

De-risking of a High-Speed Intake Distortion Facility using a Small-Scale Pilot Rig

Matteo Migliorini¹, Stijn Hersbach², Pavlos K Zachos³ and David G MacManus⁴

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

Peter G Martin⁵

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

This paper presents the development of a sub-scale pilot rig (Mini HI-Sim) designed to de-risk the operation of a large-scale, high-speed intake distortion facility (HI-Sim). The Mini HI-Sim replicates key aerodynamic features using a vacuum-driven aspirated configuration able to provide onset Mach numbers up to 1.65. This paper assesses the operating performance of the rig and compares it to estimates of operational runtime based on compressible flow theory and polytropic expansion. The study demonstrates that the runtime and flow uniformity are sensitive to vacuum level and speed of the fast-acting valve. Flow visualization using tuft tracking and image processing confirmed the presence of counter-rotating vortices in the radial exhaust which had been observed in previous computational work of the full-scale rig, offering a preliminary and qualitative validation of the radial exhaust design. Preliminary CFD analysis showed that re-pressurization of the working section was caused by an excessive geometric contraction in the exhaust, which increased the static pressure around the enclosed jet in the working section and prevented the normal shock from being fully displaced downstream. Removing this contraction yielded a uniformly low plenum pressure, improved flow extraction, and enabled the intended Mach-number distribution in the working section. Overall, the Mini HI-Sim offers a cost-effective platform for aerodynamic and instrumentation de-risking, supporting the development of robust supersonic intake systems for future propulsion applications.

Nomenclature

A	=	Cross-sectional area, m ²
AIP	=	Aerodynamic Interface Plane
M	=	Mach number
$\dot{m}$	=	mass flow rate, kg/s
n	=	polytropic coefficient, -
p	=	pressure, Pa or bar
R	=	Gas constant for air, J/(kg K)
Re	=	Reynolds number
T	=	temperature, K

¹ Senior Research Fellow, Centre for Propulsion and Thermal Power Engineering, AIAA Young Professional Member.

² MSc researcher, Centre for Propulsion and Thermal Power Engineering.

³ Reader in Propulsion Aerodynamics, Centre for Propulsion and Thermal Power Engineering, AIAA Senior Member.

⁴ Professor in Propulsion Aerodynamics, Centre for Propulsion and Thermal Power Engineering, AIAA Member.

⁵ Fellow, Defence Science and Technology Laboratory.

t = time, s
 U = streamwise velocity, m/s
 γ = specific heat ratio, -
 ρ = density, kg/m³

I. Introduction

Supersonic air induction systems are key components in high-speed propulsion systems, where complex and unsteady flow distortion can significantly impact engine operability and performance. As aircraft intake designs become more three-dimensional and integrated with the airframe, the number of their design variables increases, along with the uncertainty of their aerodynamic behaviour. This poses certain challenges on inlet-engine compatibility during the conceptual and preliminary design phases. The lack of early, high-fidelity distortion data can lead to overly conservative designs or costly late-stage redesigns, delaying development and increasing risk [1–3]. In an attempt to address these challenges early in the development cycle, a novel test rig capable of simulating high-speed intake distortion in a laboratory environment (HI-Sim) has been proposed and its development is currently in progress (Fig. 1) [4,5]. The HI-Sim test rig provides a novel research environment to de-risk the operation of supersonic intakes, including subcritical and near-critical operating points where phenomena such as shock oscillations and unstart are prevalent [6,7]. The facility is designed as a blowdown configuration, drawing compressed air from a 40 bar tank through a conditioning system into a supersonic nozzle to achieve freestream Mach numbers between 1.4 and 3.0 [8]. A working section, transitioning from 10"×10" to 18"×18" cross-sectional size allows integration of intake models while minimizing wall interference [4]. A novel radial exhaust design exposes the Aerodynamic Interface Plane of the test article, which enables the use of optical and laser-based diagnostics, such as Time-Resolved Particle Image Velocimetry, TR-PIV (Fig. 1) [5].

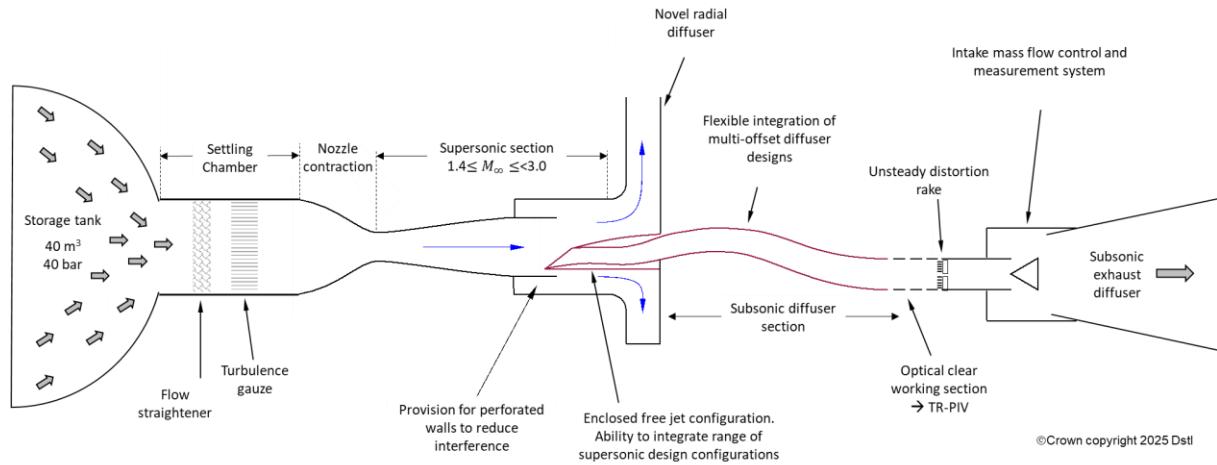


Fig. 1: Test article integration concepts in the HI-Sim intake distortion facility [5].

In parallel to the design of the HI-Sim facility, a pilot, test rig at about 1/3rd scale (Mini HI-Sim) has been developed, with the scope of de-risking the design, aerodynamics, instrumentation and operability aspects of the full-scale rig. A key aspect is the characterization of the interference between the intake test article and the working section walls, which operates as an “enclosed open-jet” configuration [4,5]. Secondly, the pilot rig validates the behaviour of the flow within the radial exhaust system that was previously designed using CFD methods [4,5]. Lastly, it is used as a vehicle to develop and de-risk the application of the advanced instrumentation, and develop flow visualization methods, including traditional Schlieren and Background-Oriented Schlieren (BOS), for later application on to the full-scale facility. This paper presents experimental work aimed at understanding the operation of the sub-scale rig, the distribution of the flow around a test article and within the radial exhaust, as well as the transient behaviour of the test rig during the acceleration and deceleration phases.

II. Overview of the small-scale test rig design operational characteristics

The Mini HI-Sim is as an intermittent in-draft rig configuration driven by a 4,000 litre air receiver that is initially pumped to near vacuum conditions (Fig. 2b). Air is drawn through a bell-mouth intake (Fig. 2a-1) and straightened with a honeycomb core panel installed at position 2 (Fig. 2a-2). Subsequently the flow is accelerated through a series of contractions and expanded in the main test section (Fig. 2a-3, 4), with a target Mach number of 1.6 at the test section centreline (Fig. 2a-5) provided by a fixed-throat convergent-divergent nozzle. The flow then enters the radial exhaust and ducting (Fig. 2a-6, 8). An AGARD-2 representative test model [9] is used as a surrogate of the test article in the full-scale rig (Fig. 2a, labelled “Bullet”).

Preliminary testing indicated that the rig test operation can be subdivided in 4 main phases (Fig. 3). During the pumping phase, the vacuum pump (Fig. 2b-7) is activated to reduce the pressure in the air receiver (Fig. 2b-5) from ambient conditions to near vacuum, typically 950mbar below ambient pressure conditions. When the target vacuum level is reached, the pump is stopped, and the main rig fast-acting valve is triggered (Fig. 2b-3). This sets the start of the acceleration phase (Fig. 3) in which airflow is drawn into the rig due to the differential pressure between ambient and the pressure in the tank. The nozzle throat Mach number increases until flow choking occurs and, provided that the ratio of stagnation pressure in the supply (approximately ambient) to static pressure at the exit of the test section (pressure ratio) is high enough, the supersonic flow is then established in the nozzle divergent section and test section ($M = 1.8$). In this condition the Mach number distribution through the nozzle and test section becomes independent of the down-stream conditions. This is denoted as steady operational phase with duration t_{OP} (Fig. 3). When the pressure ratio is not sufficient to maintain fully supersonic flow in the nozzle, the deceleration phase starts (Fig. 3). The test section and nozzle progressively un-start as a normal shock wave enters at the test section exit and moves upstream. Once the shock reaches the nozzle throat this becomes un-choked and, as pressure ratio falls further, the flow velocity through the system gradually reduces to zero and the receiver returns to ambient pressure conditions.

