

Aerodynamic Design of a High-speed Intake Test Rig for Propulsion Integration Research

Matteo Migliorini¹, Pavlos K Zachos², and David G MacManus³

Faculty of Engineering and Applied Sciences, Cranfield University, Bedfordshire, MK43 0AL, UK

Peter G Martin⁴.

Defence Science and Technology Laboratory, Fareham, PO17 6AD, UK

Inlet flow distortion is a critical factor in supersonic air induction systems, primarily due to its complex spatial variations and significant temporal unsteadiness. These characteristics can have a substantial impact on the performance and reliability of propulsion systems. Conducting experiments at an early stage of technology readiness can help mitigate risks associated with industrial testing and certification processes. This paper is part of a broader programme aimed at advancing experimental capabilities for high-speed propulsion system integration. The research focuses on incorporating advanced, non-intrusive measurement techniques alongside current industry-standard methods to better characterize engine compatibility. The paper details the optimization process for designing a fixed-throat supersonic nozzle that delivers high-quality flow at Mach 1.8, the structural and functional design of the working section. The integration of an intake test article is described, together with instrumentation for measuring synchronous total pressure and swirl distortion at the aerodynamic interface plane of the intake. Preliminary computational work is also presented, evaluating the flow quality within the working section and investigating the distribution of bypass flow in the radial exhaust.

I. Introduction

Supersonic intake systems often encounter flow distortion that is both highly complex in its spatial distribution and significantly unsteady over time. Such distortions can have critical implications for the performance and reliability of gas turbine propulsion systems [1–3]. Future designs, expected to incorporate highly three-dimensional intake geometries, will increase the design complexity and pose greater challenges for system integration [4]. Insufficient attention to these complexities in the design process may extend the time required for development and certification [3]. Similarly, the use of inlet flow control to modify inlet conditions has proven a costly exercise due to the number of iterations to obtain inlet designs with reliable performance in all flight envelope [5].

Previous distortion simulation test rigs for high-speed intakes, described in studies by Titchener [6], Kimzey and Ellis [7], Surber et al. [3,8], Gnani [9], and Fisher [10] used confined flow passage configurations to generate representative shock topologies at supersonic intake entries. These systems aimed to model the aerodynamics of the air induction stream tube, though typically only within a narrow range of key flight conditions. The focus was largely on nominal (critical) or near-nominal subcritical points, avoiding significant shock train instabilities. Constructing a model-scale facility capable of producing high-fidelity data for fan responses to such distortions,

¹ Senior Research Fellow in Experimental Propulsion Aerodynamics, Centre for Propulsion and Thermal Power Engineering, AIAA Member.

² Reader in Propulsion Aerodynamics, Centre for Propulsion and Thermal Power Engineering, AIAA Senior Member.

³ Professor in Propulsion Aerodynamics & Installations, Centre for Propulsion and Thermal Power Engineering, AIAA Member.

⁴ Fellow, Defence Science and Technology Laboratory.

under conditions closer to those encountered in real flight, is crucial. Such an approach can enhance confidence in early-stage assessments of intake-engine compatibility.

This paper outlines the design of a test rig that simulates representative inlet distortion conditions for supersonic intakes. The primary goal was to develop an experimental setup capable of replicating the internal and external aerodynamics of supersonic intakes and to analyze the impact on the flow distortion approaching the propulsion system. Building on previous work [11], the current study provides design details of the key rig components. To achieve these aims, the rig design must provide low to zero interference from the test section walls with the intake pre-entry flow whilst accommodating a wide range of intake mass flow ratios. Another critical requirement is its ability to start and remain started for a wide intake flow range, to enable research in unstable operating points including buzz [12].

The proposed test rig will leverage recent advancements in non-intrusive flow measurement methods, most of which are not industry-standard techniques and, as such, are typically part of conventional industrial programmes. Methods such as Particle Image Velocimetry (PIV), were demonstrated in subsonic environments for analyzing dynamic distortions [13–15], and thus have good potential also for high-speed applications [16]. Techniques like high-speed shadowgraph or dynamic pressure-sensitive paint could also provide high-resolution aerodynamic data around an intake entry. A scaled-down rig offers a cost-effective platform to explore such methods, enabling the testing of more configurations and producing valuable datasets to refine design tools. These datasets will also support the calibration of computational models and better align intake integration studies. Importantly, small-scale experimental setups can also serve to de-risk advanced measurement methods before their application in larger facilities, contributing to reduced costs and smoother transitions to full-scale engine tests. Such rigs provide critical insights and data necessary for certification processes, ensuring that future designs meet the demands of supersonic flight with improved efficiency and reliability.

II. Rationale and experimental facility overview

The design of the high speed intake distortion simulator was previously presented by Migliorini et al. [11], along with the initial specification of its high-pressure plant that was put in place to drive it. A blow-down, intermittent configuration was chosen, due to the cost-effectiveness, relatively easier development and wide adoption within university environments [12]. The general layout comprises a plant which delivers air at 40 bar to a 40 m³ storage tank. The flow is treated via a two-chamber desiccant drier with dew point of -50° C to prevent water vapor in the supersonic stream. A series of filters remove possible contaminants such as liquid water, oil, and particulates. The compressed air is delivered from the tank into a settling chamber installed upstream of the convergent-divergent nozzle (Fig. 1). The stagnation pressure within the settling chamber can be set between 1 and 12.5 barg, depending upon the required onset Mach number. The settling chamber contains a series of flow conditioning screens and gauzes, whereas provision for the installation of flow seeding rakes was also made to enable use of non-intrusive flow measurements methods when required. A fixed-throat, symmetric, rectangular convergent-divergent nozzle provides the working section with supersonic flow. The working section was designed as an ‘enclosed free-jet’ configuration, as previously described by Pope and Goin [12] with a size of 10x10 in² at the nozzle exit plane. The working section size subsequently increases to 18x18 inches to reduce wall interference with the aerodynamics of the intake entry. Such a design would enable both the intake captured mass flow as well as spillage flow representation within the working section. This is a unique characteristic of the chosen configuration. Previous investigations indicated that the specified working section will enable testing of an intake with 4.5-inch equivalent diameter Aerodynamic Interface Plane (AIP) to operate across nearly its entire operating range, or a partial range for 6-inch diameter AIP. The intake captured stream-tube is set via a conical plug nozzle (Fig. 2), which determines the operating Mach number at its Aerodynamic Interface Plane (AIP). The flow which bypasses the intake is collected in a radial diffuser which deflects it upwards, away from the intake ducting. This enables unrestricted access to the AIP of the intake, via an optically clear measurement section that provides line of sight for non-intrusive flow measurements of swirl and total pressure distortion. Downstream of the optical section, an industry-standard total pressure rake enables synchronous recording of total pressure distortion. This will enable direct comparison of these two approaches. The radial exhaust is exposed to ambient conditions, while the mass flow of the intake exhaust will be regulated with a plug nozzle device (Fig. 2), which will enable testing of the intake throughout the full envelope from sub-critical to critical conditions.

Acoustic silencers are installed at the exit of the exhaust streams. The rig is designed to operate between Mach numbers 1.4 - 3.0 with a runtime between 30-45 seconds depending upon the required onset Mach number. The high-pressure plant has been now commissioned and demonstrated ability to charge to the storage vessel at 40 bar in approximately 40 minutes, which is a relatively short turn-around time.

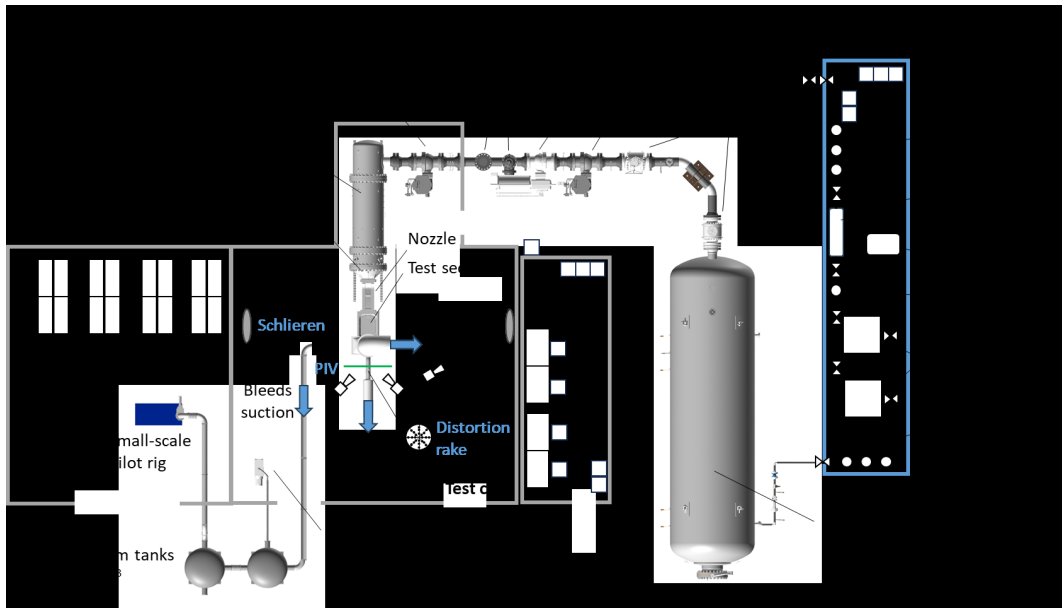


Fig. 1: Schematic of high-pressure plant and test cell configuration.

