. Wubben and Maseland [23] used RANS-based CFD to investigate the adequacy of correction methods for support and walls interference in the High Speed Tunnel (HST) of the German-Dutch Wind Tunnels (DNW). In a similar spirit, Mavriplis et al. [24] recently carried out Reynolds-averaged Navier–Stokes calculations to assess the effects of test section walls and model mounting device on a double-delta wing in a low speed wind tunnel. The effort was aimed at improving current wind tunnel corrections for configurations that are representative of long-range supersonic geometries.

Most of the available studies in the literature that have used CFD to simulate the model installed in the tunnel environment, have considered solid-body, lifting models, without powered-on effects. While prior experimental tests of high-speed, propulsive base flows in closed tunnels exist, as discussed in [25], the predictive capability of high-fidelity CFD methods for such configurations has not yet been established, especially under large blockage environments. The

latter is particularly relevant to small-scale, academic facilities, where closed transonic wind tunnels are much smaller in size compared to their industrial counterparts [26].

This work presents a combined experimental and computational analysis of a high-speed, propulsive model tested in a closed transonic wind tunnel under large blockage. Following the prior investigation reported in [25], this study aims to identify suitable practices based on RANS simulations and to quantify their accuracy in predicting the mean base flow characteristics under the experimental conditions. To achieve this, several key aspects are examined, including the experimental requirements and modelling approaches necessary to ensure that the computational model accurately represents the experimental flow topology. Simulations are carried out with and without the model mounting device to demonstrate its impact on the flow. Attention is given to the treatment of the nozzle inlet boundary condition. The adequacy of a simplified approach which neglects the upstream components of the nozzle flow within the experimental rig is evaluated. Based on the results, recommendations are provided for the wind tunnel simulations of powered-on models using RANS, which can be adopted in future CFD validation activities of high-speed propulsion systems, even in small-scale facilities with large blockage.

2 Experimental approach

2.1 Rig and sub-scale model

The sub-scale model investigated in this work was designed with the main objective of providing a base flow topology that is representative of future high-speed exhaust systems. An overview of the experimental rig is shown in the simplified schematic of Fig. 2, along with a section view of the model and its main components, numbered from 1 to 6 for reference. The model has a total length of $L = 0.25$ m and a base diameter of $D = 0.08$ m. The external surface of the model, referred to as the nacelle, is formed by a tangent-ogive nose (1), followed by the main cylindrical part which consists of the air chamber (2) and nacelle afterbody (3). The model is mounted on the tunnel wall through a mounting device, referred to as pylon, which has a NACA0026 airfoil cross-section. High-pressure air from an air-supply system flows through the pylon from a circular passage which connects to the inlet section embedded along the wall of the air chamber (Fig. 2). The high-pressure air then flows into the plenum (4) through a series of small holes, which are located at the top end, before entering a convergent-divergent (CD), truncated ideal contour (TIC) nozzle (5), embedded within the base area. The plenum was introduced to the design to improve the uniformity of the flow entering the nozzle and

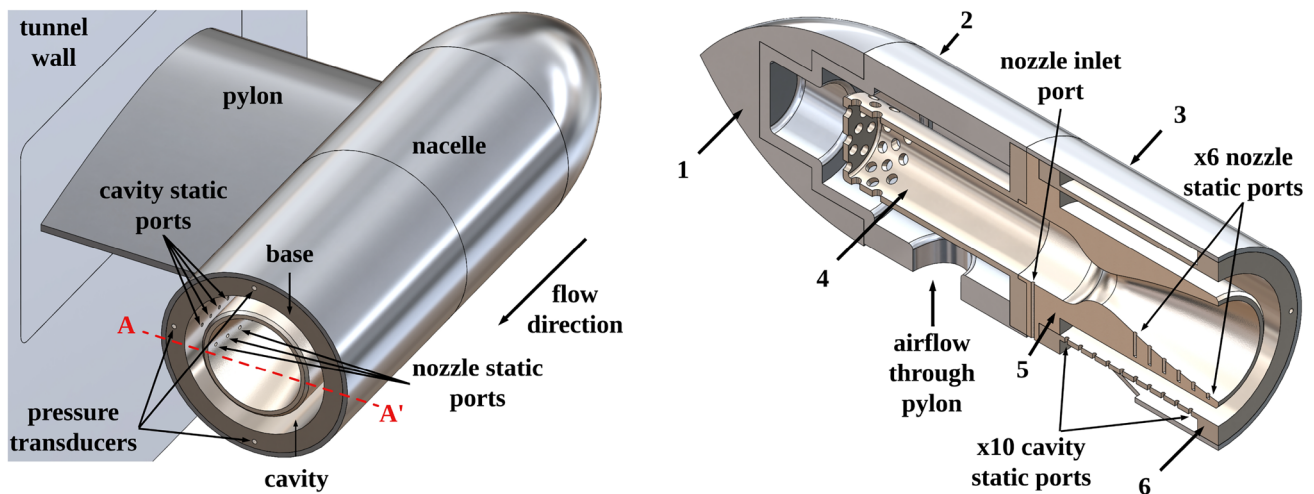


Fig. 2 Overview of the model installed on the tunnel wall (left) and cross-sectional view along mid-section A-A' (right); 1: nose, 2: air chamber, 3: nacelle afterbody, 4: plenum, 5: TIC nozzle, 6: cavity

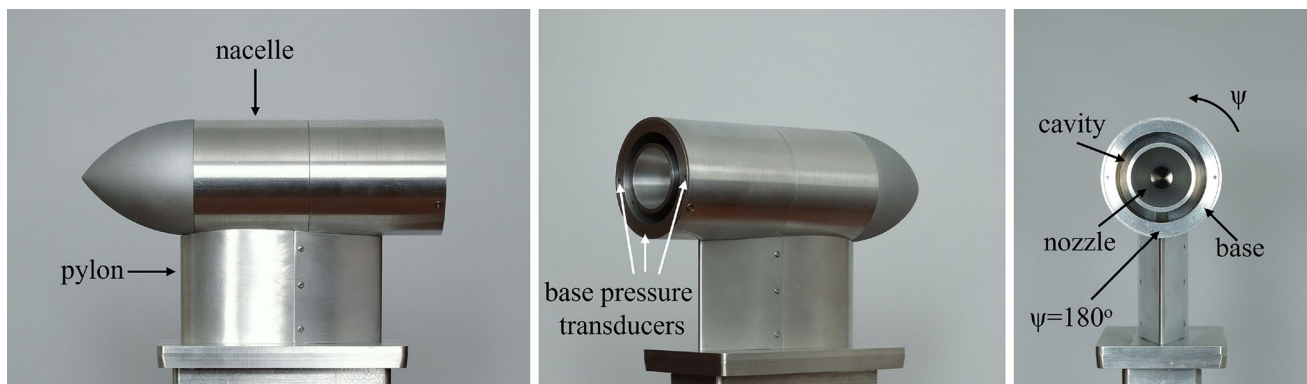


Fig. 3 Different views of the examined model. Top view as mounted on the tunnel side wall (left), location of base pressure transducers (middle) and rear view (right) with definition of azimuthal angle

mitigate the jet asymmetry as a result of the 90° turn that the flow undergoes from the pylon to the nozzle inlet, that was observed using preliminary computational analyses during the design of the experiments. This was important since it enabled the uniform treatment of the nozzle inlet boundary condition as will be described in Sect. 3.2. Several plenum designs were examined prior to the one shown in Fig. 2, with main focus of reducing pressure losses and improving flow uniformity at the nozzle inlet.