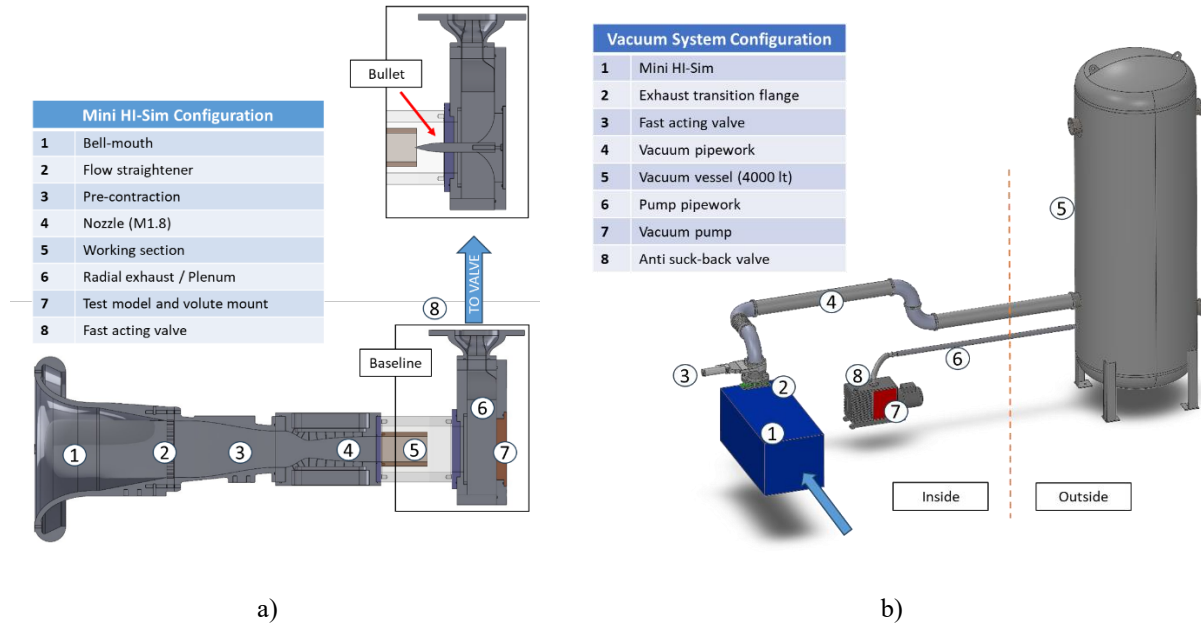


Fig. 2: General arrangement of the Mini HI-Sim pilot rig: a) main flow path, and b) suction system into vacuum vessel.

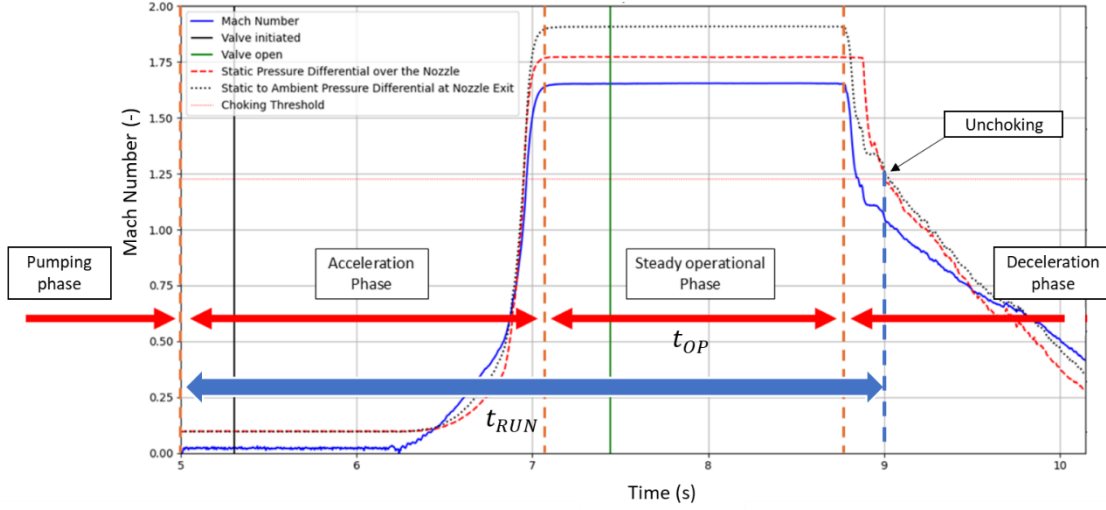


Fig. 3: Operational phases of the Mini HI-Sim test rig derived from pressure ratio measurements across the nozzle.

The duration of the run can be estimated by calculating the mass flow rate within a test section of area through compressible flow equations [8] (Eq. 1-3):

$$\dot{m} = \rho UA \quad (1)$$

$$\rho = \rho_t \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{1}{\gamma-1}} = \left(\frac{p_t}{RT_t} \right) \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{1}{\gamma-1}} \quad (2)$$

$$U = M \sqrt{\gamma RT} = M \sqrt{\gamma RT_t \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-1}} \quad (3)$$

Where subscript “t” refers to total conditions. The mass flow rate through the nozzle can be thus derived with (Eq. 4):

$$\dot{m} = \left(\sqrt{\frac{\gamma}{RT_t}} p_t A M \right) \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{\gamma+1}{2(\gamma-1)}} \quad (4)$$

The duration of the wind tunnel operation, t_{RUN} , can be estimated via the variation in density in the vacuum tank between the start of the run (ρ_i) and the end of the run (ρ_e) (Eq. 5), which is a function of the polytropic expansion coefficient n (Eq. 6) [8]:

$$\dot{m} t_{RUN} = V_{tank} (\rho_e - \rho_i) \quad (5)$$

$$\frac{\rho_i}{\rho_e} = \left(\frac{p_i}{p_e} \right)^{\frac{1}{n}} \quad (6)$$

The duration of the run can be thus estimated with Eq. 7, assuming that the volume of the vacuum tank is large enough that the temperature of the air inside the tank at the end of the run (T_e) corresponds to the temperature of air entering the nozzle (T_t):

$$t_{RUN} = \frac{V_{tank}}{MA\sqrt{\gamma RT_t}} \left(\frac{p_e}{p_t} \right) \left(1 - \left(\frac{p_i}{p_e} \right)^{\frac{1}{n}} \right) \left(1 + \frac{\gamma-1}{2} M^2 \right)^{\frac{\gamma+1}{2(\gamma-1)}} \quad (7)$$

The end-of-run pressure ratio (p_e/p_t) can be estimated as presented by Pope and Goin [8] based on previous supersonic wind tunnel data. For the design Mach number of 1.8 of the rig, this can be assumed as 1.3. Similarly, from previous wind tunnel data, the polytropic expansion coefficient n was assumed as 1.15. The operational phase from start to end of the run can then be estimated as a function of the Mach number for the given conditions and reported in Table 1 across a range of onset Mach numbers, considering a fixed initial vacuum pressure in the tank, p_i , of 950 mbar below ambient. It is interesting to note that the variation of the runtime is not linear with Mach number, as is dominated by the two pressure ratios p_e/p_t and p_i/p_e which are inversely proportional (Table 1). The operation at higher Mach numbers is penalized by the higher required end-of-the-run pressure ratio [8], but favoured by the lower mass flow requirement due to the higher expansion ratio of the air in the test section. The situation is opposite for low Mach numbers. For this reason, the operation at $M = 1.8$ is the one that offers the minimum run time.