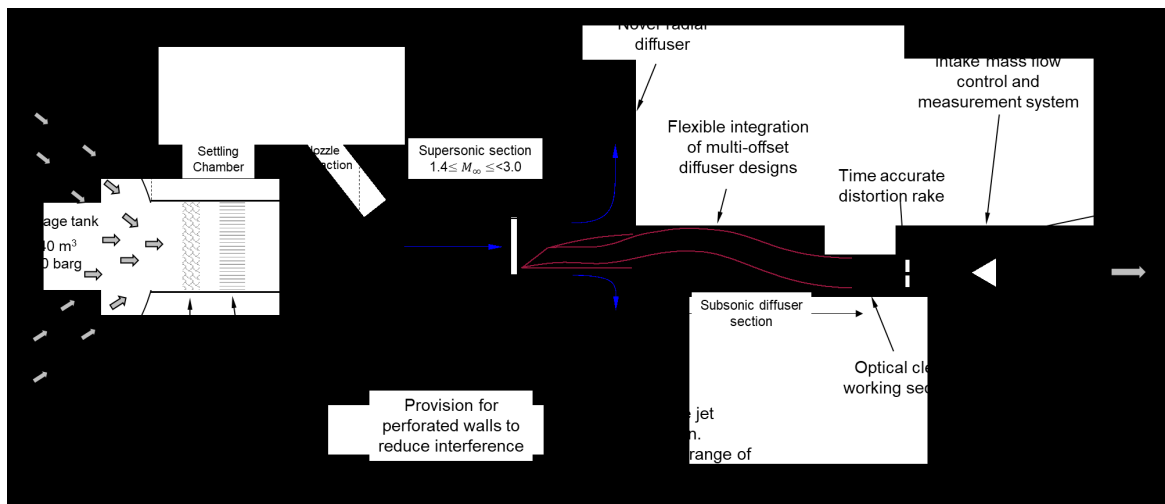
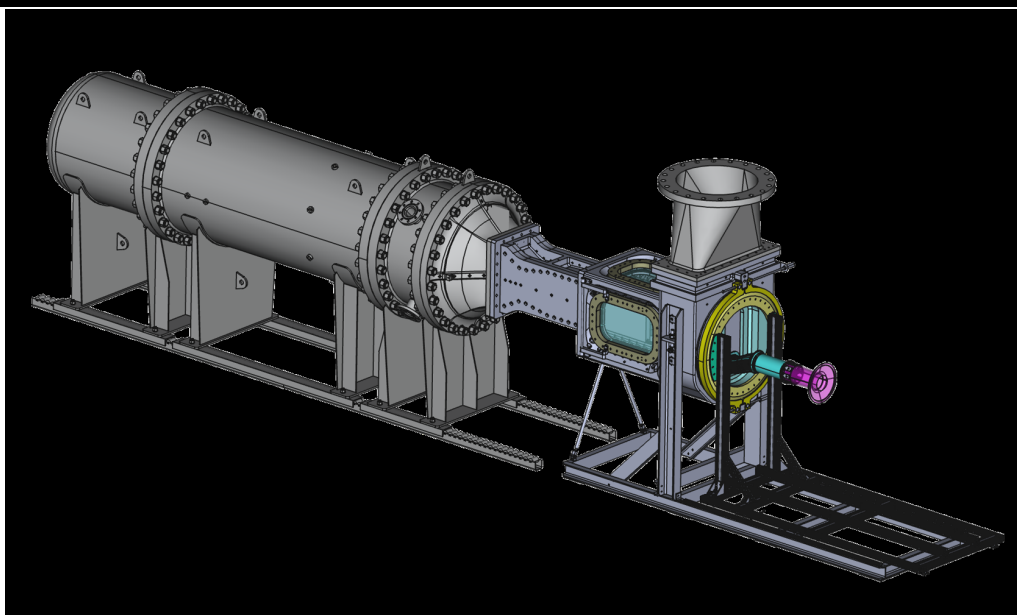


Fig. 2: Test rig schematic (top) and 3D rendered image (bottom) indicating key design characteristics.

III. Nozzle design optimization

The design of a converging-diverging nozzle to deliver uniform supersonic flow with minimal distortion and to the target Mach number, is the remit of all supersonic wind tunnel designers. Previous work by Moreno et al. [17] has established a method to design a minimum distortion 2D nozzle for target Mach 2.5 based on a parametric model and Euler simulations. The current work builds on previous methods to design a Mach 1.8 nozzle, which would be the baseline operating point of the facility.

The nozzle contours are parametrized with Bezier curves with control points a_i arranged in a polygonal shape. The Bezier curves b_i are a linear interpolation of the curve parameter t with the polygonal vertices a_i [18] (Fig. 3), defined with:

$$b_i = \sum_j a_j \binom{n}{j} t^j (1-t)^{n-j}, \quad 0 \leq t \leq 1 \quad (1)$$

The target nozzle design delivers flow acceleration via variation of the upper and lower wall contours, while the lateral walls remain parallel. Thus, the width of the nozzle is maintained constant and equal to the working section width of 10 inches. The stream-wise area distribution of the nozzle is controlled via the vertical position of control points at the nozzle inlet plane (a_1) and throat (a_4). The latter were obtained using isentropic flow equations for $M=0.2$ and 1.0 , respectively, relative to the fixed working section height $H_{WS} = 10$ inches (a_7 , Fig. 3). The vertical positions of the other control points (a_2, a_3, a_5, a_6) respects the tangency conditions of the curves for zero first derivative at the nozzle inlet plane (a_1), throat (a_3) and nozzle exit plane (a_7).

To facilitate the integration in the test rig, nozzles with fixed axial length are specified. The minimum length of the nozzle prescribed by the Method of Characteristics [19] to avoid Mach wave reflections along the supersonic section is a function of the required exit Mach number. This minimum length increases with onset Mach number. The overall length of the nozzle has been dictated by the minimum overall length for the highest operating point of the test rig, which is $L_{tot} = 911$ mm for an exit Mach = 3.0. The streamwise position of the control points was parametrized with the dimensionless quantities d_1 and d_2 . The axial position of the control points a_2, a_3, a_4, a_5 was linked with segments of equal length $d_1 L_{sup}$ to impose continuity in the second derivative, where L_{sub} is the length of the nozzle contraction (Fig. 3). This reduces the risk of unwanted pressure gradients in the nozzle contraction. Consequently, the length of the nozzle expansion section is $L_{sup} = L_t / (3d_1 + 1)$. Thus, the d_1 parameter influences the throat position within the nozzle and the aggressiveness of the nozzle contraction portion, whereas the d_2 parameter influences the contour of the nozzle expansion portion (Fig. 3).

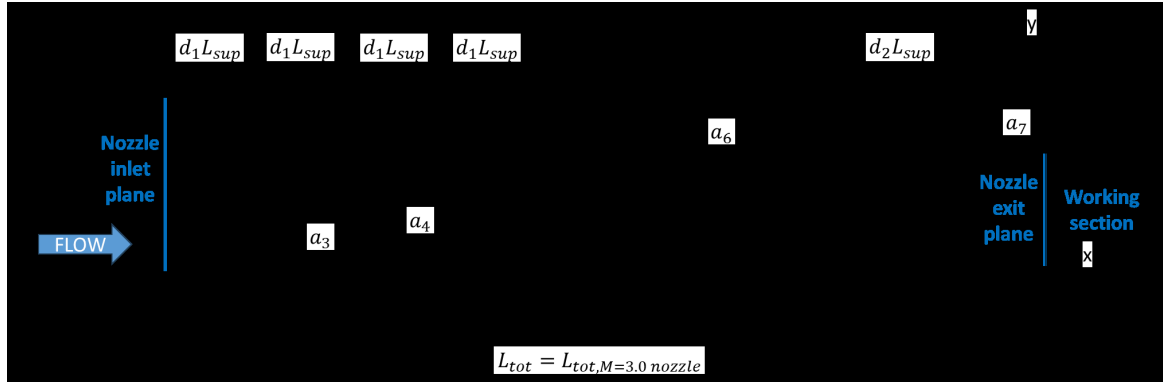


Fig. 3: Parametric geometry definition for the Mach 1.8 nozzle

Flow computations were initially conducted on a 2D, half nozzle model (Fig. 3) with extended inlet and outlet domains to favour convergence, which had an equivalent streamwise length of approximately $3H_{WS}$. Automated grid generation was developed for the parametric geometric definition to maintain approximately the same grid resolution. The selected meshing strategy generated grids with approximately 90,000 elements. Inflation layers were not included in the grid as the initial flow predictions were based on Euler simulations, where viscous effects were not considered [17]. Simulations were run using the compressible, pressure-based, steady-state Navier-Stokes solver in *ANSYS CFX 2021 R1* with a second order spatial discretization scheme. A pressure ratio of 3.5 between inlet and outlet domains, an inlet total temperature of 256 K, standard outlet ambient pressure, no slip walls, and Neumann conditions at the symmetry planes were set. Additional information on the simulation setup is shown by Moreno et. al [17].

The nozzle geometry optimization was driven by the axial distortion metric $\psi(x)$, which quantifies the fluctuations of the axial velocity (V_x) caused by Mach wave reflections at the working section [17]:

$$\psi(x) = f(d_1, d_2) = \max \left(\frac{\max \Delta V_x}{\langle V_x \rangle} \right)_x \quad (2)$$

Where $\max \Delta V_x$ is the maximum range of streamwise velocity at cross flow plane x normalized by the local mass-flow averaged streamwise velocity $\langle V_x \rangle$. The $d_1 \times d_2$ design space has been populated with a Hammersly algorithm [20] on initial bounds of $d_1 = [0.05, 0.35]$ and $d_2 = [0.40, 0.80]$ to promote the identification of designs with relatively short, aggressive contraction and smooth, long expansion portions as recommended by Pope and Goin [21]. Successively, the design space has been refined around the regions where the axial distortion metric $\psi(x)$ was found to be the lowest. The design space exploration identified two areas with two local minima (Fig. 4a). The first candidate (case 111, Fig. 5), was characterized by a shorter contraction than the second one (case 111, Fig. 5), however both designs produced a minimum axial distortion of approximately $\psi(x) \sim 0.50$ (Fig. 4b).