The contour of the nozzle in the supersonic region is designed using the axisymmetric method of characteristics (MoC) [27]. The initial value-line for the MoC corresponds to the critical flow conditions at the sonic line which are approximated using Sauer's technique [28]. This process provides an ideal nozzle profile with a shock-free flow expansion. The final nozzle is truncated at an angle of 7.6 degrees, corresponding to a design exit Mach number of around 3.4 . Additionally, the model features a base-embedded, axisymmetric cavity region (6) around the

nozzle, which has an annular shape and a depth of 0.95 base diameters D . It is noted that this component is referred to as a cavity for convenience, but its rear surface constitutes the majority of the base surface of the model (Fig. 2, left). The actual cavity region is then formed after assembling the model, in the space between the nozzle (5) and component number 6 of Fig. 2. Throughout this work the term cavity refers to the empty space between the nozzle and the cavity walls. Cavity regions will be essential for advanced, high-speed exhaust designs, in order to enable gimbal capabilities for base-embedded propulsive nozzles [29], and therefore, its impact on the base flow was of primary focus for the experimental campaign. However, this is out of the scope of this work, and thus it will not be discussed herein. The physical model that was used in the experiments is depicted in Fig. 3 in a fully assembled configuration, along with the pylon and the slot that was used to mount the model on the tunnel wall. Figure 3 also defines the azimuthal angle ψ , with the location of $\psi = 180^\circ$ being behind the pylon.

The model was instrumented with six equally spaced static pressure ports along the nozzle divergent section and ten static pressure ports distributed along the cavity wall, as annotated in Fig. 2. One static port was also placed at the nozzle inlet to obtain the pressure at that location (Fig. 2). Furthermore, three dynamic pressure transducers were located on the base at azimuthal positions of approximately $\psi = 90^\circ$, 170° , and 270° , as shown in Fig. 3. A detailed description of the instrumentation and data acquisition system is provided in Sect. 2.3

2.2 Wind tunnel facility

The experiments were carried out at the closed-circuit, ejector-driven, Transonic Wind Tunnel (TSWT) facility of Cranfield University. The facility is shown schematically in Fig. 4. The test section is 0.5 m long and has a cross-sectional area of $A_t = (0.206 \times 0.228) \text{ m}^2$ [30]. The blockage ratio, which is the ratio of the total cross-sectional area of the model and pylon to that of the test section, is approximately $A_{\text{model}}/A_t = 16.5\%$. At its current configuration, the test section has solid walls, and therefore it is critical that all associated boundary layers are taken into account in the computational analysis. Additionally, the pylon can affect the base flow characteristics. Hence, it is necessary that the entire wind tunnel experimental environment is modelled during the computations in order to assess their accuracy, especially under such large blockage.

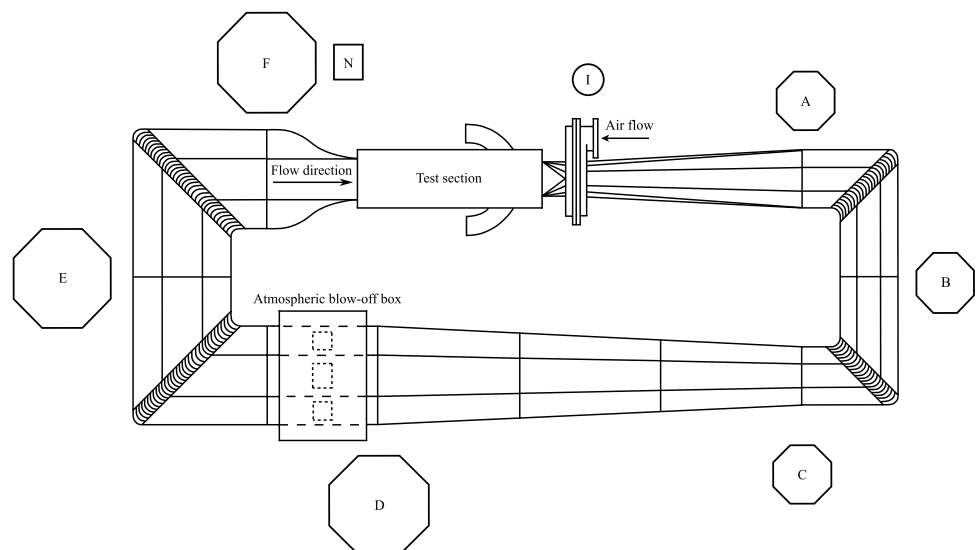
The facility features an atmospheric blow-off box (Fig. 4) which keeps the total pressure of the approaching flow, denoted as $P_{0,t}$, close to atmospheric pressure. The total pressure was measured by a Druck UNIK 5000 pressure transducer (range of 0.8–1.2 bar and accuracy ± 0.0004 bar), connected to a pitot probe before the test section nozzle. The flow conditions in the wind tunnel test section are

controlled by the downstream ejector which changes the static pressure. The mean static pressure value at the ejector's plane was not available during the runs and is generally challenging to obtain accurately. Instead, the tunnel was operated through a feed-forward system which regulates the ejector flow required to maintain a prescribed value of total to static pressure ratio at the inlet of the test section, defined as $P_{0,t}/p_{in}$, as the pressure of the external reservoir tank that drives the tunnel, drops. Here p_{in} is the reference static pressure from a port located at the tunnel wall at the inlet of the test section. Therefore, this pressure ratio translates into the corresponding Mach number, which remained within ± 0.01 of the prescribed value during the runs. Note that for an empty test section, p_{in} is essentially the same as the downstream pressure, since the flow in the tunnel is nearly isentropic. However, the introduction of the model imposes a high blockage of 16.5%, and the achieved flow conditions, in terms of approach Mach number, are affected by both the tunnel flow and the high-speed jet and are generally not known prior to the tests. This poses a big challenge in matching the flow conditions during the computational modelling.

2.3 Instrumentation and data acquisition

Static and dynamic pressure measurements were obtained during the experiments at key locations of the model described in Sect. 2.1. Additionally, 13 equidistant static ports were used to monitor the static pressure distribution along the test section side wall, opposite to the model mount, spanning a distance of $x/L = [-1, 1]$, where L denotes the length of the model and $x/L = 0$ corresponds to the base location. Therefore, the first sensor was located at the test section inlet and was aligned with the nose of the model (at $x/L = -1$), while the last was located at the

Fig. 4 Schematic of the transonic wind tunnel facility used for the experiments. Sections A–F indicate tunnel cross-sections, while sections N and I show the nozzle and ejector planes; adapted from [30]



test section outlet, one model length downstream of the base (at $x/L = 1$). This was essential in order to characterize the flow expansion around the model during the runs and thus ensure that aerodynamic similarity is established between the flow in the experimental tests and the CFD. Under this high blockage environment and with the approaching conditions being dynamically influenced by both the downstream ejector pressure and jet flow, matching the computations to the experimental conditions is not trivial.

The static pressure signals were obtained by a 32-channel Scanivalve unit (model ZOC 22B/32 Px), with range of 0.31–1.69 bar and accuracy of ± 0.00055 bar, acquired at a sampling frequency of 500 Hz for a total duration of 20 s, which encompassed the start and stop of the tunnel flow. The pressure at the nozzle inlet port, annotated in Fig. 2 (right), was acquired with a dynamic pressure transducer (type RS 797-4961) with range of 0–25 bar and accuracy of ± 0.0029 bar. The dynamic pressure measurements at the base of the model (Fig. 3) were acquired by three Kulite miniaturized transducers (XCQ-0.62–1.7barA), with range of 0–1.7 bar and accuracy of ± 0.0017 bar. Each signal of the Kulite and RS transducers was low-pass filtered by a Dewetron signal-conditioner unit at 10 kHz, before being sampled at 25 kHz. The base pressure signals were sampled for a duration of 8 s, during which the jet flow was active for 4 s. The final results correspond to the time averaged value of the base pressure signals during the period when the jet was active. The combined experimental uncertainty was quantified using independent runs from repeatability tests as well as accounting for the measurement uncertainty. For all of the results shown in this study, the maximum uncertainty in the time average pressure values was less than 2%.