It is highlighted that the measured Mach number in the test section does not coincide exactly with the design Mach number of the nozzle ($M = 1.8$), but rather a Mach number of $M = 1.6$ has been measured at the nozzle exit plane. This discrepancy discussed in section III. For this reason, it has been chosen to compare the analytical estimate derived for $M = 1.6$, for which the estimated runtime is approximately 3.3s. From the experimental observations, the runtime from the start of acceleration phase ($t_i \sim 6.3$ s, see Fig. 3) and the presence of a normal shock at the end of the nozzle ($M = 1$, $t_e \sim 9.0$ s) is approximately 2.7s, which is shorter than the analytical estimate. This can be attributed to a combination of factors, including the assumptions of the polytropic coefficient and the end-of-the-run pressure ratio, the operational parameters, discrepancies on the effective test area due to the presence of boundary layers, and the unaccounted pressure losses introduced by the flow straightener and area variations. The analytical model can be retrospectively tuned for better alignment with experimental observations. Likely, it would be more appropriate to use a slightly higher polytropic coefficient around $n = 1.2$ and a more conservative minimum pressure ratio for the tunnel of around 1.45 for $M = 1.6$.

Table 1: Estimated operational runtime of the sub-scale wind tunnel as function of Mach number in the test section, assuming a polytropic expansion coefficient $n = 1.15$.

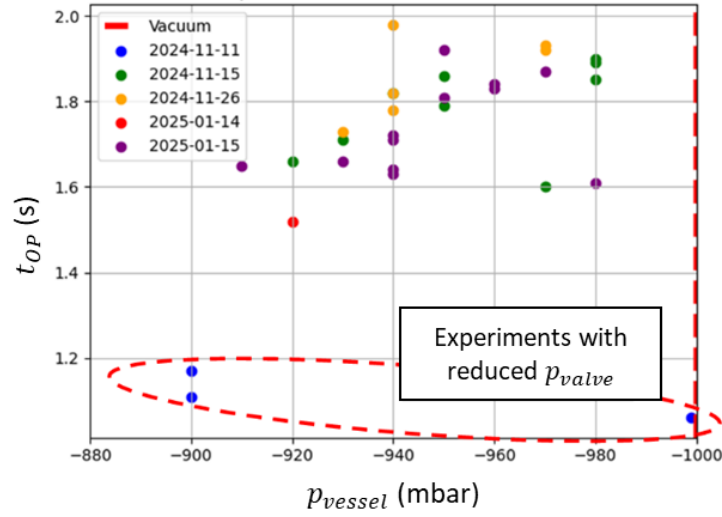
M	1.4	1.6	1.8	2.0	3.0
p_e/p_t	0.91	0.83	0.63	0.57	0.33
p_i/p_e	0.07	0.08	0.10	0.11	0.19
t_{RUN} (s)	3.3	3.1	2.8	2.9	3.9

Preliminary tests have indicated a variability in the duration of the operating runtime caused by varying vacuum levels (Fig. 4a). As demonstrated in Eq. 7, the initial pressure in the tank has a first order effect on the available runtime. The experimental observations show that runtime can be maximized by reducing as much as possible the initial pressure of the vacuum. For example, varying the initial pressure in the vessel from 900 mbar to 1000 mbar below ambient, causes a reduction in initial pressure ratio p_i/p_t from about 0.12 to about 0.02, which in turn produces an improvement of runtime t_{RUN} of approximately 18%, which is a considerable increase. In theory, this value could be improved further by bringing the initial pressure in the tank to full vacuum conditions, at approximately 1013 mbar below ambient. However, with Eq. 7 it can be demonstrated that this would provide benefits in the runtime in the order of 0.1s. Thus, the increased expense and complexity of a pumping system it is not worth the marginal gains in run time. The average pumping time t_p required for different tank to ambient pressure ratios are provided in Table 2.

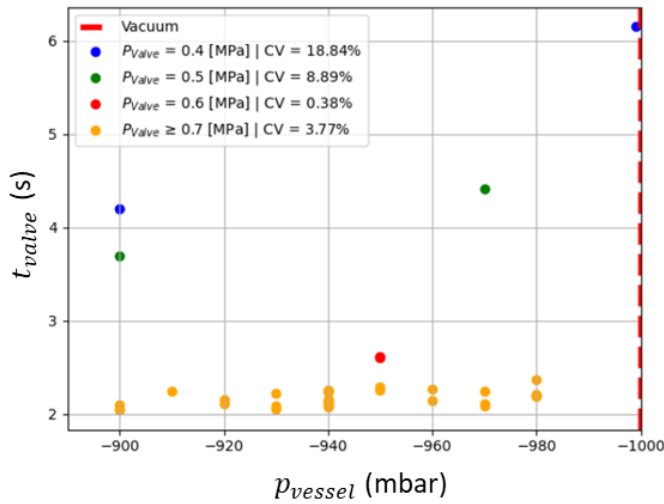
p_i/p_t (-)	0.90	0.80	0.70	0.61	0.51	0.41	0.31	0.21	0.11	0.03
t_p (mm:ss)	00:12	00:30	00:52	01:25	01:52	02:50	03:50	05:22	08:15	31:00

Table 2: Average pump time required for different vacuum levels.

The pneumatic pressure supplied to the fast-acting valve (Fig. 2b-3) was also a key parameter that influenced the runtime of the rig (Fig. 4a), as it was found that the pneumatic pressure had a first order effect on the speed at which the valve could open (Fig. 4b). In the allowable pneumatic pressure range between 0.4 – 0.8 MPa, the valve exhibited a consistent opening time of approximately 2 seconds only when it was operated at pressures above 0.7 MPa (Fig. 4b). The experimental runs undertaken with a pneumatic pressure of 0.4 MPa caused a significant penalty in the operational runtime (Fig. 4a), likely due to the failure of the valve to open quickly enough to accelerate the flow (Fig. 5).

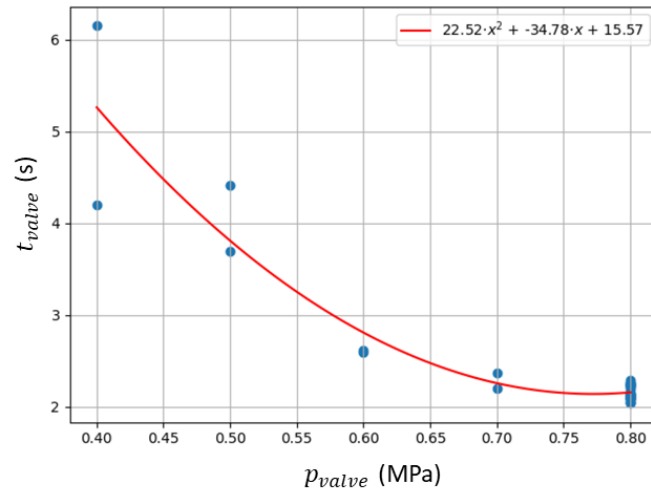


a)

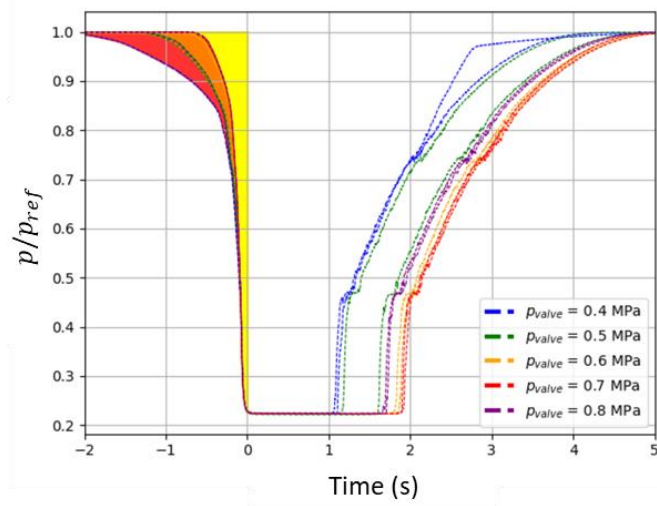


b)

Fig. 4: Parametric evaluation of Mini HI-Sim operation: a) influence of the vacuum level on the duration of the steady operational phase, and b) influence of the vessel vacuum level combined with pneumatic valve pressure on the opening time of the valve.



a)



b)

Fig. 5: Influence of the pneumatic pressure of the fast-acting valve on a) opening time of the valve, and b) the acceleration and steady operational phase.