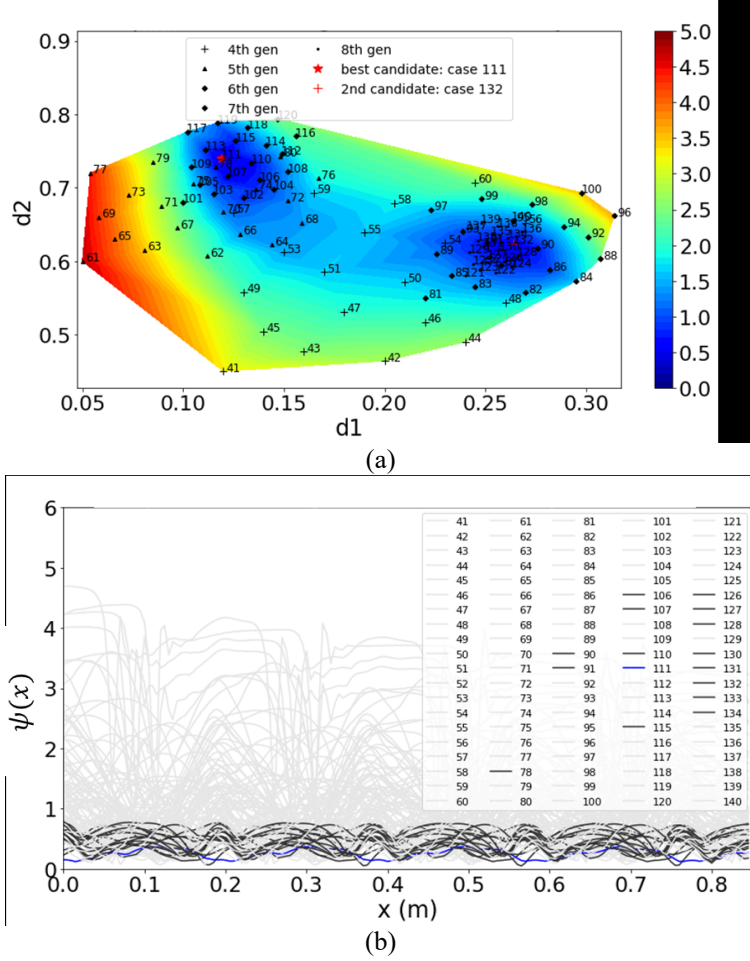


Fig. 4: a) distribution of the axial distortion parameter ψ in the $d_1 \times d_2$ design space, b) streamwise evolution of the axial distortion parameter ψ in the working section. $X=0$ denotes the nozzle exit plane.

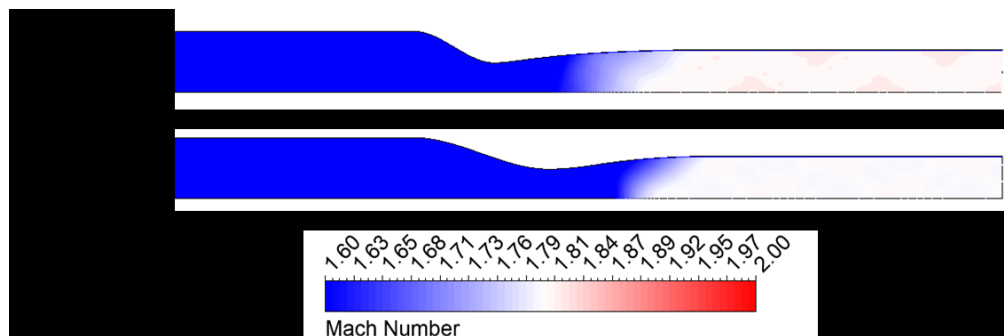


Fig. 5: Mach number distribution of the best design candidates for target $M = 1.8$.

The down selected designs were further analyzed with viscous simulations with second-order $k-\omega$ shear stress transport (SST) turbulence model in ANSYS CFX 2021 R1. Cross-sections were chosen to be circular for the settling chamber, and square for the working section, due to other considerations (see relevant section hereafter). Inflation layers were introduced in the mesh with Pointwise hybrid T-Rex mesh to maintain a $y^+ \leq 0.5$ and the working section domain has been extended to an equivalent length of $10H_{WS}$, which yielded a mesh size of about 52 million elements. Both nozzle designs generated similar levels of axial distortion, however the intrinsic density gradient generated by the expansion waves were substantially milder in case 111 compared to case 132 (Fig. 6), thus the first has been deemed most adequate.

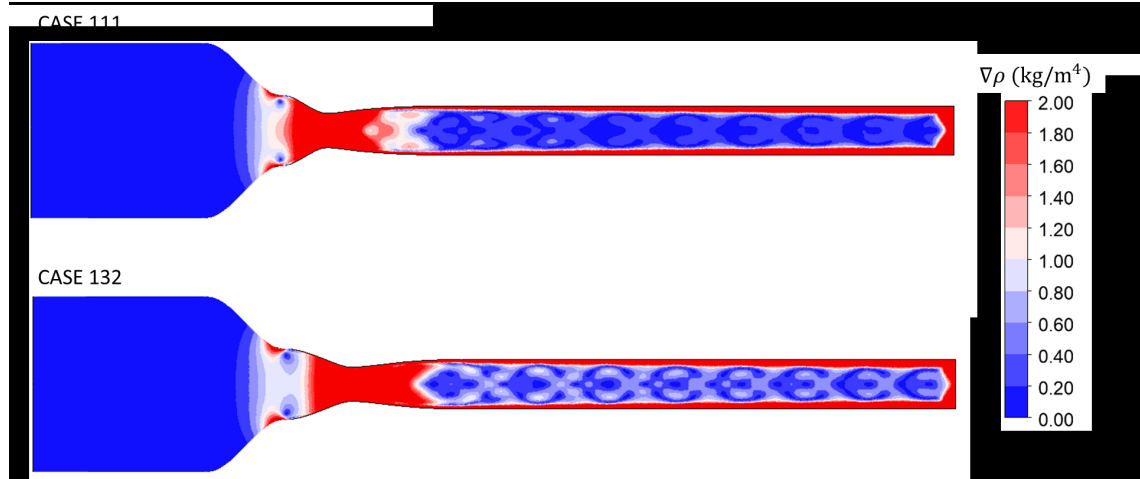


Fig. 6: Density gradient distribution for design candidates with lowest axial distortion.

The rig's settling chamber diameter was guided by the largest off-the-shelf available pipe diameter (DN900). The estimated average stream-wise flow speed within the settling chamber was approximately 12 m/s, which is aligned with flow straightener design requirements and enables the use of standard ducting size in compliance with pressure directive equipment [21,22]. The design of a pre-contraction between the settling chamber and the nozzle was undertaken, and several candidate combinations are possible in function of the selected cross-section geometry at the nozzle inlet plane, throat, and working section were identified. Various combinations of cross sections were examined including a) circular to rectangular section transition followed by a rectangular throat and a square working section (CRRS, Fig. 7a), b) circular to rectangular section transition followed by a square throat and a square working section (CRSS, Fig. 7b), and c) circular to square section transition, followed by a square throat and a square working section (CSSS, Fig. 7c). These geometries were evaluated based on the residual swirl angle delivered at the working section:

$$\alpha = \arctan\left(\frac{V_\theta}{V_z}\right) \quad (3)$$

Where V_θ is the tangential velocity relative to the working section centerline and V_z is the streamwise velocity, with α assumed positive in the counter-clockwise direction.

Computational models considering viscous flow with second-order $k-\omega$ shear stress transport (SST) turbulence model have been developed. The evaluation of inherent generation of swirling flow has evidenced that a transition between square throat and square working section in CRSS and CSSS configurations produces a pair of swirl cells with opposite sign on all sides of the working section (Fig. 7b, c). On the other hand, the transition between rectangular throat and square working section in the CRRS configuration generates swirl cells exclusively on the lateral working section walls, perpendicular to the expansion direction (Fig. 7a). Another relevant consideration is that the swirl distortion is primarily influenced by the geometrical transition in the nozzle expansion section, whereas it is uninfluenced by geometrical transition in the pre-contraction and nozzle contraction. Indeed, qualitatively, CRSS and CSSS configurations present similar swirl cell distribution near the throat and at the working section although they embed different section transitions in the contractions (Fig. 7b, c). Ease of manufacturing and tolerance control was also a key consideration. A rectangular nozzle such as in CRRS can be achieved with machining of shaped plates for upper and lower wall and flat side panels, which are later assembled together. In this case, geometrical uncertainties are primarily linked with the contour of the upper and lower wall. On the other hand, the manufacturing of a square nozzle is more complex as it may require machining shapes from a solid block and tool access may be challenging for small throats for high Mach numbers. Uncertainties are also linked with machining tolerances on all four walls. Due to these considerations, the CRRS configuration were selected. This includes a circular settling chamber, followed by a rectangular nozzle inlet plane, a rectangular nozzle throat, and terminates in a square working section.