3 Computational approach

3.1 Solver and numerical details

All simulations were performed using a steady-state RANS formulation coupled with the $k - \omega$ shear stress transport (SST) eddy viscosity model of Menter [31], which solves two additional transport equations for the turbulent kinetic energy k and specific dissipation rate ω . The formulation is not repeated here for brevity. The interested reader is referred to [31]. This turbulence model has been found to perform better over other eddy viscosity models for a number of flows involving strong separation, such as the flow over a two-dimensional backward facing step (BFS) [32], as well as other test cases including adverse pressure gradient with separation, a NACA 4412 airfoil at high angle of attack and a transonic bump case [33]. Additionally, this model has shown reasonably good accuracy for over-expanded nozzle

flows [34], as well as for afterbody flows with high-speed jets and separation [35], which is critical in this work.

For the simulations, a compressible, density-based solver with implicit formulation was used in ANSYS Fluent [36]. A second-order upwind discretization scheme was employed for the flow primitive variables as well as for the turbulent kinetic energy and specific dissipation rate. The convective fluxes were calculated using Roe's flux difference splitting method [37]. The gradients of the flow field were obtained using a Green-Gauss cell-based method, to reduce the cost of a node-based approach due to the polyhedral grid topology. An 8th order piecewise polynomial was used for the calculation of specific heat capacity as a function of static temperature [38], while thermal conductivity was computed according to kinetic theory [38]. Finally, the dynamic viscosity was obtained according to Sutherland's law [39].

3.2 Computational domain and boundary conditions

Two different computational domains were considered in this study: one representing the full test section and another modelling half of the wind tunnel test section, making use of the symmetry plane $x - z$ through the application of a symmetry boundary condition (BC). The two approaches are referred to as full and half domains throughout the paper. This allows to examine the impact of a symmetry boundary condition under such large blockage ratio on the accuracy of the simulation, primarily on the base pressure measurements. Specifically, it is of interest to investigate if a half-tunnel domain can provide accurate results with considerably reduced computational effort.

The computational domain used for the simulations considered the entire experimental environment, including the wind tunnel test section walls, mounting device and sub-scale model. The domain extends 3 and 10 model lengths upstream and downstream from the model's base, respectively, as shown in Fig. 5 for the half-tunnel approach (the full domain is not shown for brevity). A pressure inlet boundary condition was applied at the inlet of the domain to model the approaching flow conditions, prescribing the total pressure $P_{0,t}$, and temperature $T_{0,t}$. The total pressure was set to atmospheric value due to the wind tunnel operation, while the total temperature was set to 290 K. At the outlet of the domain, a pressure outlet BC was applied, specifying the static pressure value denoted as p_{out} . This value regulates the tunnel pressure ratio (TPR), defined as $TPR = (P_{0,t}/p_{out})$. Finally, no-slip, viscous and adiabatic wall boundary conditions were applied to all surfaces of the model, mounting device and test section.

The conditions at the nozzle inlet were much more challenging to specify due to the complex flow interactions

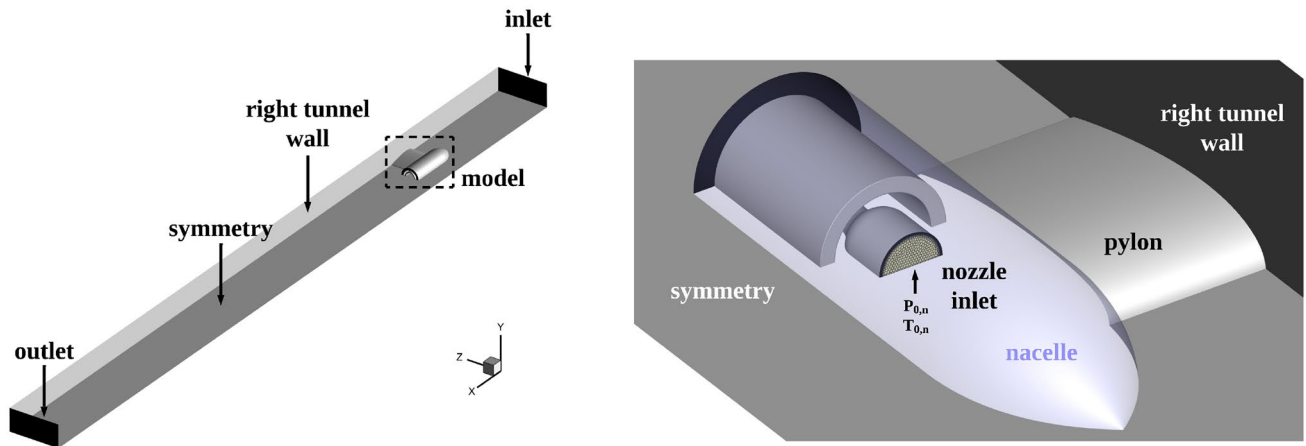


Fig. 5 Half-tunnel computational domain (left) and close-up view on nozzle inlet (right) with corresponding boundary conditions

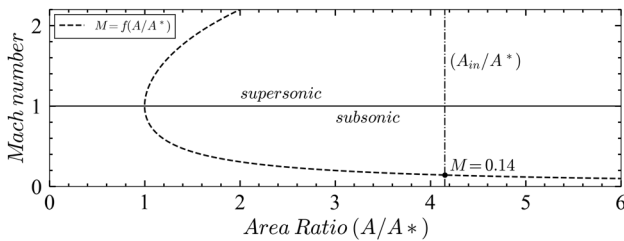


Fig. 6 Area-Mach relation for ideal gas and inviscid flow and theoretical Mach number at the nozzle inlet

occurring upstream of the nozzle in the experiments. Specifically, in the experimental setup, air flows radially from the air chamber into the plenum through a series of holes before entering the nozzle, as described in Sect. 2.1 (Fig. 2). This is very challenging to model in the computational analysis, as a complete description of the upstream geometrical details would add significant overhead, especially during mesh generation and could potentially lead to convergence issues. To address these challenges, an alternative approach is proposed in an attempt to balance accuracy and efficiency. Instead of modelling the entire upstream system, the nozzle inlet is treated with a uniform pressure inlet boundary condition, where the total pressure and temperature ($P_{0,n}$ and $T_{0,n}$) are specified.

The total pressure value can be approximated based on the mean pressure measurement obtained through the static port at the nozzle inlet, as detailed in Sect. 2.3 (Fig. 1, right). Note that the flow at this location is not at stagnation conditions, and therefore that pressure reading corresponds to the static pressure of the flow. This is inevitable, since obtaining the total pressure value directly would require a pitot probe at that location which would negatively affect the flow uniformity at the nozzle inlet. The Mach number at the nozzle inlet can then be approximated numerically using quasi-one-dimensional inviscid flow theory, based on the given area ratio between the inlet and throat sections of the

nozzle, as described in Eq. (1) [40] and depicted in Fig. 6. As observed, the subsonic solution for the given TIC nozzle corresponds to a Mach number that is approximately equal to 0.14 at the nozzle inlet location. Therefore, based on isentropic flow, the total pressure at the nozzle inlet $P_{0,n}$ is within less than 1.4% of the measured static pressure value, and thus can be corrected accordingly for the computational analysis. The total pressure value specifies the nozzle pressure ratio (NPR), which in this work is defined as $\text{NPR} = (P_{0,n}/p_{out})$. Here p_{out} is the static pressure at the outlet boundary of the computational domain, and is used to avoid ambiguous definitions, since the flow field in the vicinity of the model is largely affected by the blockage. The total temperature $T_{0,n}$ was set to 290 K due to the cold-flow tests.

$$\frac{A}{A^*} = \frac{1}{M} \left[\frac{2}{\gamma + 1} \left(\frac{\gamma - 1}{2} M^2 \right) \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \quad (1)$$

This approach enables a much preferable modelling, avoiding the need to consider the entirety of the upstream components of the secondary flow, with drastically reduced computational overhead. However, measurement of the mean static pressure at the nozzle inlet is required during the experiments. It is noted that similar challenges will arise in terms of CFD modelling for most propulsive wind tunnel models, where the secondary flow to the nozzle is provided through the support (or pylon), and the flow has to undergo a sharp turn. It is therefore of broader interest when it comes to experimental testing of powered-on models, to examine if such an approach can provide reasonable results.