III. Static pressure distribution of in the supersonic nozzle and working section

Static pressure tapings were installed in the supersonic nozzle and working section to characterize the variation of the static pressure during the acceleration and deceleration transients and the steady operational phase. The nozzle was equipped with 10 static pressure tapings at each of the top and the bottom wall centreline, while the working section was equipped with 7 static pressure tapings at each of the 4 side wall centrelines (Fig. 6a). Due to the need of pressure sensing in the inner working section, the outer working section was equipped with flexible pass-through tubulations in the gap between the two walls and through wall tapings at the outer walls (Fig. 6b). The distribution of the isentropic Mach number M_{is} across the sections has been calculated with the total-to-static pressure ratio assuming ambient pressure as total conditions and no pressure loss between the bell mouth and the working section:

$$M_{is} = \sqrt{\left(\frac{2}{\gamma - 1}\right) \left(\left(\frac{p_t}{p}\right)^{\frac{\gamma-1}{\gamma}} - 1 \right)} \quad (7)$$

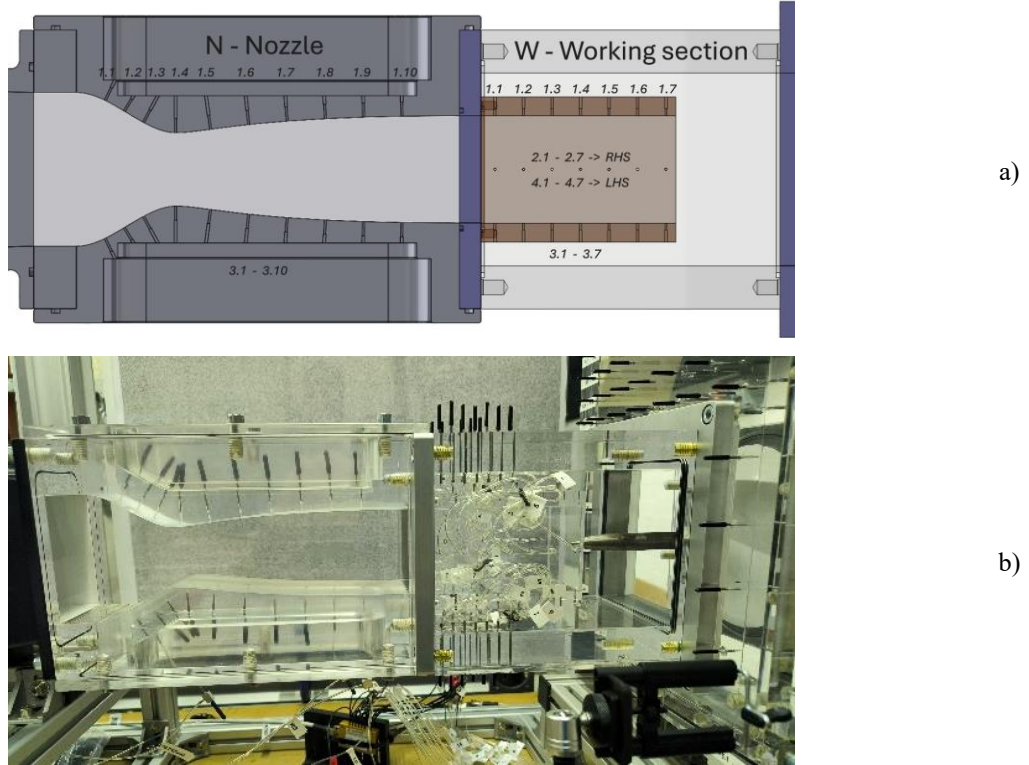


Fig. 6: a) Overview of the schematic for placement of the static tapping ports in the nozzle and working section, and b) real implementation.

It must be highlighted that the supersonic nozzle was designed with an area ratio intended to achieve a Mach number of 1.8 at the outlet plane. However, experimental results indicate that only a Mach number of 1.65 is achieved at the exit of the nozzle. This discrepancy arises from viscous effects, as the development of boundary layers along the nozzle walls effectively reduces the flow area and, consequently, the area ratio between nozzle throat and exit plane. In the full-scale configuration, the nozzle geometry has been optimized to account for these viscous effects by correcting the area ratio using CFD simulations [5]. When the design was scaled down for the small-scale test rig, however, the change in Reynolds number alters the boundary layer thickness relative to the nozzle dimensions. As the nozzle has been scaled geometrically and a further correction for different boundary layer thickness has not been

carried out, this results in a reduction of the effective area ratio and leads to the observed undershoot in exit Mach number (Fig. 7).

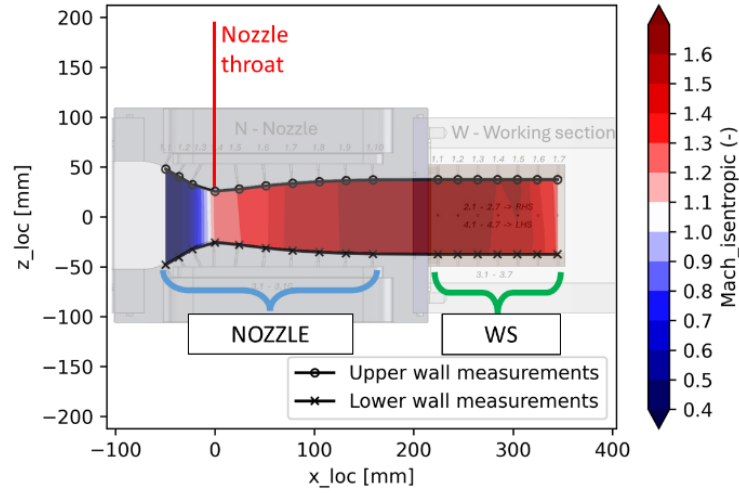
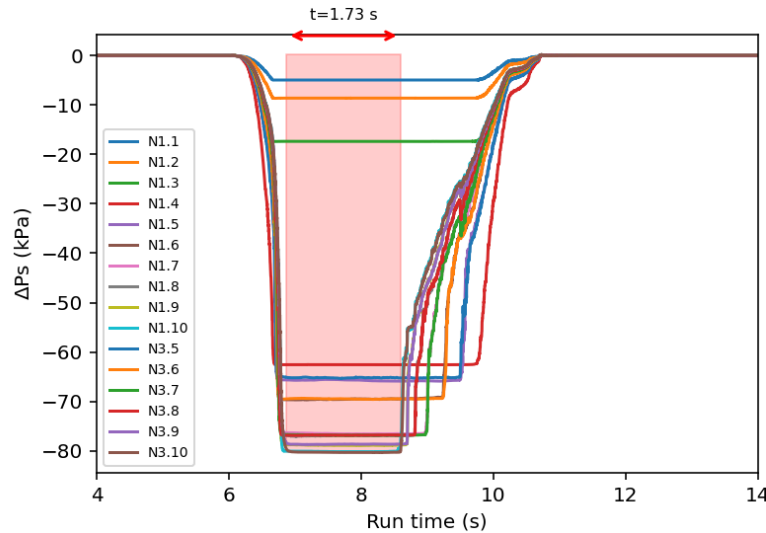
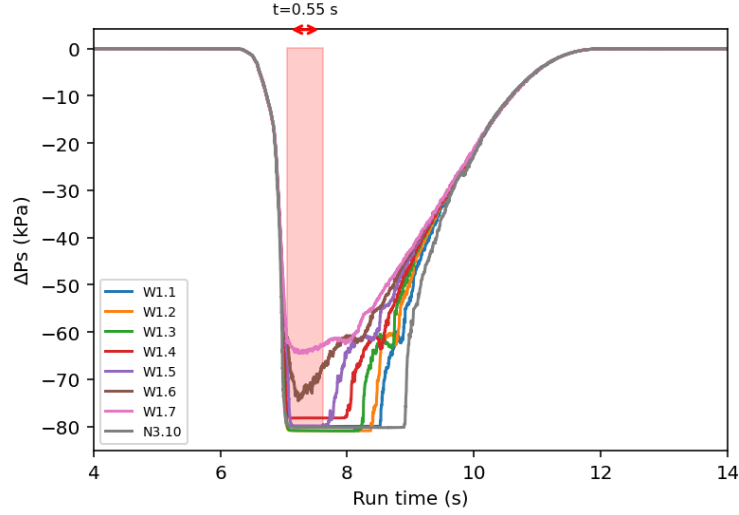


Fig. 7: Distribution of isentropic Mach number at the nozzle and working section.