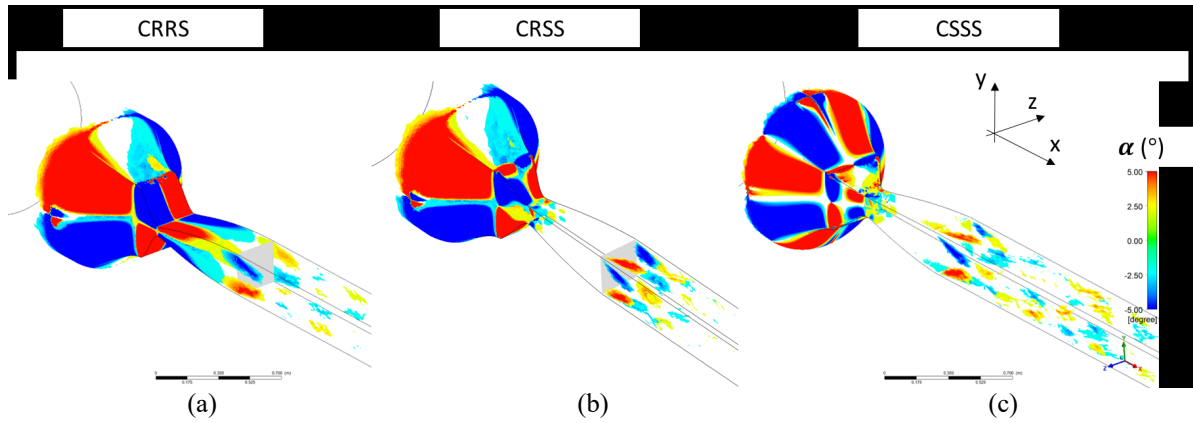


Fig. 7: Volumetric swirl angle distribution for different geometry transition between settling chamber, nozzle inlet plane, throat, and working section, respectively: CRRS (circular, rectangular, rectangular, square), CRSS (circular, rectangular, square, square), and CSSS (circular, square, square, square). Swirl distributions are capped at values exceeding ± 0.5 deg.

As the experimental rig is required to operate in a range of Mach number between 1.4 - 3.0, the nozzle must embed different throat heights to deliver the required exit Mach number. Although the design of variable throat nozzles is possible [23], for simplicity it was chosen was to manufacture different nozzles to achieve the different target Mach numbers. The pre-contraction and nozzle have been designed and manufactured as separate parts, so that the same pre-contraction can be assembled with different nozzles, for cost-effectiveness. The manufacturing of a pre-contraction and a Mach 1.8 nozzle is now complete (Fig. 8).

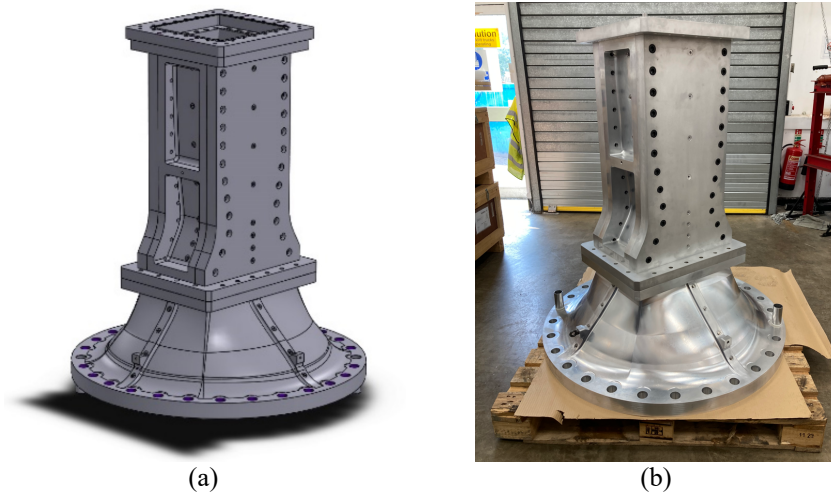


Fig. 8: a) CAD model of the pre-contraction and Mach 1.8 nozzle and b) picture of the pre-contraction and Mach 1.8 nozzle on a pallet after manufacturing.

The boundary layer development along the nozzle walls has a direct impact on the Mach number delivery in the working section and its evolution in the working section. The nozzle design method presented in Section III assumed isentropic, inviscid flow. However, the presence of viscous boundary layers, reduces the effective throat to exit area ratio in the nozzle, which produces a reduced Mach number in the working section. 3D viscous simulations predicted a freestream Mach number of $M_{Fr,3D\,CFD} = 1.778$ at the nozzle exit plane ($x=0$, Fig. 9), which is lower than the target delivery Mach number of $M_{Fr,target} = 1.800$. This discrepancy between target and achieved Mach number is further accentuated in the working section (Fig. 9). Indeed, in a working section with parallel walls, the Mach number decays as the distance from the nozzle exit plane increases due to the growth of the boundary layer, as visible in the distribution of the density gradients (Fig. 6).

Adjustment of the delivery Mach number in the working section was conducted in two steps. First, the nozzle throat cross section was adjusted to deliver $M_{Fr} = 1.8$ at the nozzle exit plane. Then, a corrected throat area A_{TH} was computed:

$$A_{TH} = f(M_{Fr,target} + \Delta M) \quad (4)$$

$$\Delta M = M_{Fr,target} - M_{Fr,3D\ CFD} \quad (5)$$

This yielded a target Mach number at the nozzle exit within 0.001 (Throat correction case, Fig. 9). Secondly, a strategy for minimizing stream-wise variation of Mach number was developed. To counteract the growth of the boundary layers, the cross-sectional area of the working section was gradually increased using tapered upper and lower walls (Fig. 10). The taper angle θ_T was evaluated by assessing the reduction of the working section M_{Fr} at a sufficiently high distance from the working section, at $x = 5H_{WS}$ and increasing the working section area by the same amount:

$$\Delta M_{Fr,\%} = (M_{Fr,x=0} - M_{Fr,x=5H_{WS}})/M_{Fr,x=0} = 3.32\% \quad (6)$$

$$\Delta M_{Fr,\%} = \Delta A_{Fr,\%} = (A_{Fr,x=0} - A_{Fr,x=5H_{WS}})/A_{Fr,x=0} = \Delta M_{Fr,\%} \quad (7)$$

$$A_{Fr,x=5H_{WS}} = A_{Fr,x=0} + \Delta A_{Fr,\%} \quad (8)$$

$$H_{WS,x=5H_{WS}} = A_{Fr,x=5H_{WS}}/W_{WS} \quad (9)$$

$$\delta_T = \text{atan}\left(\frac{H_{WS,x=5H_{WS}} - H_{WS,x=0}}{x_{5H_{WS}} - x_0}\right) = 0.19^\circ \quad (10)$$

Where subscripts $x = 0$ and $x = 5H_{WS}$ indicate quantities at the nozzle exit plane and $5H_{WS}$ downstream of the nozzle exit plane respectively, and W_{WS} is the width of the working section cross-section which is kept constant at 10 inches. This enabled a constant Mach number in the working section (Throat correction with tapered WS walls, Fig. 9).

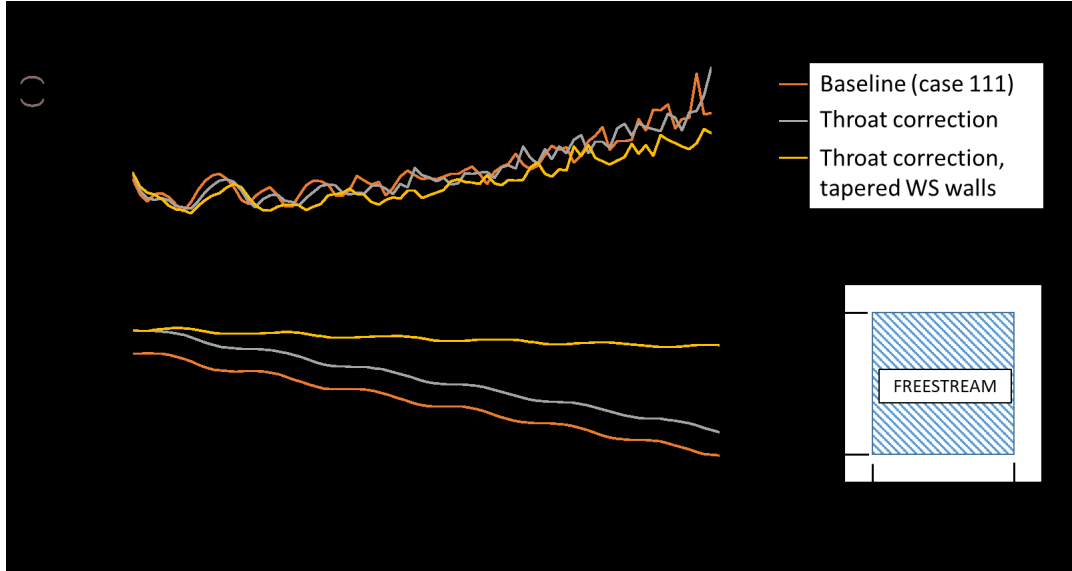


Fig. 9: Comparison of the evolution of the axial distortion parameter $\psi(x)$ and freestream Mach number M_{Fr} in the working section, where $X=0$ represents the nozzle exit plane. Quantities were calculated considering at cross-flow planes in a 7×7 in² freestream region that represents the core flow outside of boundary layers.

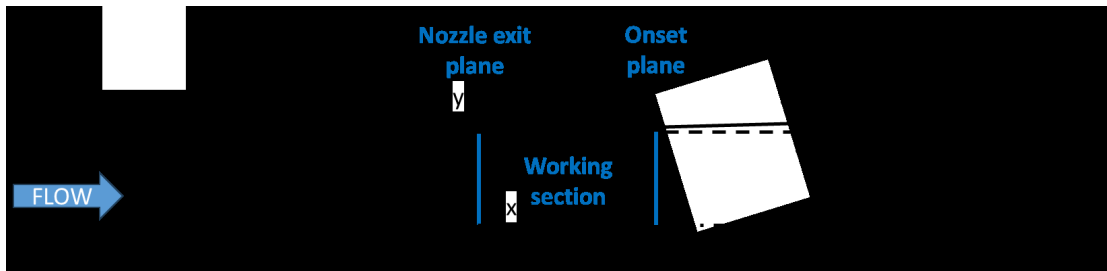


Fig. 10: Schematic of the working section with tapered upper and lower walls with angle δ_T .