3.3 Computational mesh

The computational mesh was constructed using a hybrid approach with polyhedral elements along with 35 prism layers near the wall surfaces for sufficient boundary layer

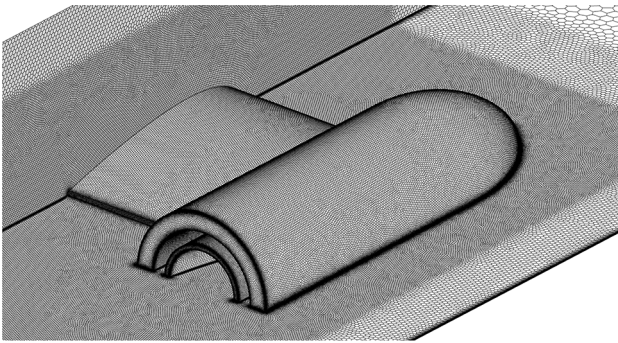


Fig. 7 Computational mesh for the half-tunnel domain

resolution, ensuring a non-dimensional wall distance of $y^+ < 1$ for all wall-adjacent nodes. Refinement regions were used closer to the base, pylon and near-wake regions for increased resolution, with the finest region extending approximately 3.2 base diameters downstream of the base. Figure 7 shows a close-up view of the surface mesh for the half-tunnel domain. The uncertainty due to grid discretization has been assessed through a rigorous, second-order Grid Convergence Index (GCI) analysis [41] for the mesh corresponding to the half-tunnel domain. Three grids were used, consisting of approximately 3, 13.7 and 44 million cells, thus giving a grid refinement factor greater than 1.3, as recommended in [41]. A detailed summary of the results of the GCI study can be found in previous work [25]. The GCI corresponding to the medium mesh for the gross propulsive force (GPF) was 0.68%. The GPF represents the overall aerodynamic force component produced by the propulsive system in the freestream direction and is calculated as the standard gross thrust minus the sum of the aerodynamic forces exerted on the wall surfaces of the model and pylon that reside within the drag domain. The reader is referred to [25, 29] for more information on the thrust and drag accounting of this configuration. The final grid used for the simulations consisted of approximately 13.7 and 26 million cells for the half and full domains, respectively.

4 Results and discussion

4.1 Impact of model support

The mounting device of the model can have a notable impact on the achieved flow field, especially under large blockage ratios in closed, transonic tunnels. To examine its effect, simulations have been carried out at several operating conditions, where the mounting device has been removed from the computational domain, making the examined geometry axially symmetric. It is noted that the comparison is carried out at the same nominal TPR and NPR conditions to assess

the pylon impact from a wind tunnel operation perspective. Therefore, the approach Mach number is not the same between the two cases. The aerodynamic effect of the pylon at identical approaching conditions should be examined in future work. The grid topology and refinement regions have been maintained identical as shown for the installed model (Fig. 7), to isolate the impact of the pylon. The computed RANS flow field at a TPR value of 1.39 and two different NPRs—a low NPR of 5 and a high NPR of 20—is shown in Fig. 8 for the installed model and the one where the pylon is removed. The effect of the model support on the base flow is considerable, and manifests primarily through the modification of the flow conditions around the model due to reduced blockage.

Specifically, the blockage ratio of 16.5% in the installed model case, drops to around 10.7% when the pylon is removed. As a result, the achieved Mach number in the vicinity of the base and $\psi = 0^\circ$ is reduced by nearly 6% in the low NPR condition, decreasing from 0.803 to approximately 0.754, as annotated in Fig. 8. This effect is notably enhanced in the higher NPR of 20, where the Mach number reduction reaches approximately 12%, decreasing from a value of 0.925 in the installed model to 0.815 when not accounting for the support. This can be readily observed from the base flow structure, especially at the condition of NPR=20. The jet flow in the no-ptylon case features reduced expansion with a stronger Mach disk reflection due to the increase in the base pressure where the nozzle expands, which reduces the effective NPR relative to the installed case. Specifically, the effective NPR, defined as the ratio of the nozzle inlet total pressure over the local static pressure at the base, is approximately 24.5 and 28 for the cases without and with the pylon, respectively. The increased base pressure is a consequence of the reduced blockage when the pylon is removed. The same effect is obvious further downstream in the jet shock-cell pattern, as highlighted in the figure.

The effect of the modified blockage environment due to the presence of the mounting device on the jet flow can be quantified in terms of the location of incipient separation along the nozzle wall during over-expansion. The location of incipient separation, denoted as x_i , is defined as the axial location of minimum wall pressure, corresponding to the point where the nozzle wall pressure gradient changes sign and the pressure distribution departs from the ideal expansion profile predicted by the MoC. In the present work, x_i is used as an indicator of the onset of the separation process due to shock-wave/boundary-layer interaction. It is noted that this point does not correspond to the exact flow separation location, which is where the wall shear stress becomes zero. The non-dimensional incipient locations are depicted in Fig. 9 for 5 different TPRs, ranging from 1.28 to 1.79 and

Fig. 8 Mach number flow field at $TPR = 1.39$ and NPRs of 5 and 20 with and without the pylon

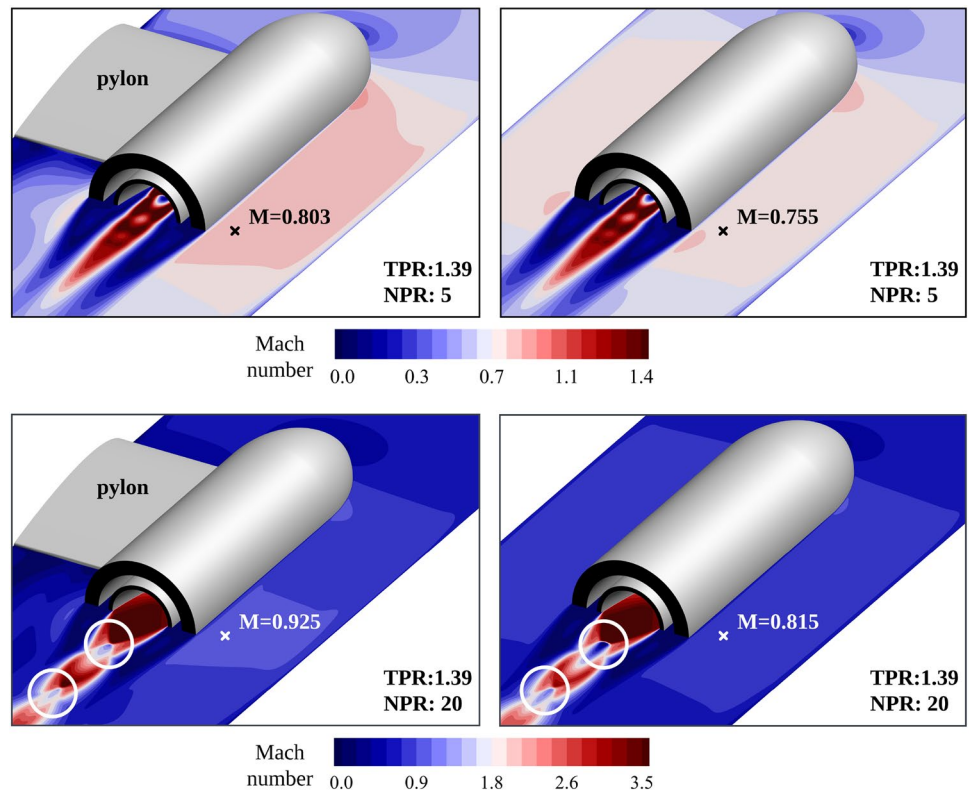
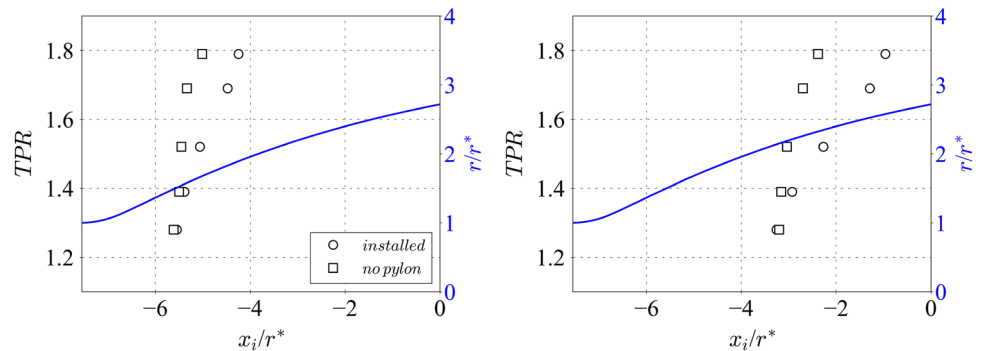