The development of the supersonic stream in the nozzle can be evaluated by the temporal development of the pressure readings in the nozzle (Fig. 8). The pressure measurement by the nozzle top wall (N1.1-10) and bottom wall (N3.1-10) outline the region of steady operational phase, when the nozzle is in a choked condition (Fig. 8a). The length of this phase has been defined by the pressure plateau of duration 1.73s observed on line N3.10, which is located just upstream the nozzle exit plane, on the centreline of the bottom nozzle wall (Fig. 6a). However, in the temporal development of the pressure along the working section, it was observed that the locations near the exit plane of the inner working section (pressure taps W1.6 - W1.7) did not reach a pressure plateau (Fig. 8b). Rather, the pressure differential reaches a peak difference and decays again. For the duration of the plateau in the locations upstream of the inner working section exit, the plateau is maintained for a shorter period of time compared to the steady operational time in which the nozzle is choked. Up to location W1.5 the plateau of pressure is observed, but for a duration of only about 0.55s. This means that although the nozzle is choked, there is a mechanism that re-pressurizes the working section. The derivation of the isentropic Mach number (Eq. 7) during the steady operational phase of the working section of 0.55s confirmed that the nozzle accelerates the onset flow to a Mach number of approximately 1.65, however the working section re-pressurization causes a drop of Mach number as low as 1.2 near the inner working section exit plane.



a)



b)

Fig. 8: Differential pressure measurements on a) nozzle and b) working section according to the static pressure port locations of Fig. 6a.

The re-pressurization of the working section has been investigated with preliminary CFD studies on the full Mini HI-Sim rig. Simulations were performed using the compressible, pressure-based, steady-state Navier-Stokes solver in *ANSYS CFX 2021 R1* with a second-order spatial discretization scheme. To simplify the problem from transient to steady-state, constant were applied with an overall pressure ratio of 4.5 between the total pressure at the bell mouth of the mini HI-Sim (Fig. 2a-1) and the static pressure at the exhaust outlet, represented at the interface with the fast-acting valve (Fig. 2a-8). This condition is representative only for the beginning of the steady operational phase, just after the acceleration to supersonic conditions, as the overall pressure ratio keeps reducing during the run (Fig. 3). Ambient pressure and temperature at the inlet bell mouth (Fig. 2a-1) were set at 101,325 Pa and 293 K, respectively. The simulations revealed that there is sufficient pressure ratio to choke the nozzle throat and drive the normal shock out of the nozzle divergent section, but this is not sufficient to drive the shock completely out of the test section (Fig. 9a, c). The nozzle thus remains in an over-expanded state at the start of the test, which causes the flow to drop to lower Mach numbers through the development of shock structures just after the exit plane.

The cause of re-pressurization of the plenum has been attributed at the excessive contraction embedded in the top of the plenum, where inner liners and a sudden convergent section with rectangular to circular cross-section had been installed to converge the flow to the fast-acting valve (Fig. 2a-6, 8). This flow area reduction causes the flow to accelerate to sonic or near-sonic conditions (Fig. 9a) with consequent increase of static pressure in the plenum (Fig. 9c), potentially choking the exhaust outlet. The CFD simulations showed that the removal of the contraction feature above the plenum to increase the cross-sectional area would allow a better extraction of the flow as the pressure remains at constant low throughout the nozzle plenum (Fig. 9d). In turn, this configuration would allow the normal shock to be pushed downstream the enclosed free-jet, and stabilize the Mach number distribution to the target levels (Fig. 9b).

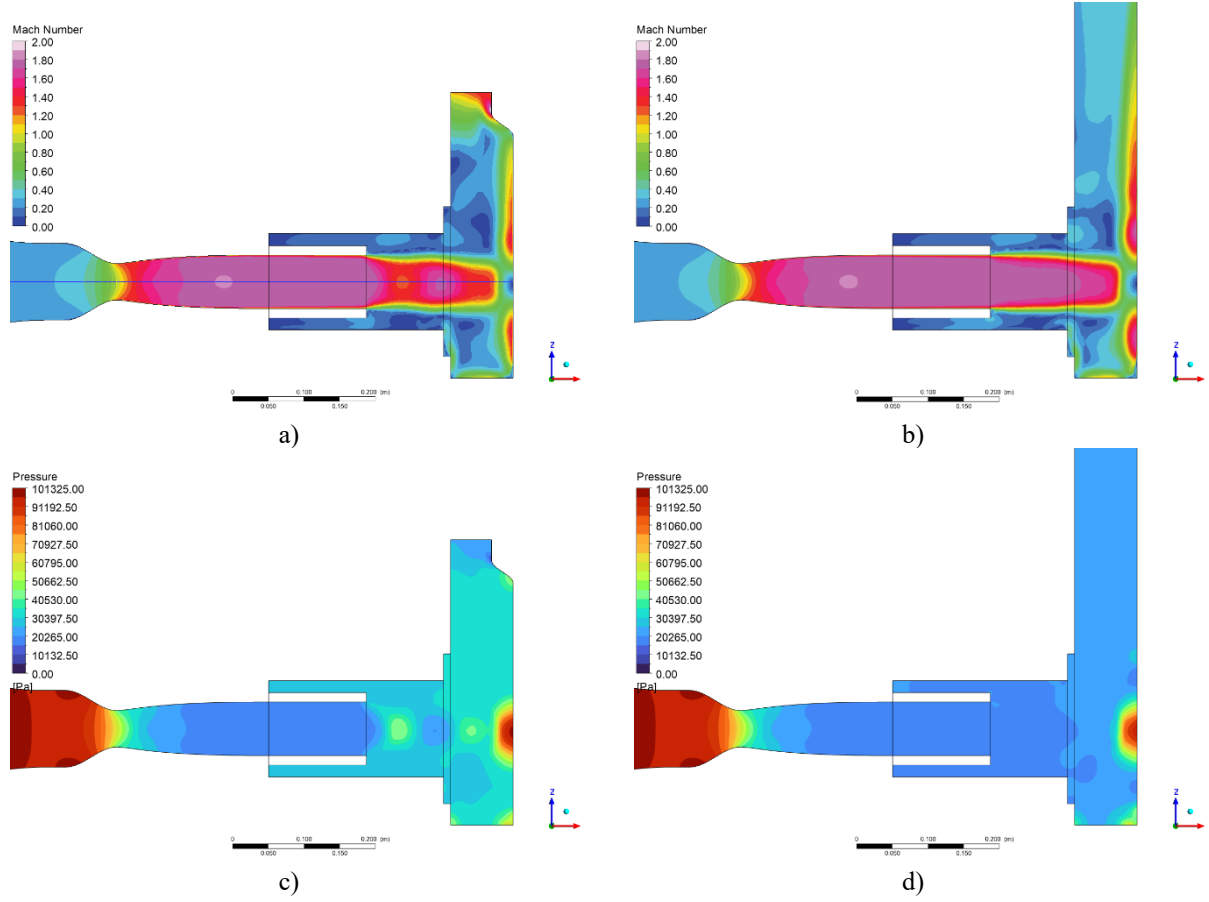


Fig. 9: Preliminary CFD simulations on the Mini HI-Sim rig to evaluate Mach number and static pressure distributions for a-c) original design, and b-d) modified exhaust.

A design modification has thus been implemented to increase the cross-sectional area in the transition between the plenum and the fast-acting valve. A preliminary measurement of the isentropic Mach number M_{is} through the tappings installed at the end of the nozzle (N3.10) and at the start and end of the working section (W1.1 and W1.7, respectively), indicated a plateau during the initial part of the steady operational phase at all locations (Fig. 10b). This means that the flow in both the nozzle and the working section are running full at the target Mach number. This shows an improvement against the original design where the target Mach number was reached only up to the beginning of the working section (W1.1, Fig. 10a). Strategies to improve further the operability of the test rig are currently being investigated and will be reported in a subsequent publication.

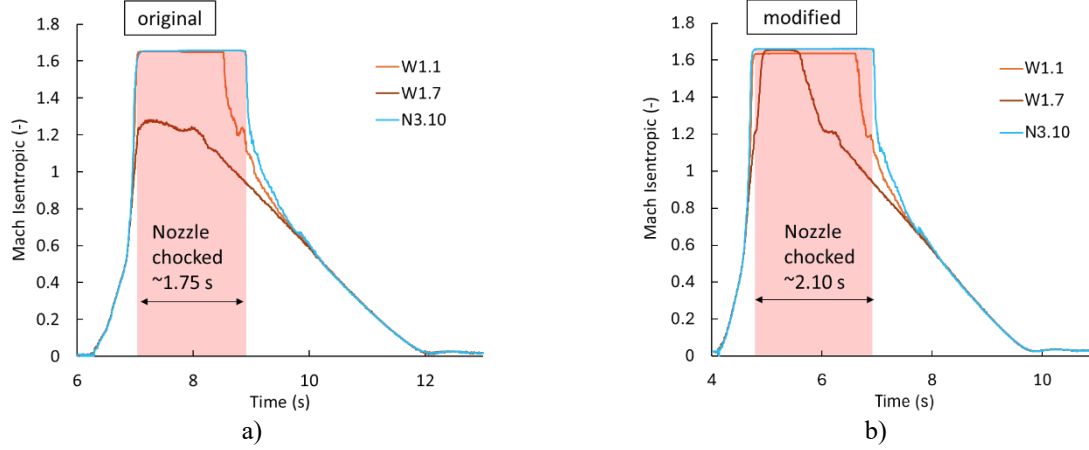


Fig. 10: Measurement of isentropic Mach number in at the nozzle exit plane (N3.10), working section inlet plane (W1.1) and outlet plane (W1.7) for a) original design, and b) modified exhaust.

