

Article

Internal Flow Measurements in Converging Ducts with Favorable and Localized Adverse Pressure Gradients

Vincent Onoja ^{*,†} , Keerthan Ganeshan [†]  and Daniel Cuppoletti

Department of Aerospace Engineering and Engineering Mechanicss, University of Cincinnati, Cincinnati, OH 45221, USA

* Correspondence: onojava@mail.uc.edu

† These authors contributed equally to this work.

Abstract

This paper presents internal flow measurements in two shape-transitioning nozzle ducts using planar Particle Image Velocimetry (PIV). Nozzle 1 is a converging nozzle transitioning from a square cross-section to a rectangular exit with an equivalent diameter (D_e) of 2.2 in, an exit aspect ratio of 6.68, and a length-to-diameter ratio (L/D_e) of 7.8. Nozzle 2 also converges globally but incorporates a diverging sidewall that introduces localized adverse pressure gradients. Both nozzles are tested at an exit Mach number of 0.2, corresponding to $Re_{D_e} \approx 2.50 \times 10^5$. Wall-normal velocity profiles reveal boundary layer thinning under favorable pressure gradients followed by thickening in regions of streamwise curvature and local adverse pressure gradients. In nozzle 2, the adverse streamwise pressure gradient along the diverging wall produces thicker boundary layers than in nozzle 1, while a cross-stream pressure imbalance shifts the velocity peak toward the diverging wall. Complementary steady RANS simulations using the $k-\omega$ SST turbulence model yield wall-normal velocity profile agreement within 2% mean absolute error for both nozzles in the upstream and mid-duct regions, with errors increasing toward the exit. Discharge coefficients from CFD and experiment agree within approximately 1%, with nozzle 1 exhibiting greater integrated losses than nozzle 2 despite thinner boundary layers at the measured plane, indicating a three-dimensional loss distribution. Independent pitot probe measurements at the nozzle exit confirm the PIV trends over the CFD predictions in the near-exit region.

Keywords: pressure gradients; particle image velocimetry; non-axisymmetric jets; turbulence; boundary layer growth



Academic Editors: Budi Chandra and Jun Yao

Received: 20 May 2026

Revised: 19 June 2026

Accepted: 22 June 2026

Published: 29 June 2026

Copyright: © 2026 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

The tendency for engine integration in future generation aircraft designs has resulted in increased complexity of the fluid passages connecting the turbomachinery and the airframe. In the blended-wing body (BWB) concept, the engines are moved from their traditionally mounted locations in pylons to integrated locations in the fuselage, providing superior lift-to-drag performance, acoustics, fuel efficiency, and safety compared to the classic tube-and-wing design [1]. A direct consequence of this integration is that both inlet and nozzle ducts must negotiate transitions in cross-sectional shape; these are typically between circular sections matched to the turbomachinery and rectangular sections, dictated by airframe geometry.

On the exhaust side, rectangular nozzle exits are desirable for airframe integration, thrust vectoring, and maneuverability [2,3]. However, the curvature required to transition from circular turbine exit planes to rectangular nozzle exits introduces complex three-dimensional flow features. Studies on circular-to-rectangular and square-to-rectangular transition ducts [2–6] have identified several characteristic phenomena, including flow recirculation in interior corners, counter-rotating vortices driven by secondary flows from convex and concave surfaces, and regions of flow separation. Wu et al. [2] established a link between global geometric parameters such as aspect ratio, length-to-diameter ratio, and area ratio to the onset of flow separation, producing stall margin charts to guide nozzle design. Subsequent parametric studies [7–10] have further explored how changes in these global descriptors affect nozzle performance and flow structure transport.

Early computational work within the NASA transition duct program confirmed that these three-dimensional flow features arise even in geometrically simple configurations. Parabolized Navier–Stokes computations by Abbott et al. [3] for circular-to-rectangular ducts revealed separation and secondary flows even in constant-area configurations, driven entirely by the shape change. Their flow regime maps delineated the boundary between attached and separated flow as a function of exit aspect ratio and length-to-diameter ratio, yet only a single experimental data point was available for validation at the time, a case of massive separation, underscoring the need for experimental characterization of these nozzle-type transition ducts.

On the inlet side, Brear et al. [11] demonstrated the strong coupling between duct curvature and performance in an Uninhabited Combat Air Vehicle (UCAV) inlet, where a flow separation within the curved section grew in intensity with increasing flow rate and produced stagnation pressure deficits at the Aerodynamic Interface Plane (AIP) that were directly attributable to adverse pressure gradients imposed by the inlet shape. These findings, along with the broader inlet literature [12,13], underscore that internal flow quality in transitioning ducts is governed by the interaction between wall curvature and the resulting pressure field.