Fig. 9 Variation of incipient separation location x_i with TPR at NPR = 5 (left) and NPR = 10 (right) with and without including the model support; the blue line denotes the nozzle contour



two different NPR values of 5 and 10, corresponding to nozzle over-expansion state. The value of $x_i/r^*=0$ corresponds to the nozzle exit, with r^* denoting the nozzle throat radius. As can be seen, the presence of the pylon gradually shifts the incipient location further downstream inside the nozzle as the TPR increases. Additionally, the effect is intensified at the higher NPR case, in line with the observations shown in Fig. 8.

These results demonstrate that under non-negligible blockage conditions, it is essential to include the mounting device in the simulation in order to obtain a proper description of the state of the flow achieved experimentally. This is particularly critical for transonic testing of high-speed, powered-on models, where the need for increased support thickness to accommodate the secondary flow, combined with the size of most closed, transonic facilities in academia [26],

lead to large blockage ratios. In addition to the flow modification due to blockage variation, the presence of the pylon creates asymmetry in the base flow which in turn leads to a jet deflection, especially at lower NPRs [25].

4.2 Assessment of computational approach

4.2.1 Aerodynamic similarity

A critical aspect of any CFD validation activity is to ensure that aerodynamic similarity has been established between the calculations and the experimental conditions. In other words, it is essential to ensure that the flow conditions of the experimental environment have been properly matched in the CFD prior to any comparison. This task is not trivial even for simple models under negligible blockage conditions.

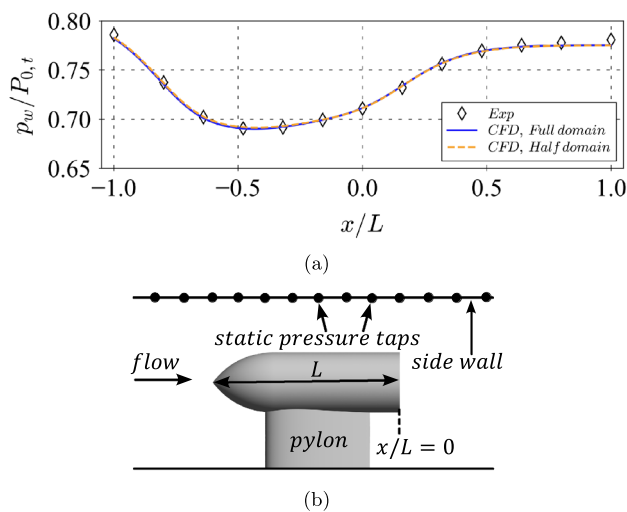
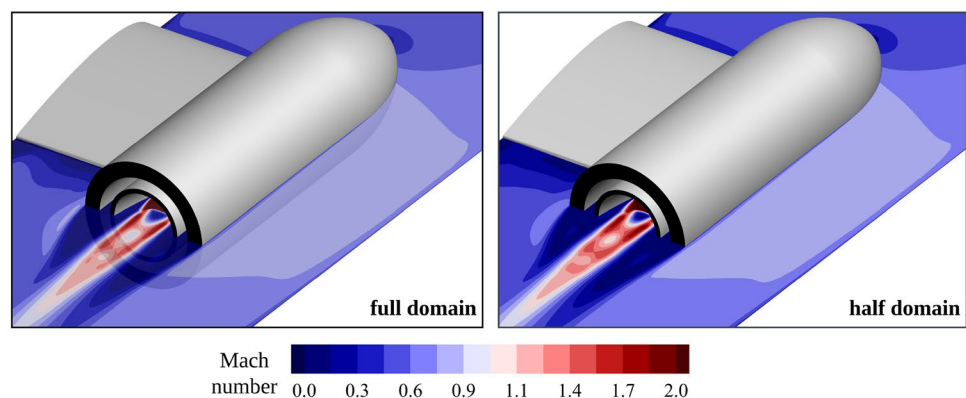


Fig. 10 Pressure distribution comparison between experimental measurements and CFD results along the test section side wall centerline (a), and schematic of the rig showing the side wall and the pressure tap locations (b)

The situation becomes much more challenging in the case of powered-on models under high blockage ratios, where the jet flow affects the approaching tunnel flow and vice versa.

As described in Sect. 3, the CFD analysis is carried out on the basis of TPR and NPR. However, the mean value of the downstream ejector plane was not available during the tests. Therefore, a direct match in the boundary conditions is impractical. This can be addressed by tuning the TPR and NPR used in the CFD to match the operating conditions achieved experimentally. Specifically, the total to static pressure ratio at the test section inlet ($P_{0,t}/p_{in}$), along with the nozzle inlet total pressure ($P_{0,n}$), uniquely define the aerodynamic conditions during each run for pre-choking tunnel operation. These values can then be used in the computational analysis to backtrack to the appropriate boundary conditions in terms of TPR and NPR. This approach was adopted in this work. The conditions of the experimental test used for validation translate to values of TPR=1.35 and NPR=6.6. This was the maximum NPR achievable within the experimental facility.

Fig. 11 Mach number flow field at the x-z plane (symmetry) computed with the full and half domains



Additionally, under such large blockage environment, small variations in the approaching flow and/or nozzle inlet conditions can affect considerably the state of the flow around the model, especially in the vicinity of the base. This can lead to erroneous conclusions regarding the accuracy of the CFD in predicting the base and nozzle flows which are directly affected by the upstream flow expansion. To mitigate this, the pressure distribution along the test section side wall opposite to the model mount was monitored within a distance of $x/L = [-1, 1]$ (Sect. 2.3), in order to characterize the flow conditions around the model. The CFD results were then compared with the experimental data to ensure that aerodynamic similarity is established in the test section. This is shown in Fig. 10a for both full and half test section computational domains, while Fig. 10b shows a schematic of the apparatus with the side wall and pressure taps annotated. As observed, the computations properly capture the pressure drop over the forebody of the model as well as the pressure recovery over the afterbody and the near wake further downstream. Note that the pressure distribution shown in Fig. 10a reflects the local wall pressure characteristics in the vicinity of the model, whereas the TPR is defined based on the prescribed outlet static pressure, which is located 10 model lengths downstream of the base. Both full and half tunnel domains yield similar results which agree with the mean experimental static pressure data within less than 0.7%, demonstrating that good aerodynamic similarity is established in the test section between the simulations and the experiment. This is essential prior to further assessment of the CFD accuracy. It is noted that the pressure distribution along the centerline of the top wall yielded similar results between the two computational domains (not shown for brevity).

Figure 11 presents the mean flow field in terms of Mach number, obtained with RANS for the examined conditions, for both the full and half test section domains. As shown, at this NPR, the nozzle operates under a severe over-expansion state, exhibiting a free shock separation (FSS) with a typical Mach disk reflection established inside the divergent section at a location of $x/D \approx -0.23$, where $x/D = 0$ corresponds to

the base. The Mach number close to the base of the model and zero degrees azimuthal angle (i.e., opposite to the pylon location) is approximately 0.75. The base flow asymmetry caused as a result of the presence of the mounting device is also evident in the separated flow at the wake. The two modelling approaches, using the full and half test sections, produce very similar results, in line with the agreement in the pressure distribution along the side wall shown previously (Fig. 10). Only minor differences can be observed in the shock-cell pattern of the high-speed jet downstream of the nozzle exit.