IV. Investigation of flow distribution in the radial exhaust via flow visualization techniques.

A key element of the research through the small-scale experimental rig was also to better understand the flow characteristics within the radial diffuser during the 90° diversion towards the exhaust system. Previous CFD work on the full-scale facility showed that main two main counter-rotating vortical structures develop in the radial exhaust, at each side of the intake [5]. In the small-scale rig, tuft flow visualization was applied with the objective to further verify this behaviour. This technique provides a rapid and qualitative assessment of flow behavior without requiring complex measurement systems. Although the method is intrusive and the aerodynamic interaction of the tufts with the flow cannot be neglected, the cost-effectiveness and simplicity of the approach offer useful qualitative insights into local flow directionality and unsteadiness.

Tufts were installed at the internal walls of the radial exhaust and secured with adhesive tape. The tuft length was chosen as 20 mm on average, which exhibited a good compromise between the ability to follow the flow and the visibility (Fig. 11). Bright-colored neon-green tufts made of single thread nylon wire were preferred to enhance visibility on the black background. The ambient lighting was reduced, and additional light sources were placed around the radial exhaust walls to increase contrast. Image acquisition was undertaken with a smartphone camera capable of 240 Hz at 1920 x 1080 px resolution, with a stand-off distance of approximately 500mm and a camera lens with equivalent focal length of 24 mm. Image perspective correction was applied during post-processing via a Python OpenCV library (Fig. 11). The sequence of images corresponding to the steady operational Mach number has been identified through synchronization of the acquisition with the nozzle pressure measurements via a trigger sound. Approximately 350 frames were captured during each run in the steady operational phase.

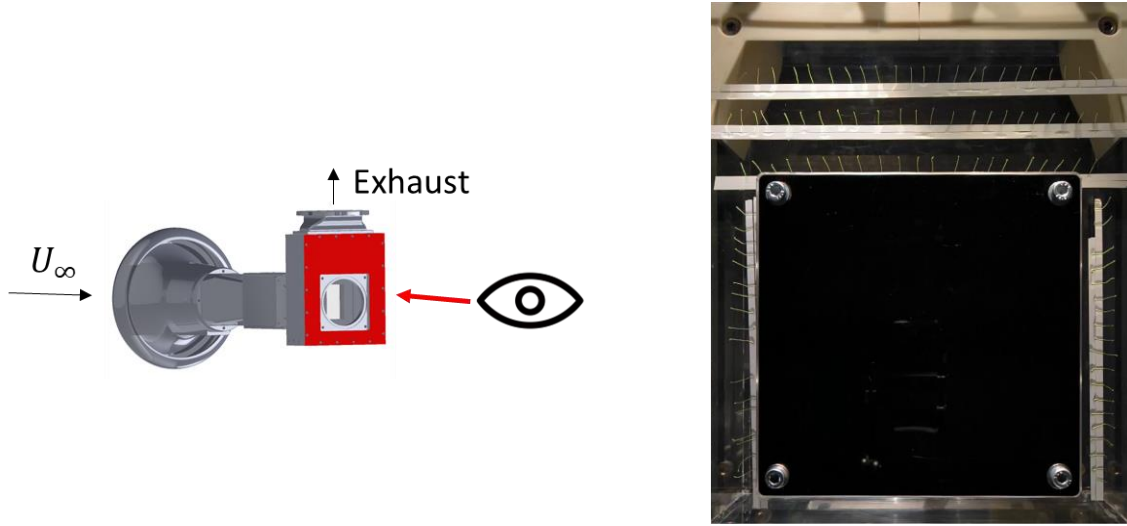


Fig. 11: Arrangement of tufts on adhesive tape at the back plane of the radial exhaust, with applied perspective correction. View looking upstream.

Despite the use of black paper to minimize background reflections, residual noise persisted due to light reflections and diffraction through the plexiglass walls, which complicated the assessment of the tuft position in time. The image sequence was initially captured in the BGR colour format and subsequently converted to HLS and HSV colour models for improved segmentation, with the scope to identify the tuft position across the image sequence. These colour models separate hue, lightness, and saturation, enabling the identification of pixel ranges corresponding to tuft colours (Fig. 12a, b). A thresholding approach was iteratively refined across multiple frames and the two-colour models to account background and lighting variation, to isolate the tufts. A binary mask was then generated for each frame, isolating the tuft pixels within the defined colour range (Fig. 13). Each mask was multiplied by the corresponding intensity frame to account for the brightness distribution across the image. Normalizing the pixel intensities enhanced the visual depth and allowed for accurate identification of tuft motion, even when motion blur was present (Fig. 14a). The average intensity across all frames, with grayscale inversion, revealed the superposition of the tuft position during the steady operational Mach number, with background noise significantly reduced (Fig. 14b, c).

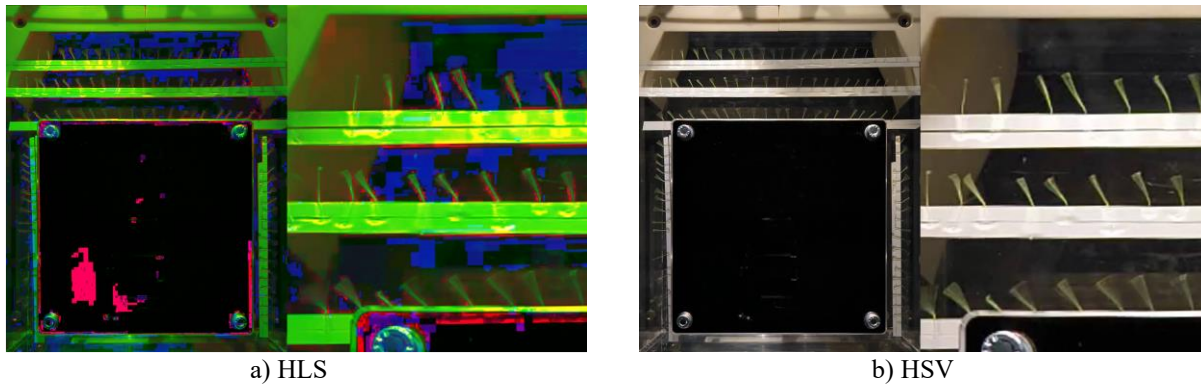


Fig. 12: Portion of the region of interest shown with a) HLS and b) HSV color formats.



a) Unmasked



b) Binary mask applied

Fig. 13: Assessment of the locations of the tufts through binary mask.

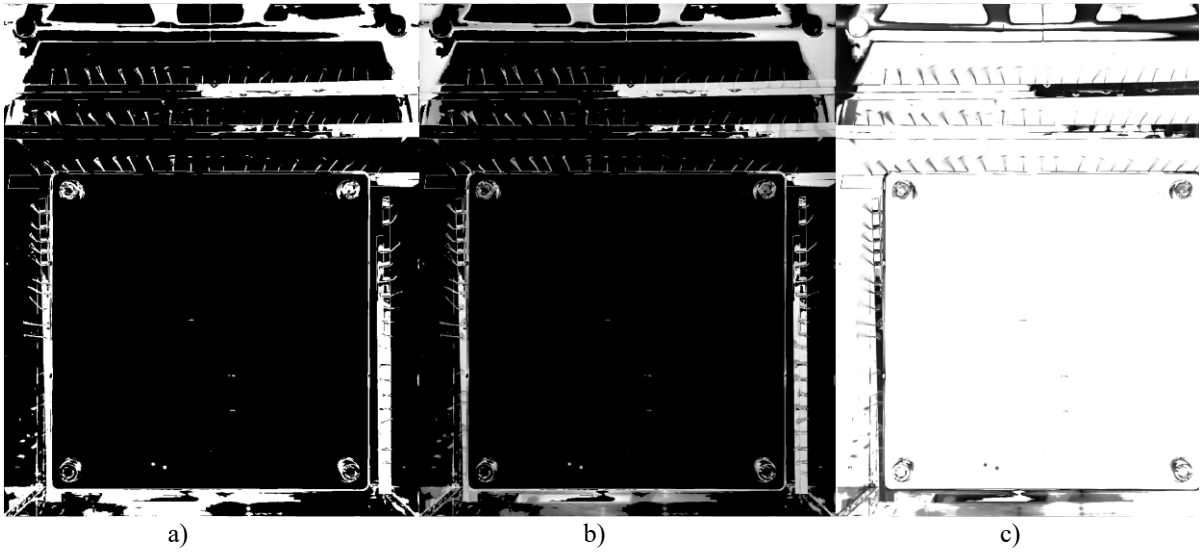


Fig. 14: Extraction of the superposition of the tuft locations in time during the steady operational phase through: a) intensity normalization, b) averaging of intensity, and c) greyscale inversion.