An assessment of the flow angularity in the working section with tapered walls was also carried out. 3D CFD simulations showed flow angularity caused by the residual motion of the flow that is expanding towards the upper

and lower wall. Indeed, velocity vectors in the freestream area show an in-plane velocity component moving radially towards the walls (Fig. 11a). The streamwise flow angularity reduces gradually as the distance from the nozzle exit plane increases (Fig. 11b-e) until it becomes $\alpha < 0.4^\circ$ at the onset plane ($x_{0+H_{WS}}$, Fig. 11f). A pair of vortical structures is visible at each side wall of the working section, as expected due to the geometrical configuration CRRS presented in Fig. 7a. Although these structures persist throughout the working section until the nominal onset plane, these extend only within the boundary layers and do not appear to affect the flow angularity of the core flow ($x_{0+H_{WS}}$, Fig. 11f). Flow angularity surveys will be conducted during the rig commissioning phase to verify that the nozzle can deliver near swirl-free supersonic onset flow at prescribed operating point.

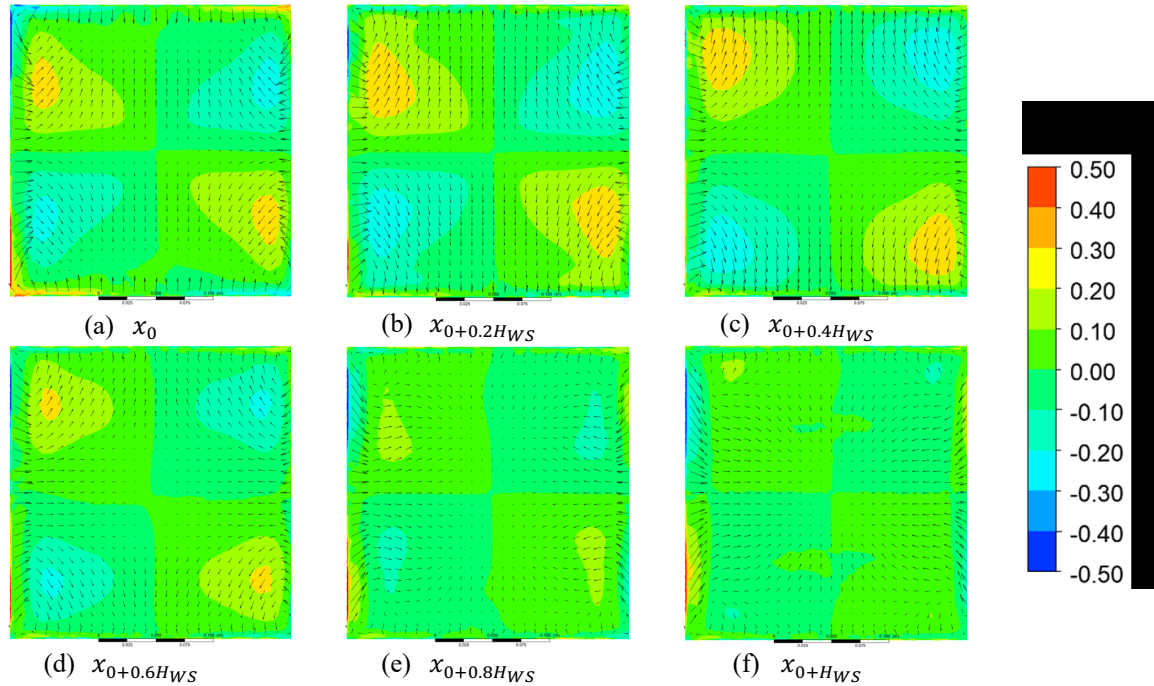


Fig. 11: Development of the flow angularity (swirl angle) between the nozzle exit plane (x_0).

IV. Working section design characteristics

A key characteristic of this test rig is optical access for the application of non-intrusive measurement methods to assess the influence of supersonic onset conditions to the inherent flow distortion of complex air induction systems. This is enabled by the design of the working section with optical windows. In addition, optical access was a design driver that influenced the sizing of the working section. Previous work showed that a square working section of 10x10 in² could accommodate test models with up to 4.5-inch equivalent AIP diameter across most of their operating mass flow ratio range. Further increase of the test section into a larger 18x18 in² was found able to ensure sufficient rig staring capability and to mitigate part of the interference of the rig walls with the aerodynamics of the intake entry [24].

A schematic indicating a representative test article installed within the working section is shown in Fig. 12. The proposed design of the working section offers a flexible integration of the test intake to enable spillage flow without significant wall interference. Downstream of the nozzle, a 10 x 10 in² working section with tapered walls leads the flow to the test article. The four working section walls are retained by guide rails. As recommended by previous work, the flow expands into an 18 x 18 in² cross section in the vicinity of the intake cowl leading edge (Fig. 12). This is achieved with an outer shell on which the 10 x 10 in² wall guiderails are installed (Fig. 13). Previous work also highlighted how the relative distance between the transition between 10- and 18-inch cross section influences the cowl aerodynamics. Thus, the working section is designed to allow flexibility in the length of the working section by modifying the length of the 10-inch cross section walls and thus the relative distance to the intake. Flexibility in accommodating test intakes with single or double plane diffuser offset is offered through the back wall of the radial exhaust. A blank retained by a v-band clamp allows azimuthal repositioning in steps of 5 degrees to accommodate test intakes with combinations of multiple degrees of vertical and lateral offsets.

All four walls of the 10-inch working section will be manufactured with clear acrylic material to enable optical access for the aerodynamic characterization of the intake ramp and cowl. The 18-inch outer shell also include windows on all four walls, which makes the working section completely transparent from four directions. This provides scope to integrate various non-intrusive techniques including Particle Image Velocimetry (PIV), shadowgraph (standard Schlieren and Background Oriented Schlieren, BOS) and pressure sensitive paints.

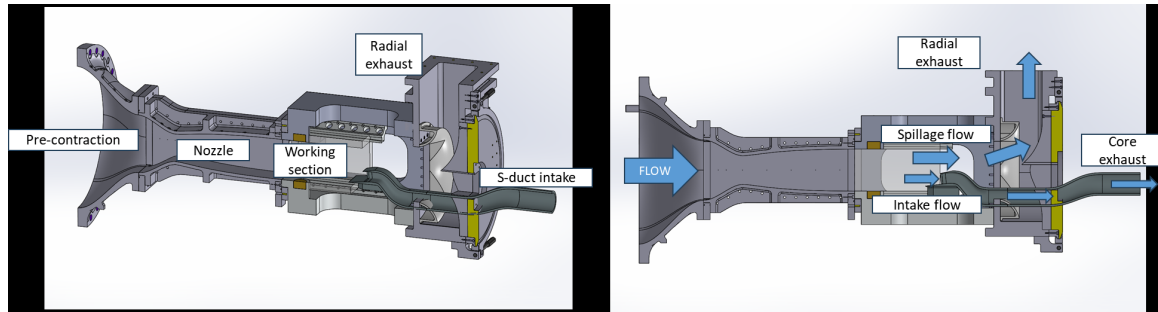


Fig. 12: Outline of the working section assembly with indication of main component and flow path.

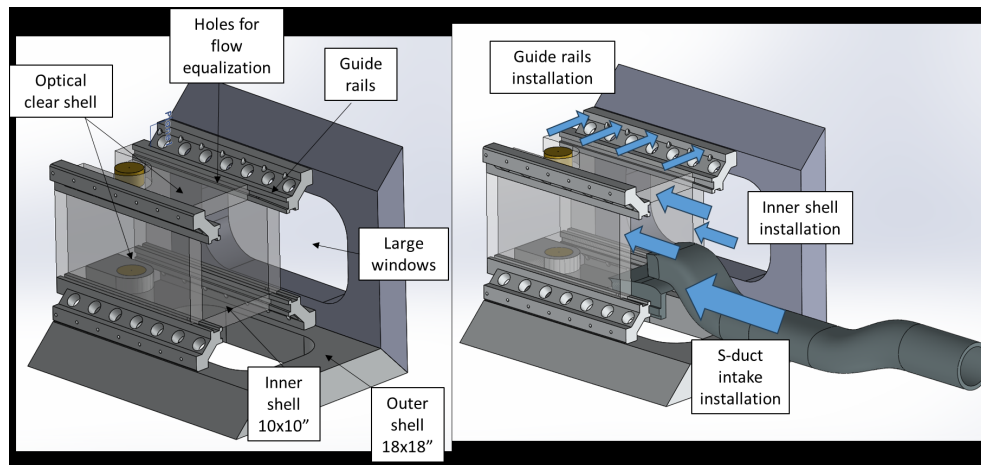


Fig. 13: Design characteristics for the working section shells, guiderails, and integration of an S-duct intake.

V. Industry-standard unsteady total pressure distortion measurement capability

Recommended practices for assessing intake flow distortion include the measurement of total pressure, swirl and, if relevant, total temperature distribution at the aerodynamic interface plane (AIP) [25–28]. A novel aspect of the current rig is capability to measure synchronous total pressure and swirl distortion at the AIP of the test article. In the configuration presented in Fig. 2, the AIP is located externally to the working section, which enables access for both optical and standard instrumentation. Optical access is provided via an acrylic section upstream the AIP plane (Fig. 14a) for time-resolved particle image velocimetry measurements. These will provide unsteady measurements of 3D velocities across the full AIP plane with an expected spatial resolution of $0.015D_{AIP}$. These measurement will provide the characterization of flow modes, unsteady swirl distortion metrics and also insights on the unsteady distortion perceived from rotating compressor blades [13–15,29].