Despite the recognized importance of these flows, high-fidelity experimental characterization of the internal flow field within transitioning nozzle ducts remains limited. Davis and Gessner [5] reported mean velocity and Reynolds stress variations at discrete streamwise stations within circular-to-rectangular ducts using pressure probes, Preston tubes, and hot-wire anemometry. Their measurements revealed secondary flow vortex pairs along the sidewalls that significantly distorted the primary velocity and stress fields, consistent with earlier observations by Taylor et al. [14] in square-to-circular inlet transitions. However, these measurements were confined to a small number of cross-sections and lacked the spatial resolution to capture the full evolution of the internal flow. Sobota and Marble [15] similarly acknowledged the difficulty of internal characterization in their study of annular-to-rectangular transitions, resorting to numerical simulation to describe the flow development between the inlet and exit planes.

The thermal environment within these nozzles adds further complexity. Crowe and Martins [6,16–18] investigated hot streak formation in S-shaped nozzles with varying aspect ratios, finding that flow separation provided an additional mixing mechanism that could reduce hot streak intensity. Their RANS-based predictions, however, changed significantly when unsteady simulations were performed on the same geometries, highlighting the need for high-spatial-resolution experimental data to validate the computational results.

A common theme across these studies is the reliance on measurements restricted to the inlet and exit planes, with limited or no optical access to the internal flow domain. Where internal data have been obtained, they have relied on intrusive probes at discrete locations, constraining the spatial resolution and preventing full-field characterization of the velocity

and turbulence fields within the duct. Notably, the majority of experimental investigations into shape-transitioning ducts have focused on the inlet side of the engine, where global adverse pressure gradients and engine face distortion are the primary concerns [11–13,19]. By comparison, the nozzle side, where the flow is globally accelerating but may encounter localized adverse pressure gradients due to wall curvature and cross-sectional divergence, has received considerably less experimental attention, with design guidance relying predominantly on computational predictions.

The present work addresses this gap by providing quantitative, spatially resolved internal flow measurements in two shape-transitioning nozzle ducts using planar Particle Image Velocimetry (PIV), extending preliminary results reported in Onoja [20]. The first nozzle converges from a square inlet to a rectangular exit with $D_e = 2.2$ in, an exit aspect ratio of 6.68, and $L/D_e = 7.8$; the second converges from a rectangular inlet of approximately half the area to a similar rectangular exit, but features a diverging sidewall that introduces localized adverse pressure gradients. The paper presents a detailed boundary layer analysis using wall-normal velocity profiles, examines the influence of streamwise curvature and local pressure gradients on flow development, and assesses exit flow degradation in total pressure and discharge coefficient. Complementary RANS simulations yield wall-normal velocity profile agreement within 2% mean absolute error for both nozzle cases, excluding near-exit regions.

2. Experimental Facility and Technique

The experiments were conducted in a modular research facility at the University of Cincinnati, depicted in Figure 1. The facility was designed to accommodate nozzle configurations with varying degrees of streamline curvature and cross-sectional shape transition. High-pressure air is supplied through a 2-inch pipe connected to a storage tank and expanded into a 6-inch flanged plenum equipped with a honeycomb flow straightener to reduce upstream turbulence. The total pressure in the plenum is regulated by a pair of gate valves (one coarse and one fine) positioned along the supply line. Inflow conditions are measured by a PX329 series Omega pressure transducer and a Type-K thermocouple installed upstream of the test section, near the plenum exit. The plenum and piping are constructed from materials rated to 1000 K (1340 °F) and 13.8 bar (200 psi). The test section is fitted with polycarbonate optical access windows that admit pulsed laser illumination for PIV measurements of the seeded internal flow.

During testing, the operating condition was set by regulating the plenum stagnation pressure with the gate valves to achieve a nozzle pressure ratio (NPR) of 1.029, corresponding to the target exit Mach number of 0.2. The nozzle exhausted to ambient laboratory conditions, and the exit static pressure was taken equal to the ambient pressure, measured by a second PX329 transducer. Together, the regulated plenum stagnation condition and the ambient exit pressure define the inlet and exit boundary conditions of the experiment, and correspond to those applied in the numerical simulations described in Section 2.4.