A vectorization of the image processing results has been carried out to offer a quantitative interpretation of the direction and unsteadiness of the flow at the tuft measurement regions. A measurement algorithm has been implemented to quantify the spread angle during the steady operational phase through the overlap of the tuft in the average image. This was based on the definition of an area of interest (Fig. 15a), the detection of the origin, peak and right and left boundary (RB and LB) of the tuft locations (Fig. 15b), and the calculation of the mean and range of the positions (Fig. 15c). The mean flow vector and corresponding spread angle were computed, representing the local flow direction and unsteadiness, respectively. The resulting vector field (Fig. 15c) provides a quantitative and qualitative visualization of the flow streamlines near the radial exhaust walls, which enables the assessment of the local flow alignment and the extraction characteristics of the exhaust.

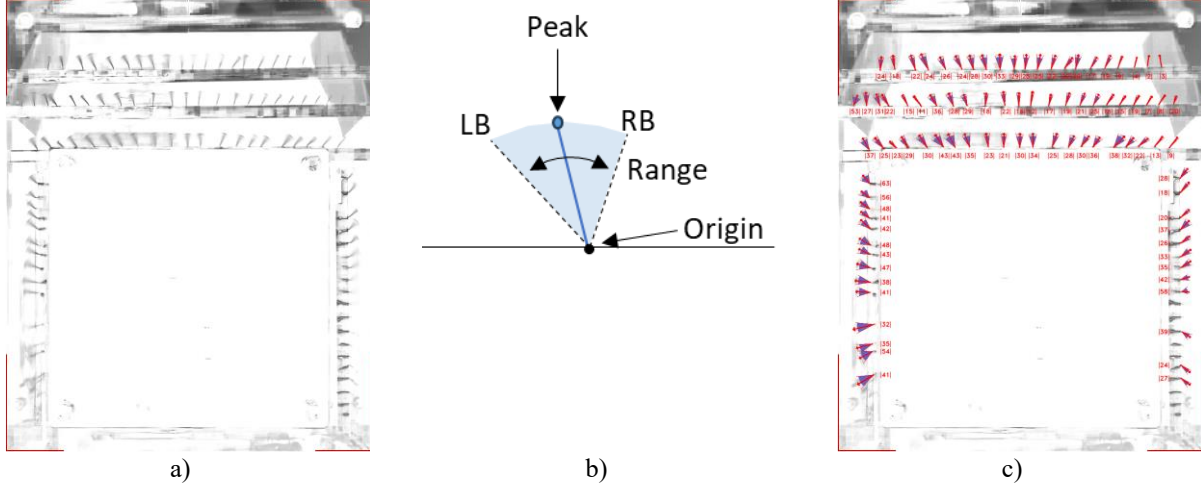


Fig. 15: Evaluation of the mean and range of the tuft locations during the steady operational phase through: a) definition of the area of interest, b) identification of the tuft origin, left and peak, and c) calculation of the angles.

The unsteady flow characteristics within the radial exhaust can be better understood by following the tufts' temporal variations. Due to the high flow velocities in the working section, and the sudden expansion to a larger area, the flow is discharged from the inner working section as an enclosed, free-jet [5]. The supersonic flow surrounds the bullet-shaped AGARD-2 [9] test article (Fig. 17a) and impinges on the flare, deflecting radially and away from the centerline. This can be seen by the radial direction assumed by the tufts at the back plane of the radial exhaust (Fig. 16c). The unsteadiness of the flow angularity measured at this location is relatively high as shown by the high range of flow angularity at measured locations, and likely caused by the interactions with the back and side panels, and the development of a counter-rotating paired-swirl vortices (Fig. 17a). These swirling structures can be confirmed by the coherent alignment of the tufts on the side panel in which the direction of the flow axially towards the front panel of the radial exhaust combines with the upward motion (Fig. 16c, Fig. 17b). As the flow proceeds in the upward motion towards the exhaust, it interacts also with the inner liners installed in the side panels and converges in the centerline of the vertical exhaust (Fig. 16a). The flow from the side panels is pushed laterally towards the vertical centerline, and this is envisaged as a stabilizing action as the vectors derived from the tuft position show very pronounced directionality with low unsteadiness (Fig. 16a). However, the fluctuations of the flow near the vertical centerline in the front panel of the radial exhaust is considerably more unsteady as shown by the large fluctuations of the tufts (Fig. 16a). This is attributed to the likely generation of a separation bubble in the central-upper region of the front panel of the radial exhaust (Fig. 17c). Indeed, the sudden 90-degree turn of the flow from axial in the working section to vertical in the radial exhaust likely causes a separation after the turning edge (Fig. 17c). In general, the fluid motion revealed with the tufts in the radial exhaust is aligned with the observations of previous computational work in the full-scale rig, which revealed paired and counter-rotating vortical motion in the extraction of the flow through the radial exhaust [5]. This is considered a qualitative validation of the design of the bypass and radial exhaust of the intake distortion facility.

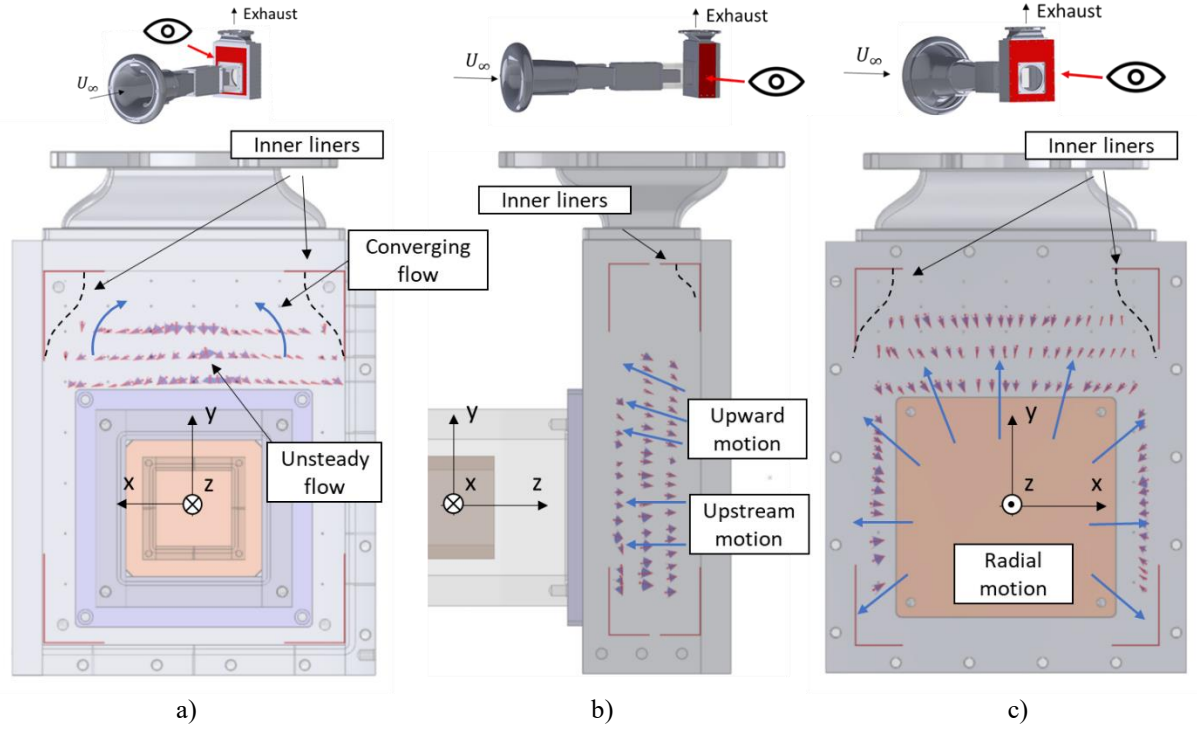


Fig. 16: Assessment of the flow distribution in the radial diffuser on the a) front, b) side, and c) back wall.