These measurements will be complemented by industry-standard, unsteady total pressure measurements with 40 transducers arranged in 5 rings and 8 rake configuration as prescribed in the ARP1420 [25]. The measurement section is sized at 4.5 inches; however, it is anticipated that additional ducting will be manufactured to adapt to a range of intake AIP between 4.5-6 inches. This consideration is also crucial for the unsteady rake as the position of the probes would need to adapt to equal-area arrangement for 4.5-6 inches AIP test models. To enable the spatial re-configuration of the high-speed pressure transducers, to reduce the risk of damaging the sensors in the process, and to replace them easily in case of failure, a novel arrangement of the rakes was designed (Fig. 14b). The sensors are housed in a sleeve, which is secured on the rake with a bayonet fitting. The bayonet fitting has a threaded connection on the rake strut. The sleeve and the rake strut have bored holes to pass-through the four

sensor wires outside the duct for connection to the amplifiers. A special four-pin nano-connector enables the disconnection of the sensor sleeve from the rake strut. This allows the use of the same sensors and associated sleeves on rakes with a different spatial arrangement of the probes and size and makes this suitable for reconfiguration in the 4.5-6 inches AIP range. The use of the sleeve reduces the risk of damaging the sensor and the risk of false electrical contacts during re-configuration. The bayonet fitting pushes the sleeve against the rake strut and offers structural rigidity to the assembly as well as a robust connection between the connectors. Due to the relative complexity and novelty of the assembly, a trial assembly was fabricated ahead of the final commissioning. The trial has successfully proven the integrity of the probes, the manufacturing capability and the feasibility of the concept (Fig. 15). The unsteady total pressure rake and time-resolved PIV (TR-PIV) capability will unlock fundamental knowledge on the correlation between swirl and total pressure distortion of the intake, which is an acknowledged research gap [26].

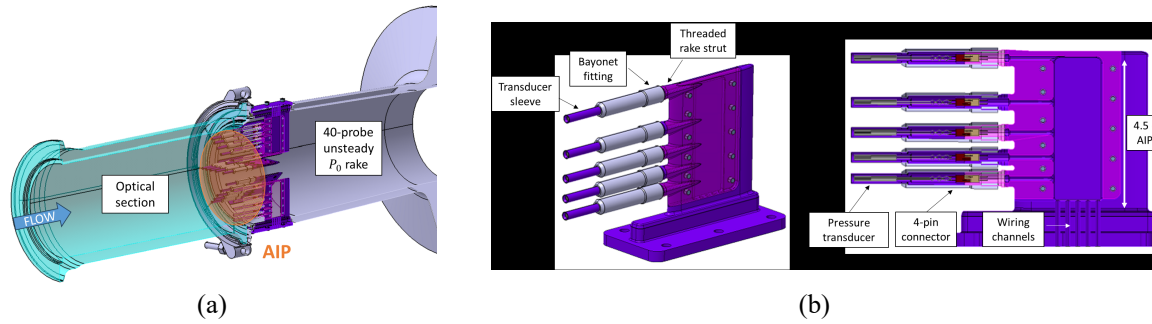


Fig. 14: 3D representation of the measurement instrumentation at the AIP: a) overview of the full AIP section detailing the optical access, unsteady rake and AIP plane, b) integration of the pressure transducer sleeve in the rake through removable bayonet-type fittings.

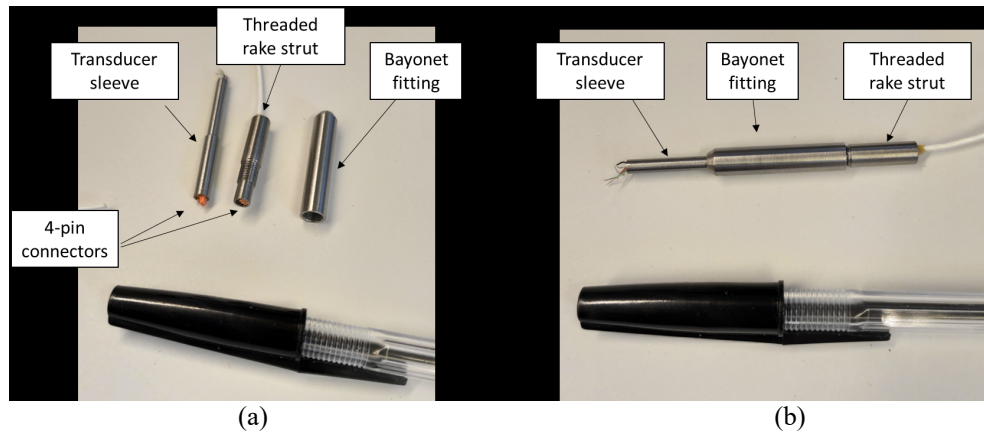


Fig. 15: Manufacturing of a trial rake for assembly testing showing a) decoupled and b) integrated assembly. Pen included for scale comparison.

The unsteady rake section is required to operate in choke-free conditions up to a streamwise Mach number of 0.6 for AIP ranging between 4.5-6 inches diameter. Thus, the choking effect of the unsteady rake was assessed for the most challenging configuration for area blockage (4.5-inch AIP). The design of the unsteady rake was aimed at minimizing the frontal area of the rake, which depends on the frontal area of the probes and the thickness and length of the struts. To offer additional headroom to avoid choking, the internal diameter of the duct was increased with a conical expansion in correspondence of the rake struts. The critical area ratio A^*/A is evaluated with the compressible isentropic flow equations ($\gamma = 1.4$ and $M_{AIP} = 0.6$):

$$\frac{A}{A^*} = \frac{\left\{ \left[1 + M^2 * \frac{\gamma - 1}{2} \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \right\}}{M} \left(\frac{\gamma + 1}{2} \right)^{\frac{-\gamma + 1}{2(\gamma - 1)}} = 1.1882 \quad (11)$$

The choking margin was estimated with:

$$CM = \frac{A^*}{A_{open}} - 1 \quad (12)$$

Where A^* is the critical area for choking and A_{open} is the difference between the full duct area (A_{duct}) and the frontal area of the rakes (A_{rakes}). The minimum choking margin CM at $M_{AIP} = 0.6$ was maintained at 7% in the design. The selection of the unsteady pressure transducers, signal conditioning and data acquisition system was aimed at minimizing the total uncertainty in measurements of total pressure at the AIP for typical operating points of Mach 1.6 and 2.0 at the onset plane (10 inches down-stream the nozzle exit plane, Fig. 10). The selected configuration comprised 40 unsteady pressure transducers, a bridge amplifier, and a data acquisition interface (DAQ). The calculation of uncertainty in pressure measurements considered the error contributions of the DAQ resolution and typical accuracy, and the transducer typical error (best straight fit line):

$$\sigma_{p_{0AIP,TOT}} = \sqrt{\sigma_{p_{0AIP,res}}^2 + \sigma_{p_{0AIP,XCQ}}^2 + \sigma_{p_{0AIP,DAQ}}^2} \quad (13)$$

$$\sigma_{p_{0AIP,res}} = R_{DAQ(V)} \frac{FSO_{(Pa)}}{FSO_{(V)}} \quad (14)$$

$$\sigma_{p_{0AIP}} = A_{trans} FSO_{(Pa)} \quad (15)$$

$$\sigma_{p_{0DAQ}} = A_{DAQ} FSO_{DAQ(V)} \quad (16)$$

Where R_{DAQ} is the resolution of the DAQ in V, $FSO_{(Pa)}$ is the transducer range in Pa, $FSO_{(V)}$ is the amplified signal range in V, A_{XCQ} is the typical accuracy BSFL of the transducer on $FSO_{(V)}$ range, A_{DAQ} the typical calibrated accuracy of the DAQ including gain and offset error at $23^\circ\text{C} \pm 5^\circ\text{C}$, and $FSO_{DAQ(V)}$ is the range of the DAQ card. The deterministic, overall measurement uncertainty in total pressure measurements at the AIP is 0.90% and 0.72% respectively for the operating points $M = 1.6$ and 2.0 (Table 1).

Table 1: Expected uncertainty levels for total pressure measurement at the AIP for rig operating point of M=1.6 and 2.0 for an AIP Mach number of 0.6.

Operating point (Mach)	1.6	2.0
DAQ card resolution R_{DAQ} (μV)	159.59	
Transducer range $FSO_{(Pa)}$ (Pa)	413,685	
Amplified signal range $FSO_{(V)}$ (V)	10	
Transducer error BSFL A_{trans} ($\%FSO_{(V)}$)	0.1%	
DAQ calibrated range $FSO_{DAQ(V)}$ (V)	10.5	
DAQ accuracy A_{DAQ} ($\%FSO_{DAQ(V)}$)	0.021%	
DAQ resolution R (μV)	159.59	
DAQ resolution uncertainty $\sigma_{p_{0AIP,res}}$ (Pa)	6.3	
DAQ accuracy uncertainty $\sigma_{p_{0DAQ}}$ (Pa)	91.2	
Transducer accuracy uncertainty $\sigma_{p_{0AIP,XCQ}}$ (Pa)	413.7	
Total uncertainty $\sigma_{p_{0AIP,TOT}}$ (Pa)	423.7	
Δp_{0AIP} (Pa)	46935	58668
Total uncertainty $\sigma_{p_{0AIP,TOT}}$ ($\%\Delta p_{0AIP}$)	0.90%	0.72%

VI. Test article integration

Due to the complexity of the proposed rig configuration, flow computations were carried out to derisk the starting ability of the tunnel and the ability of the radial exhaust to divert the bypass flow upwards (Fig. 2). A CFD model of the full rig comprising settling chamber, pre-contraction, Mach 1.8 nozzle, working section, radial collector and a typical test article was generated. The selected meshing strategy generated a hybrid T-rex grid with approximately 128 million elements with inflation layers for a target $y^+ \leq 0.5$. Simulations were conducted using the compressible, pressure-based, steady-state Navier-Stokes solver in *ANSYS CFX 2021 R1* with a second order spatial discretization scheme and a second-order $k-\omega$ shear stress transport (SST) turbulence model. An ambient pressure boundary condition was imposed at the exit plane of the radial exhaust. A pressure ratio of 3.5 was set between the inlet plane of the settling chamber and the exit plane of the radial exhaust. A pressure ratio of 2.5 was set between the inlet plane of the settling chamber and the intake exit plane. A total temperature of 256K was set at the settling chamber inlet plane. All walls were set to an adiabatic, no slip, boundary condition. Simulations were run without initialization to simulate a non-time-accurate surrogate of the flow acceleration process through iterations of the steady-state until convergence was reached.