4.2.2 Base flow prediction

The accuracy of RANS for the downstream base flow is now assessed. The comparison between CFD and experimental data for the mean pressure distribution along the wall surfaces of the base, cavity and nozzle is shown in Fig. 12. Both full and half tunnel domains are included in the comparison. The azimuthal distribution of the base pressure predicted by both computational approaches is close to the experimental values with a maximum error below 1.6%. The CFD model captures the reduced pressure levels behind the mounting device ($\psi = 180^\circ$), although it does not predict the pressure asymmetry of the experimental data between $\psi = 90^\circ$ and 270° . The difference between the full and half tunnel domains is negligible (less than 0.3%). Similarly, the pressure inside the cavity region predicted by RANS is close to the mean experimental data, with a maximum error which

is less than 0.8%, identified for the location closest to the base (i.e., at $x/L = -0.175$). However, the pressure recovery towards the cavity wall is steeper in the experiments relative to the computations, suggesting that the internal cavity flow development is not fully captured by the simulations. The results by both CFD domain approaches are very close.

The nozzle flow prediction is shown in the bottom graph of Fig. 12, along with the ideal flow expansion profile from the MoC. The axial location is non-dimensionalized with the throat radius ($x/r^* = 0$ denotes the nozzle exit). The computations with both domains yield very similar predictions which seem to capture the flow separation behavior with reasonable accuracy. Note that to identify accurately the location of incipient separation in the experiments, a considerably higher number of sensors would be required to increase the spatial resolution of the pressure profile. This was not feasible in this campaign and is typically challenging due to practical limitations, including the model size, and available space to pass the instrumentation tubing. Regardless, within the available resolution of the experimental sensors, CFD agrees with the experiment with a maximum error of approximately 1.25%. The predicted pressure in the separated flow region recovers to the experimental values. These results indicate that the nozzle inlet treatment approach described in Sect. 3.2, can be adopted without the need to model the upstream components, thereby reducing the computational effort. Overall, it can be argued that the RANS approach as adopted in this work, accurately captures the mean flow state at the examined conditions of TPR=1.35 and NPR=6.6, with a maximum error below 1.6% identified in the base pressure prediction.

4.3 Relevance to unbounded flow

In many experimental campaigns within the context of propulsive base flows, the main objective is to identify and understand the fundamental features that drive the complex flow topology at the base. One characteristic example is the assessment of active and passive base flow control techniques, and their capability in yielding base drag improvements for high-speed systems, particularly at transonic speeds. However, it is noted that even if free-air corrections are not the primary aim, relevance to free-air conditions could be compromised due to large blockage effects or model support interference and could lead to misleading interpretations of the obtained results. This is particularly important in academia, where the limited size of most transonic facilities [26], combined with the need to test larger models (especially if powered-on), leads to high blockage environments. However, examining the relevance of the base flow relative to free-air conditions is challenging, as even establishing a fair comparison between a large-blockage,

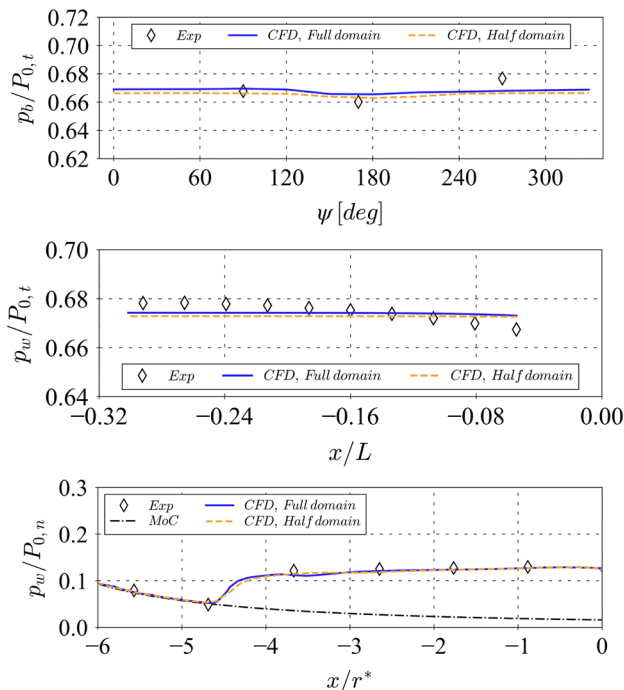


Fig. 12 Comparison between experimental measurements and CFD results; top: base, center: cavity wall, bottom: nozzle wall

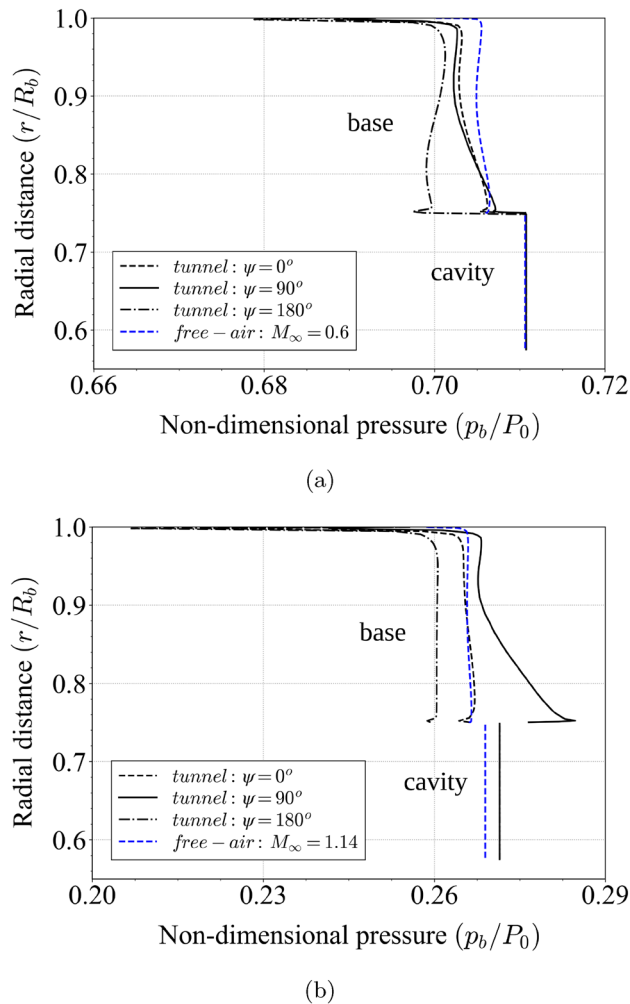


Fig. 13 CFD pressure distribution along the walls of the base and cavity for the tunnel-installed and free-air cases at NPR=10: **a** TPR=1.28 (pre-choked tunnel) and average $M_{aft}^{eff} = 0.743$, and **b** TPR=1.79 (post-choked tunnel) and average $M_{aft}^{eff} = 1.460$

tunnel-installed model and free-air conditions is far from straightforward.