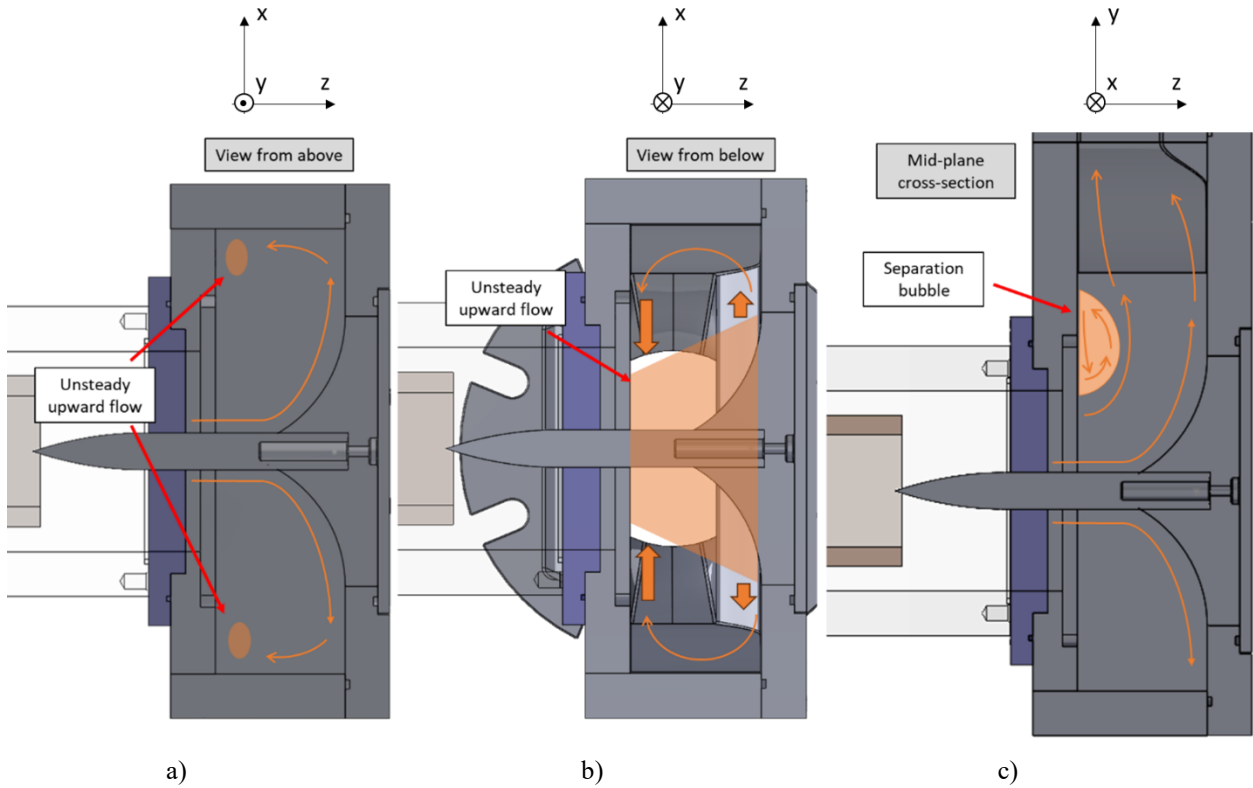


Fig. 17: Interpretation of the flow distribution in the radial exhaust as result of the tuft flow visualization experiments, for a) view from above, b) view from below, and c) mid-plane cross-section.

Conclusions

This research reported on the de-risking of a high-speed intake distortion test rig (HI-Sim), using a sub-scale, pilot test rig, known as Mini HI-Sim. Supersonic intakes present significant aerodynamic complexity. Where the intake is designed for sub-critical supersonic operation and is also large relative to the wind tunnel test section, integration with an experimental facility can be expected to be challenging. The flow field which develops around the intake due to spillage may strongly interact with walls of the wind tunnel, leading to unstable test section operation. The design of the HI-Sim rig mitigates this interference through an enclosed, free-jet. The design of the Mini HI-Sim replicates critical flow features of the full-scale HI-Sim test rig within a compact, vacuum-driven aspirated configuration, enabling controlled experimentation at Mach numbers up to 1.65, and a preliminary validation of the full-scale rig concept.

The originality of the current study stems from its integration of a radial exhaust system with optical diagnostic access, and its use of tuft-based flow visualization enhanced by image processing techniques. These innovations allow for both qualitative and quantitative assessment of the flow behaviour, particularly within the radial exhaust region where vortical structures and flow unsteadiness are prevalent. The image-based vectorization of tuft motion provides a new tool for evaluating flow directionality and unsteadiness, offering insights into the effectiveness of the radial exhaust design.

Analytical models based on compressible flow theory and polytropic expansion were developed to estimate the operational runtime, and experimental data were used to refine these models. The findings demonstrate that runtime and flow uniformity are highly sensitive to vacuum levels and valve actuation pressure. The reduction in test section Mach compared to the full-scale rig illustrates the importance of accounting for viscous effects in supersonic nozzle design. The observed over-expansion of the nozzle and the re-pressurization of the working section also highlighted sensitivity of rig performance to pressure losses in the exhaust and vacuum pipe system. Preliminary CFD investigations revealed that the re-pressurization of the plenum originates from an excessive geometric contraction in the exhaust system, which limits the effective pressure ratio available to fully displace the normal shock downstream of the enclosed free-jet. As a result, the nozzle remains over-expanded at the start of the test, and the shock system remains partially within the test section, reducing the achievable Mach number. The simulations further demonstrated that removing this contraction maintains a low, uniform pressure distribution in the plenum, which promotes stable positioning of the normal shock downstream of the jet, thereby enabling the intended Mach-number distribution at test onset. This has been validated with measurement of the isentropic Mach number in the working section through pressure tapings. The contribution of these insights is to inform future design iterations of the Mini HI-Sim and highlight the sensitivity of enclosed free-jet performance to plenum geometry.

In general, this work contributes to the broader understanding of supersonic intake aerodynamics towards a scalable, cost-effective platform for early validation of intake distortion phenomena. The Mini HI-Sim serves as a critical precursor to the full-scale HI-Sim, informing design decisions and instrumentation strategies. The methodologies are expected to benefit future research in propulsion integration and experimental fluid mechanics, supporting the advancement of next-generation high-speed aircraft and propulsion systems.

Acknowledgments

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

References

- [1] 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>
- [2] Seddon, J., and Goldsmith, E. L., “Intake Aerodynamics: An Account of the Mechanics of Flow in and around the Air Intakes of Turbine-Engined and Ramjet Aircraft and Missiles,” Collins Professional and Technical, London, 1985.
- [3] Anderson, B. H., “The Aerodynamic Characteristics of Vortex Ingestion for the F/A-18 Inlet Duct,” Reno, NV, US, 1991. <https://doi.org/10.2514/6.1991-130>
- [4] 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>
- [5] Migliorini, M., Zachos, P. K., MacManus, D., and Martin, P. G., “Aerodynamic Design of a High-Speed Intake Test Rig for Propulsion Integration Research,” presented at the AIAA SCITECH 2025 Forum, Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-2110>
- [6] Trapier, S., Deck, S., Duveau, P., and Sagaut, P., “Time-Frequency Analysis and Detection of Supersonic Inlet Buzz,” *AIAA Journal*, Vol. 45, No. 9, 2007, pp. 2273–2284. <https://doi.org/10.2514/1.29196>
- [7] Grenson, P., and Beneddine, S., “Analysis of Shock Oscillations of an External Compression Supersonic Inlet through Unsteady Numerical Simulations,” presented at the 2018 Applied Aerodynamics Conference, Atlanta, Georgia, 2018. <https://doi.org/10.2514/6.2018-3011>
- [8] Pope, A., and Goin, K. L., “High Speed Wind Tunnel Testing,” Wiley, 1965.
- [9] “A Review of Measurement on AGARD Calibration Models,” AGARDograph 64, November 1961.



DSPACE

<https://dspace.org/>

De-risking of a high-speed intake distortion facility using a small-scale pilot rig

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

2026-01-12

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2026-1978>

Migliorini M, Hersbach S, Zachos PK, et al., (2026) De-risking of a high-speed intake distortion facility using a small-scale pilot rig. In: Proceedings of the AIAA SCITECH 2026 Forum, 12-16 January 2026, Orlando, Florida, Article number AIAA 2026-1978