2.1. Uncertainty Estimates

The mass flow rate is measured using an orifice plate with a 1.5 in bore diameter. Two Meriam 2110P smart pressure transducers record the upstream pressure (P_1) and the differential pressure (Δp) across the orifice plate, each accurate to $\pm 0.05\%$ of full scale. An Omega Type-K thermocouple measures the upstream temperature with a standard limit of error of ± 2.2 °C. The combined standard uncertainty of the mass flow rate, propagated from these component uncertainties, is $\pm 0.37\%$.

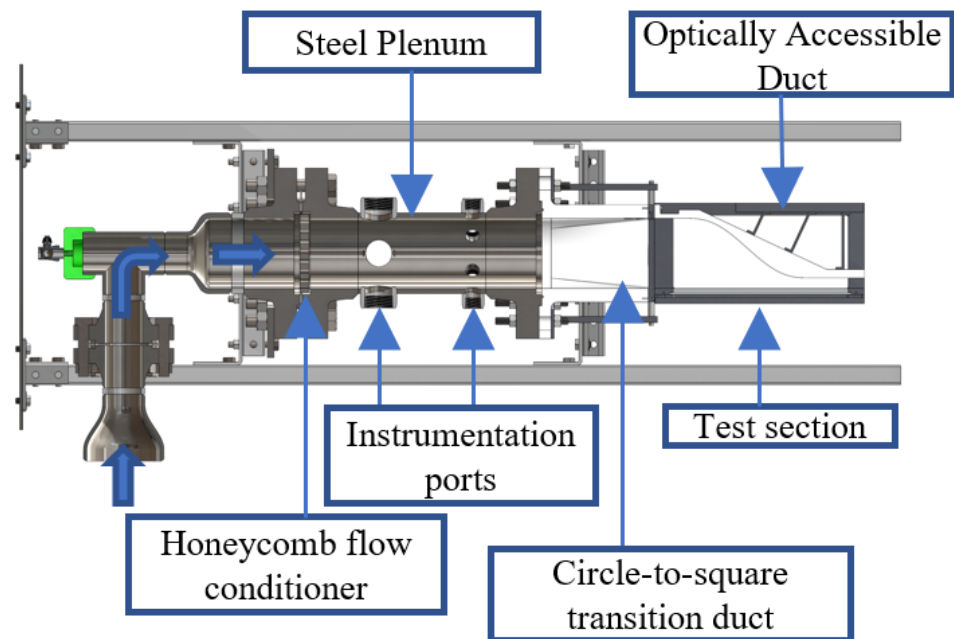


Figure 1. Test facility for nozzle studies.

Two PX329 series Omega absolute pressure transducers, accurate to $\pm 0.25\%$ of full scale, record the plenum stagnation pressure and atmospheric pressure. These measurements provide the nozzle pressure ratio (NPR) from which the nozzle exit Mach number is evaluated. The flow through the nozzle test section is treated as isentropic at all operating conditions reported herein, with the static pressure at the nozzle exit equal to the recorded ambient pressure. All pressure and temperature signals are acquired using an NI 9215 analog input module with 16-bit resolution; the resolution uncertainty of the acquisition system is negligible relative to the transducer uncertainties. Actual operating conditions at each set point varied slightly between test days due to ambient temperature and pressure fluctuations.

2.2. Nozzle Geometries

Initial cold flow studies considered two nozzle configurations with characteristics listed in Table 1. The first is a converging nozzle whose wall profile is derived from the fifth-order polynomial contraction recommended by Bell and Mehta [21] for small, low-speed wind tunnels. Equation (1) gives the resulting profile using their recommendation for the inlet and exit heights of the present study. Equation (2) is a modified form of this profile that departs from the well-behaved contraction shape, with the aim of increasing exit flow non-uniformity. A comparison of the wall profiles produced by both equations is shown in Figure 2.

The second nozzle features a gradually converging section in the streamwise direction with diverging sidewalls defined by the fifth-order polynomial given in Equation (3), introducing localized adverse pressure gradients within a globally converging passage. All equations are presented in non-dimensional form with $X = x/L$. The corresponding nozzle models are depicted in Figure 3, and a comprehensive visualization of the two- and three-dimensional nozzle attributes along with the coordinate system used throughout this study is shown in Figure 4.