A methodology that addresses this challenge has been introduced in previous work [25] based on CFD simulations using the symmetrical domain. The main issue is matching the two cases under large blockage, as there is no way to know what are the free-air parameters (i.e., Mach number) that need to be applied to match the flow near the base of the tunnel-installed model. The proposed method is based on matching the effective Mach number at the end of the afterbody M_{aft}^{eff} , defined in Eq. (2). Note that this is the isentropic Mach number based on the pressure ratio of the atmospheric total to local static pressure near the end of the nacelle afterbody, evaluated as an average over approximately three computational cells upstream of the base edge [25]. This location was selected to characterize the flow approaching the base while avoiding the

local discontinuity associated with the base cut-off point. The superscript ψ refers to the azimuthal angle as shown in Fig. 3, which for the CFD study varied from $0^\circ - 180^\circ$, with 180° behind the mounting device.

$$M_{aft}^{eff}(\psi) = \left[\frac{2}{(\gamma - 1)} \left(\left(\frac{P_0}{p_{aft}^\psi} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right) \right]^{\frac{1}{2}} \quad (2)$$

A series of free-air calculations was performed for Mach numbers between 0.5 and 1.5 with a 0.05-step, using an axisymmetric approach. One can then map the TPR, which regulates the tunnel flow, with the free-air Mach number, such that the same effective Mach number (within 0.6%) is obtained at the end of the nacelle afterbody. The latter in the installed case represents an azimuthal average extracted from 5 locations of $\psi = 0^\circ, 45^\circ, 90^\circ, 135^\circ$ and 180° to account for three-dimensionality effects. The radial pressure distribution along the base and cavity walls is then compared between the two cases for which the M_{aft}^{eff} is matched, to assess their similarity.

An example of that comparison is shown in Fig. 13 for two different TPRs of 1.28 (Fig. 13a) and 1.79 (Fig. 13b) corresponding to pre-choking and post-choking tunnel conditions, respectively. For TPR = 1.28, the free-air Mach number that provides the same effective Mach number of approximately 0.743 is equal to 0.6, while in the TPR = 1.79 case, a super-

sonic Mach number of 1.14 is required to give $M_{aft}^{eff} = 1.46$. The comparison is good for pre-choking tunnel operation, with a maximum difference below 0.7% for the base pressure. This value increases to around 1.4% behind the mounting device, as expected, since it is not present in the free-air cases. Note that the short region close to the base edge ($r/R_b \approx 1$) where higher differences are observed, corresponds to less than 1.5% of the base area. Therefore, its influence on the area-averaged base-pressure characteristics is expected to be limited. The pressure along the cavity wall is in excellent agreement ($< 0.1\%$). For the post-choking tunnel condition, the base pressure of the free-air case still falls within the azimuthal range of the tunnel-installed conditions with a maximum difference of approximately 6% identified at $\psi = 90^\circ$. Note that the comparison is much better at $\psi = 0^\circ$. This difference is expected, as the test section is not square, which at choked conditions has a large effect. The cavity pressure is close within 1%. Overall, good base pressure similarity can be established with respect to free-air conditions through the proposed method for the medium NPR examined. This demonstrates that the base flow can still be relevant to free-air conditions under large blockage and is promising for the use of small-scale facilities for base drag studies.

5 Conclusions

A combined experimental and computational study has been conducted to investigate the base flow of a high-speed, powered-on model tested in a small-scale transonic wind tunnel under large blockage conditions. The main conclusions are as follows:

- Proper description of the flow requires modelling of the full experimental environment, including the test section walls and mounting device. This is particularly important for powered-on models where the support typically accommodates the jet flow and has appreciable size, thus increasing blockage effects.
- The nozzle inlet treatment can be simplified by prescribing a uniform boundary condition which eliminates the need to model the detailed upstream geometry of the experiment. This requires experimental measurement of the static pressure at the nozzle inlet, which can be corrected based on inviscid flow theory to approximate the total pressure. This approach has shown to yield reasonably accurate nozzle flow prediction.
- Matching the experimental flow conditions for a powered-on test is not trivial, particularly under large blockage. The pressure distribution on the tunnel side wall can be used to characterize the flow state over the model and near-wake. Therefore, wall pressure measurements should be included in the experimental design to ensure aerodynamic similarity and support CFD validation.
- RANS computations coupled with the $k - \omega$ SST eddy viscosity model are capable of accurate base pressure prediction with a maximum error below 1.6% relative to the experimental data for the examined operating conditions.
- Local aerodynamic similarity relative to free-air conditions can be established based on the proposed metric. This is promising for future base flows studies under large blockage conditions, which are typically encountered in small-scale transonic facilities.

The novelty of this work is the generation of new experimental and computational data for powered-on, high-speed base flows under transonic wind tunnel conditions, including blockage and support interference effects. Based on the combined analysis of this dataset, the contribution of this study is the synthesis of suitable computational practices and the quantification of their accuracy for in-tunnel simulations of powered-on models. The overall conclusion is that the proposed approach can accurately predict the mean base flow even under large blockage conditions. This was previously unreported based on the open literature. The study also highlights that such tunnel-constrained flows can be systematically mapped to equivalent free-air conditions. The impact of the work is that it enables

the use of small-scale, transonic facilities for future base flow studies of high-speed systems, thereby reducing the need for high-cost, large-scale experiments.

Acknowledgements The authors would like to thank 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. Mark Finnis during the experimental campaign is greatly acknowledged.

Author contributions S.T.: Conceptualization, Methodology, Formal Analysis, Visualization, Writing—Original Draft, Writing—Reviewing and Editing M.D.: Conceptualization, Methodology, Writing—Original Draft, Writing—Reviewing and Editing I.G.: Conceptualization, Supervision, Project Management, Writing—Reviewing and Editing S.P.: Conceptualization, Supervision, Project Management, Writing—Reviewing and Editing V.Z.: Conceptualization, Supervision, Project Management

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

Declarations

Conflict of interest Simon Prince, co-author of this paper, is also a guest editor of this special issue, but was not involved in the review process. All other authors have no conflict of interest to declare that are relevant to the content of this article.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

1. Cumpsty, N.A.: Jet Propulsion: A Simple Guide to the Aerodynamic and Thermodynamic Design and Performance of Jet Engines, 2nd edn. Cambridge University Press, Cambridge (2003)
2. McClinton, R.C.: High speed/hypersonic aircraft propulsion technology development. In: Advances on Propulsion Technology for High-Speed Aircraft. RTO-EN-AVT-150. NATO Research and Technology Organization, Neuilly-sur-Seine, France (2008). <https://doi.org/10.14339/RTO-EN-AVT-150-01-pdf>
3. Bogar, T., Eiswirth, E., Couch, L., McClinton, C.: Conceptual design of a mach 10, global reach reconnaissance aircraft. In: 32nd Joint Propulsion Conference and Exhibition (1996). <https://doi.org/10.2514/6.1996-2894>. AIAA 1996-2894
4. Curran, F., Hunt, J., Lovell, N., Maggio, G., Bilardo, V.: The benefits of hypersonic airbreathing launch systems for access to space. In: 39th AIAA/ASME/SAE/ASEE Joint Propulsion