Convergence was achieved with the nozzle fully started, indicating that the selected pressure ratio between the settling chamber and the radial exhaust was sufficient to overcome the pressure losses of the nozzles, working section and exhaust system (Fig. 16). Downstream of the onset plane, the presence of the intake, which operates at fixed flow rate, causes the flow to spill as expected at the top and around the sides of the intake ramp. Due to the selected sudden expansion provided by the working section steps, limited interaction of the bypass flow with the working section outer walls is present. Shock-wave reflections are not present in the bypass flow stream (Fig. 16). The bypass flow impinges on the rear face of the radial exhaust and then is diverted upwards with a sharp bend. The Mach contours indicate that this is the likely position of a terminal shock which converts the supersonic bypass flow in the, mostly subsonic, exhaust stream. Most of the flow is expelled within 5ms or less, as shown by the residence time on streamlines seeded from the onset plane (Fig. 17). Flow re-circulations are mostly limited to lower regions of the plenum and in the pocket gaps between inner and outer shell walls of the working section where the flow is mostly subsonic.

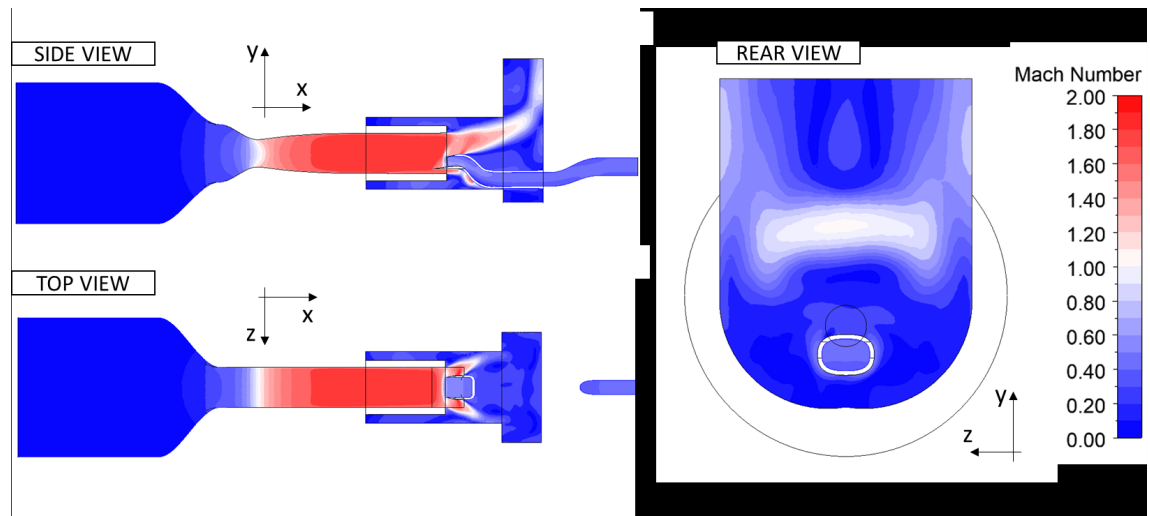


Fig. 16: CFD assessment of the Mach number distribution on the full rig configuration with Mach 1.8 nozzle.

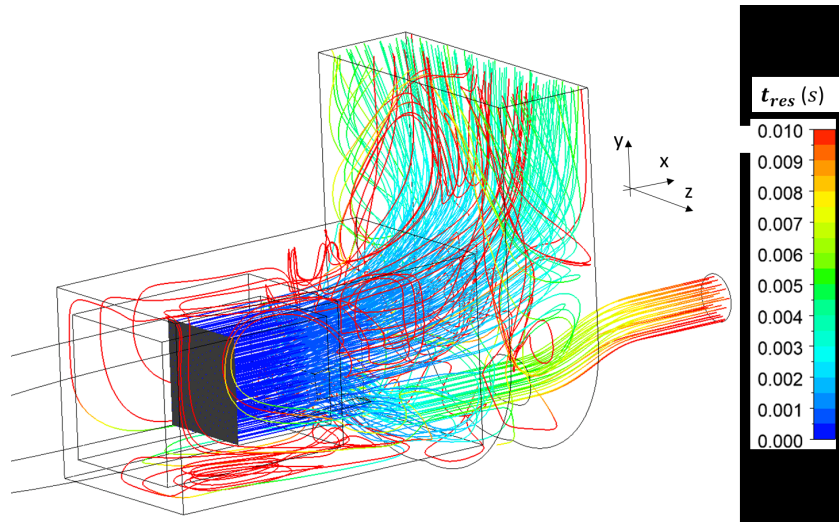


Fig. 17: Streamline tracers originating at the onset plane indicating resident time t_{res} in the working section, radial exhaust and intake.

Velocity vectors were extracted at different plane locations to provide an initial qualitative analysis of the flow distribution in the working section and radial exhaust. At the lowest vertical position (Fig. 18a), the flow mainly spills around the intake and generates a local recirculation area underneath the intake ramp and in the radial exhaust. In the horizontal plane above the intake (Fig. 18b), the spilled flow is directed towards the rear wall of the radial exhaust and diverted vertically. The recirculation vortices in the radial exhaust are diverted laterally away from the symmetry plane. Further above (Fig. 18c), most of the flow with high velocity magnitude resides in the radial exhaust, which is then deflected upwards towards the exit plane (Fig. 19a). Most of the flow within the pocket gap in-between the working section shells follows a vortical motion where the flow spilled laterally, and initially recirculated in the lower side of the radial exhaust. The flow rolls up in the side gaps and flows into the upper gap (Fig. 19b). This initial analysis will inform the design of inner liners aimed at facilitating the passage of the flow through the preferred routes limiting additional pressure losses and re-circulations which could influence the onset conditions.

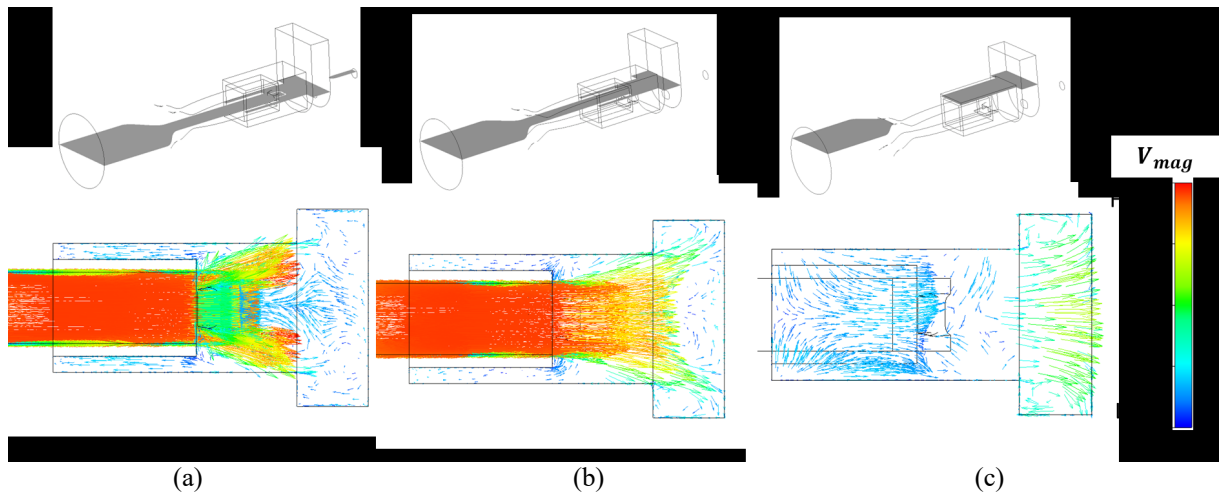


Fig. 18: Qualitative analysis of the flow distribution with velocity vectors on horizontal planes extracted at various vertical positions.

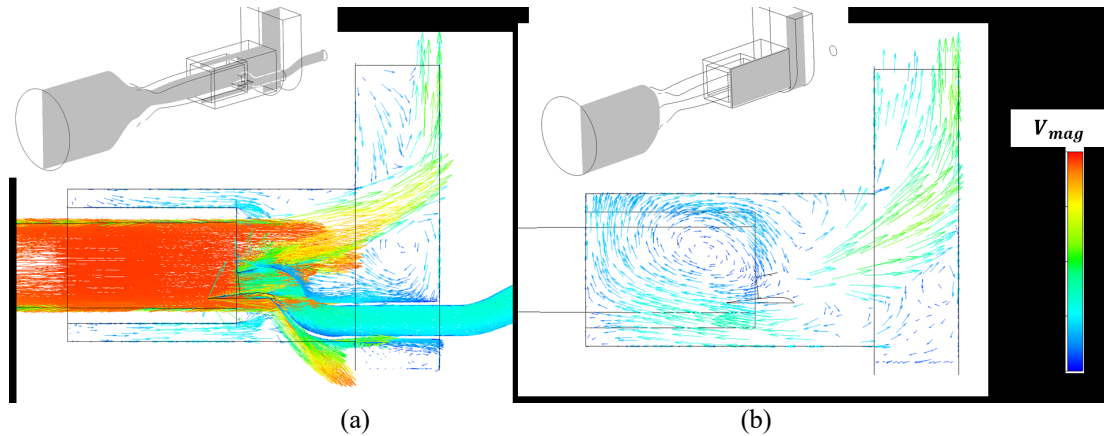


Fig. 19: Qualitative analysis of the flow distribution with velocity vectors on symmetry plane and side plane.