- Conference and Exhibition (2008). <https://doi.org/10.2514/6.2003-5265>. AIAA 2003-5265
5. Botelho, A., Martinez, M., Recupero, C.: Design of the landing guidance for the retro-propulsive vertical landing of a reusable rocket stage. *CEAS Space J.* **14**, 551–564 (2022). <https://doi.org/10.1007/s12567-022-00423-6>
 6. Jian, D., Zhang, Q.: Key technologies for thermodynamic cycle of precooled engines: a review. *Acta Astronaut.* **177**, 299–312 (2020). <https://doi.org/10.1016/j.actaastro.2020.07.039>
 7. Tsentis, S., Gkoutzamanis, V., Gaitanis, A., Kalfas, I.: Multi-platform app-embedded model for hybrid air-breathing rocket-cycle engine in hypersonic atmospheric ascent. *The Aeronaut. J.* **125**(1291), 1631–1653 (2021). <https://doi.org/10.1017/aer.2021.3>
 8. Villace, V.F., Paniagua, G.: Simulation of a combined cycle for high speed propulsion. In: 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition (2010). <https://doi.org/10.2514/6.2010-1125>. AIAA 2010-1125
 9. Schwane, R.: Numerical prediction and experimental validation of unsteady loads on ariane5 and vega. *J. Spacecr. Rocket.* **52**(1), 54–62 (2015). <https://doi.org/10.2514/1.A32793>
 10. Saile, D., Gülhan, A.: Plume-induced effects on the near-wake region of a generic space launcher geometry. In: 32nd AIAA Applied Aerodynamics Conference. American Institute of Aeronautics and Astronautics, Atlanta, GA (2014). <https://doi.org/10.2514/6.2014-3137>. AIAA 2014-3137
 11. Pain, R., Weiss, P.E., Deck, S.: Zonal detached eddy simulation of the flow around a simplified launcher afterbody. *AIAA J.* **52**(9), 1967–1979 (2014). <https://doi.org/10.2514/1.J052743>
 12. Engineering Sciences Data Unit (ESDU): Blockage corrections for bluff bodies in confined flows. Technical Report ESDU 80024, ESDU, London, UK (1980). With Amendments A (1998)
 13. Saathoff, P.J., Melbourne, W.H.: Freestream turbulence and wind tunnel blockage effects on streamwise surface pressures. *J. Wind Eng. Ind. Aerodyn.* **26**(3), 353–370 (1987). [https://doi.org/10.1016/0167-6105\(87\)90005-5](https://doi.org/10.1016/0167-6105(87)90005-5)
 14. Choi, C.-K., Kwon, D.-K. (1998): Wind tunnel blockage effects on aerodynamic behavior of bluff body. *Wind Struct.* **1**(4), 351–364. <https://doi.org/10.12989/was.1998.1.4.351>
 15. Huang, J., Gu, M., Gao, Y.: Blockage effects on aerodynamics of isolated tall buildings under uniform turbulent flows. *J. Wind Eng. Ind. Aerodyn.* **212**, 104607 (2021). <https://doi.org/10.1016/j.jweia.2021.104607>
 16. Katz, J.: *Race Car Aerodynamics*, 2nd edn. Robert Bentley, Cambridge, MA (2006)
 17. AGARD: Subsonic Wind Tunnel Wall Corrections. AGARDo-graph No. 109. Advisory Group for Aerospace Research and Development, Neuilly-sur-Seine, France (1966). <https://doi.org/10.14339/AGARD-AG-109>
 18. AGARD: Wind Tunnel Wall Correction, 2nd Edition. AGARDo-graph No. 336-2. Advisory Group for Aerospace Research and Development, Neuilly-sur-Seine, France (2026). <https://doi.org/10.14339/STO-AG-336-ALL-PDF>
 19. Chen, T.Y., Liou, L.R.: Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines. *Exp. Thermal Fluid Sci.* **35**(3), 565–569 (2011). <https://doi.org/10.1016/j.expthermflusci.2010.12.005>
 20. Eckert, D., Hegen, G.H., Kühn, W.: DNW's Method to Correct for Support and Wall Interference Effects on Low Speed Measurements with a Large Propeller Powered Transport Aircraft Model. In: Proceedings of the 25th International Congress of the Aeronautical Sciences (ICAS), Hamburg, Germany (2006)
 21. Ashill, P.R., Keating, R.F.A.: Calculation of tunnel wall interference from wall-pressure measurements. *Aeronaut. J.* **92**(911), 36–53 (1988). <https://doi.org/10.1017/S0001924000021813>
 22. Lessard, W.B.: Analysis of post-support and wind-tunnel wall interference on flow field about subsonic high-lift high-speed research configuration. NASA, (2000). (NASA/TP-2000-210555 Technical Report)
 23. Wubben, F., Maseland, H.: Verification of wind tunnel model support and wall interference assessments in dnw-hst by cfd simulations. In: Advances in Wind Tunnel Test Techniques, (2018). (STO-MP-AVT-284)
 24. Mavriplis, N., Ting, K.Y., Soltani, R.M., Nelson, C.P., Livne, E.: Supersonic configurations at low speeds (scalos): Cfd aided wind tunnel data corrections. In: AIAA SciTech Forum (2023). <https://doi.org/10.2514/6.2023-0231>. AIAA 2023-0231
 25. Tsentis, S., Goulos, I., Debiassi, M., Prince, S., Pachidis, V., Zmijanovic, V., Saavedra, J.: Wind tunnel installation effects on a high-speed exhaust flow under large blockage. *J. Spacecr. Rocket.* **62**(5), 1698–1712 (2025). <https://doi.org/10.2514/1.A36092>
 26. Greenwell, D.I.: Transonic industrial wind tunnel testing in the 2020s. *The Aeronaut. J.* **126**(1295), 125–151 (2022). <https://doi.org/10.1017/aer.2021.107>
 27. Zucrow, J.M., Hoffman, D.J.: *Gas Dynamics*. Wiley, New York (1976)
 28. Sauer, R.: General characteristics of the flow through nozzles at near critical speeds, NACA, Washington, DC (1947). (Memorandum No. 1147 Technical Report)
 29. Tsentis, S., Goulos, I., Prince, S., Pachidis, V., Zmijanovic, V., Saavedra, J.: Propulsion aerodynamics for a novel high-speed exhaust system. *ASME J. Eng. Gas Turbines Power* **145**(12), 121011 (2023). <https://doi.org/10.1115/1.4063416>
 30. Saddington, A., Knowles, K., Thangamani, V.: Scale effects on the performance of sawtooth spoilers in transonic rectangular cavity flow. *Exp. Fluids* (2016). <https://doi.org/10.1007/s00348-015-2088-2>
 31. Menter, F.R.: Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* **32**(8), 1598–1605 (1994). <https://doi.org/10.2514/3.12149>
 32. Haque, A., Fareed, A., Shunsuke, Y., Sajid, R.C.: Assessment of turbulence models for turbulent flow over backward facing step. *Proceed. World Congress Eng.* **2**, 1340–1345 (2007)
 33. Menter, F.R.: A comparison of some recent eddy-viscosity turbulence models. *J. Fluids Eng.* **118**(3), 514–519 (1996). <https://doi.org/10.1115/1.2817788>
 34. Allamaprabhu, C., Raghunandan, B., Morinigo, J.: Improved prediction of flow separation in thrust optimized parabolic nozzles with fluent. In: 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibition (2011). <https://doi.org/10.2514/6.2011-5689>. AIAA 2011-5689
 35. Hasan, R.G.M., McGuirk, J.J., Apsley, D.D., Leschziner, M.A.: A turbulence model study of separated 3d jet/afterbody flow. *The Aeronaut. J.* **108**(1079), 1–14 (2004). <https://doi.org/10.1017/S0001924000004942>
 36. Ansys Inc: *Ansys Fluent Theory Guide*. 275 Technology Drive, ANSYS Inc, Canonsburg, PA (2021)
 37. Roe, P.L.: Approximate riemann solvers, parameter vectors, and difference schemes. *J. Comput. Phys.* **43**(2), 357–372 (1981). [https://doi.org/10.1016/0021-9991\(81\)90128-5](https://doi.org/10.1016/0021-9991(81)90128-5)
 38. Ansys Inc: *Ansys Fluent User's Guide*. 275 Technology Drive, ANSYS Inc (2021)
 39. Sutherland, W.: The viscosity of gases and molecular forces. *Philosophical Magazine* **36**, 507–531 (1893) <https://doi.org/10.1080/14786449308620508>
 40. Anderson, J.D.: *Fundamentals of Aerodynamics*, 5th edn. McGraw-Hill, New York (2010)

41. Celik, I.B., Ghia, U., Roache, P.J., Freitas, C.J., Coleman, H., Raad, P.E.: Procedure for estimation and reporting of uncertainty due to discretization in cfd applications. *J. Fluids Eng.* (2008). <https://doi.org/10.1115/1.2960953>

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.



DSPACE

<https://dspace.org/>

Experimental and computational assessment of powered-on base flows in a transonic wind tunnel under large blockage conditions

Tsentis, Spyros; Debiasi, Marco; Goulos, Ioannis; Prince, Simon A.; Zmijanovic, Vladeta

2026-12-31

Springer

<https://doi.org/10.1007/s13272-026-00994-8>

Tsentis S, Debiasi M, Goulos I, et al., (2026) Experimental and computational assessment of powered-on base flows in a transonic wind tunnel under large blockage conditions. CEAS Aeronautical Journal, Available online 15 July 2026