VII. Conclusions and Future Outlook

This work outlines the development of an experimental rig for high-speed air induction system research. The Mach 1.8 nozzle was optimized to minimize flow angularity in the working section. A detailed exploration of nozzle contour and pre-contraction geometry led to a design that ensures shock-free flow and limited angular distortion. Throat adjustments and tapered working section walls were incorporated to mitigate boundary layer and maintain a nearly constant Mach number between the nozzle exit and the inlet to the working section. The design prioritized optical access and flexibility to accommodate various intake geometries. Preliminary computational simulations of the full system, including the settling chamber, nozzle, working section, and radial exhaust, confirmed the robustness of the design. These simulations showed minimal recirculation and wall interaction, effective containment of the bypass flow within the radial exhaust and good starting ability of the test rig. Key innovative features of the design include a modular, unsteady total pressure rake and an optical section at the Aerodynamic Interface Plane (AIP) of the intake, enabling synchronous measurements of total pressure and swirl distortion. Reynolds numbers up to full scale will be possible for most applications. This capability will allow researchers to better understand the interactions between different flow distortion types, critical for reducing risks in intake-engine integration, an area that is still not fully understood. The next phase includes numerical studies on rig wall interferences, as well as the design of a mass flow control system to test intakes under a range of aerodynamic conditions, from sub-critical to critical operation. This facility will significantly contribute to the understanding of supersonic flow distortion and its impact on intake-engine integration for next-generation aircraft designs.

Acknowledgments

The authors would like to thank the Defence Science and Technology Laboratory (Dstl) for supporting this research programme.

References

- [1] Li, Z., Sun, D., Dong, X., and Sun, X., "A Review on Aero-Engine Inlet-Compressor Integration and Inlet Flow Distortion in Axial Compressors," *Fundamental Research*, 2024. <https://doi.org/10.1016/j.fmre.2024.03.018>
- [2] Kurzke, J., "Effects of Inlet Flow Distortion on the Performance of Aircraft Gas Turbines," *Journal of Engineering for Gas Turbines and Power*, Vol. 130, No. 4, 2008, p. 041201. <https://doi.org/10.1115/1.2901190>
- [3] Surber, L., and Robinson, C., "Survey of Inlet Development for Supersonic Aircraft," *19th Joint Propulsion Conference*, American Institute of Aeronautics and Astronautics, 1983. <https://doi.org/10.2514/6.1983-1164>
- [4] Bowditch, D. N., and Coltrin, R. E., "A Survey of Inlet/Engine Distortion Compatibility," NASA TM-83421, Cleveland, OH, June 1983.

- [5] Surber, L., and Tinapple, J., "Inlet Flow Control Technology: Learning from History, Reinventing the Future," *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, American Institute of Aeronautics and Astronautics, 2012. <https://doi.org/10.2514/6.2012-12>
- [6] Titchener, N., "An Experimental Investigation of Flow Control for Supersonic Inlets," 2013. <https://doi.org/10.17863/CAM.14077>
- [7] Kimzey, W. F., Ellis, S. H., Kimzey, W. F., and Ellis, S. H., "Supersonic Inlet Simulator - A Tool for Simulation of Realistic Engine Entry Flow Conditions," presented at the Aerospace Engineering and Manufacturing Meeting, 1974. <https://doi.org/10.4271/740824>
- [8] Surber, L., Syberg, J., and Koncsek, J., "PERFORMANCE OF HIGHLY INTEGRATED INLETS FOR SUPERSONIC AIRCRAFT," presented at the Fluid Dynamics Panel Symposium held at the Ecole Nationale Supérieure de l'Aéronautique et de l'Espace, Toulouse, France, 1981.
- [9] Gnani, F., "Investigation on Supersonic High-Speed Internal Flows and the Tools to Study Their Interactions," PhD. University of Glasgow, 2018.
- [10] Fisher, S. A., "Internal Performance of a Variable Ramp Mixed Compression Intake at Mach 3.05,"
- [11] Migliorini, M., Szymanski, A., Zachos, P. K., MacManus, D., and Martin, P. G., "Design of a High-Speed Intake Distortion Simulator for Propulsion Integration Research," *AIAA SCITECH 2023 Forum*, American Institute of Aeronautics and Astronautics, 2023. <https://doi.org/10.2514/6.2023-2356>
- [12] Pope, A., and Goin, K. L., "High Speed Wind Tunnel Testing," Wiley, 1965.
- [13] Zachos, P. K., MacManus, D. G., Prieto, D. G., and Chiereghin, N., "Flow Distortion Measurements in Convolved Aeroengine Intakes," *AIAA Journal*, Vol. 54, No. 9, 2016, pp. 2819–2832. <https://doi.org/10.2514/1.J054904>
- [14] Gil-Prieto, D., Zachos, P. K., MacManus, D. G., and McLelland, G., "Unsteady Characteristics of S-Duct Intake Flow Distortion," *Aerospace Science and Technology*, Vol. 84, No. 1, 2019, pp. 938–952. <https://doi.org/10.1016/j.ast.2018.10.020>
- [15] Migliorini, M., Zachos, P. K., MacManus, D. G., and Haladuda, P., "S-Duct Flow Distortion with Non-Uniform Inlet Conditions," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 2022, p. 095441002211016. <https://doi.org/10.1177/09544100221101669>
- [16] Doll, U., Migliorini, M., Baikié, J., Zachos, P. K., Röhle, I., Melnikov, S., Steinbock, J., Dues, M., Kapulla, R., MacManus, D. G., and Lawson, N. J., "Non-Intrusive Flow Diagnostics for Unsteady Inlet Flow Distortion Measurements in Novel Aircraft Architectures," *Progress in Aerospace Sciences*, Vol. 130, 2022, p. 100810. <https://doi.org/10.1016/j.paerosci.2022.100810>
- [17] Moreno, M., Migliorini, M., Zachos, P. K., Haslam, A., and MacManus, D., "Design and Optimisation of a Mach 2.5 Wind Tunnel Nozzle," *AIAA SCITECH 2023 Forum*, American Institute of Aeronautics and Astronautics, 2023. <https://doi.org/10.2514/6.2023-1468>
- [18] Prautzsch, H., Boehm, W., and Paluszny, M., "B-Spline Representation," *Bézier and B-Spline Techniques*, edited by H. Prautzsch, W. Boehm, and M. Paluszny, Springer, Berlin, Heidelberg, 2002, pp. 59–75. https://doi.org/10.1007/978-3-662-04919-8_5
- [19] Anderson, J. D., "Modern Compressible Flow: With Historical Perspective," McGraw-Hill, Boston, 2003.
- [20] "Comparing Initial Sampling Methods — Scikit-Optimize 0.8.1 Documentation." Retrieved 1 November 2024. https://scikit-optimize.github.io/stable/auto_examples/sampler/initial-sampling-method.html
- [21] Pope, A., and Goin, K. L., "High-Speed Wind Tunnel Testing," R. E. Krieger Publishing Company, 1978.
- [22] G. Paniagua, J. S., "Flow Conditioning of Subsonic and Supersonic Blowdown Wind Tunnels," Manchester, UK, 2017.
- [23] Kenney, J. T., and Webb, L. M., "A Summary of the Techniques of Variable Mach Number Supersonic Wind Tunnel Nozzle Design," AGARD-AG-3, October 1954, p. 148.
- [24] Zachos, P. K., Haslam, A., MacManus, D. G., Migliorini, M., and Szymanski, A., "Development of a High-Speed Air Intake Distortion Simulation Capability (HI-Sim) - WP3.1 Deliverable Report D3.1.1/2 – HI Sim Test Rig Detailed Design and Specification," Cranfield University, June 2022, p. 52.
- [25] "Gas Turbine Engine Inlet Flow Distortion Guidelines," Aerospace Recommended Practice ARP1420C, Society of Automotive Engineers, Warrendale, PA, 2017.
- [26] "A Methodology for Assessing Inlet Swirl Distortion," Aerospace Information Report AIR 5686, Society of Automotive Engineers, Warrendale, PA, 2017. <https://doi.org/10.4271/AIR5686>
- [27] "Inlet Total-Pressure-Distortion Considerations for Gas-Turbine Engines," Aerospace Information Report AIR 1419A, Society of Automotive Engineers, 2017, p. 226.
- [28] "A Current Assessment of the Inlet/Engine Temperature Distortion Problem," Aerospace Resource Document ARD 50015, Society of Automotive Engineers, 1991.
- [29] Migliorini, M., Zachos, P. K., and MacManus, D. G., "Novel Method for Evaluating Intake Unsteady Flow Distortion," *Journal of Propulsion and Power*, 2021, pp. 1–13. <https://doi.org/10.2514/1.B38127>



DSPACE

<https://dspace.org/>

Aerodynamic design of a high-speed intake test rig for propulsion integration research

Migliorini, Matteo; Zachos, Pavlos K.; MacManus, David G.; Martin, Peter G.

2025-01-06

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2025-2110>

Migliorini M, Zachos PK, MacManus D, Martin PG. (2025) Aerodynamic design of a high-speed intake test rig for propulsion integration research. AIAA SCITECH 2025 Forum, 6-10 January 2025, Orlando, Florida, Paper number AIAA 2025-2110