The streamwise area distributions of both nozzles are shown in Figure 5. The inlet area of nozzle 2 is approximately half that of nozzle 1, and its passage converges with a diverging side surface to approximately match the exit area of nozzle 1.

$$Y_1(X) = -2.556 \times 10^{-4}(X \cdot L - 2.5)^5 + 6.572 \times 10^{-3}(X \cdot L - 2.5)^4 - 4.506 \times 10^{-2}(X \cdot L - 2.5)^3; \quad [0.144 \leq x/L \leq 0.739], \quad (1)$$

$$Y_2(X) = -5.803 \times 10^{-5}(X \cdot L - 2.5)^5 + 2.196 \times 10^{-3}(X \cdot L - 2.5)^4 - 1.787 \times 10^{-2}(X \cdot L - 2.5)^3 - 3.058 \times 10^{-2}(X \cdot L - 2.5)^2 - 1.26 \times 10^{-2}(X \cdot L - 2.5); \quad [0.144 \leq x/L \leq 0.739], \quad (2)$$

$$Z_3(X) = -9.516 \times 10^{-5}(X \cdot L)^5 + 4.983 \times 10^{-3}(X \cdot L)^4 - 9.509 \times 10^{-2}(X \cdot L)^3 + 0.802(X \cdot L)^2 - 3.104(X \cdot L) + 7.409; \quad [0.2604 \leq x/L \leq 0.942] \quad (3)$$

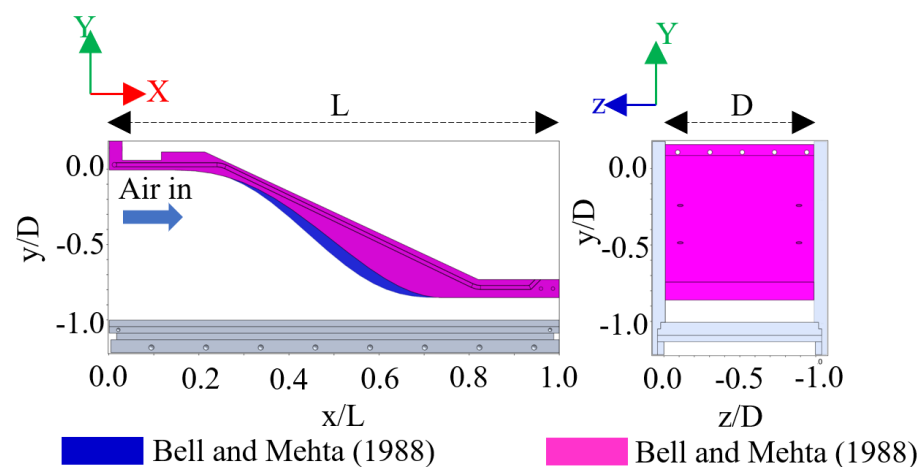


Figure 2. Wall surface profile definition for nozzle 1. $D = 146.3$ mm, $L = 438.9$ mm [21].

The origin of the coordinate system is located at the nozzle entrance, and all axial locations are referenced to this origin. Both nozzles were fabricated from ABS material via fused deposition modeling (FDM) 3-D printing with 100% infill density, a slice height of 0.013 in, and a wall thickness of 0.18 in. Surface finish was improved by alternating applications of a light black paint coating with progressive sanding using 120- to 400-grit sandpaper; any print imperfections were filled with a sandable filler prior to final finishing.

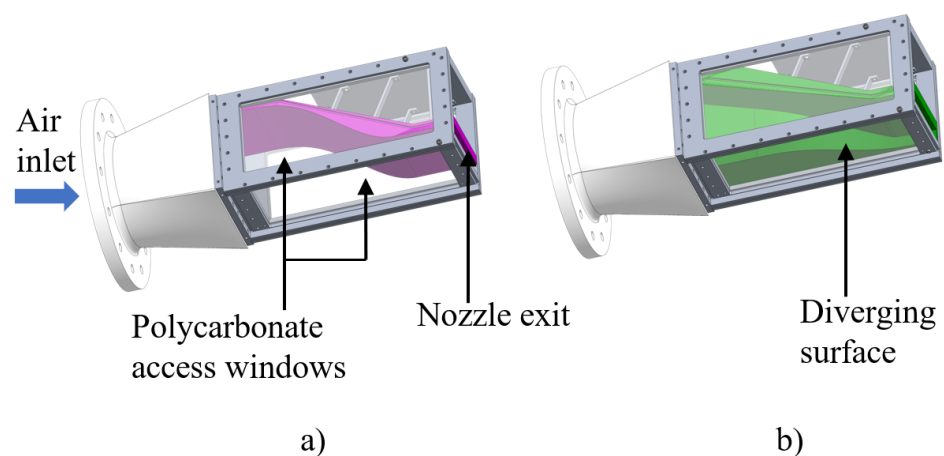


Figure 3. Nozzle configurations: (a) nozzle 1 (b) nozzle 2.

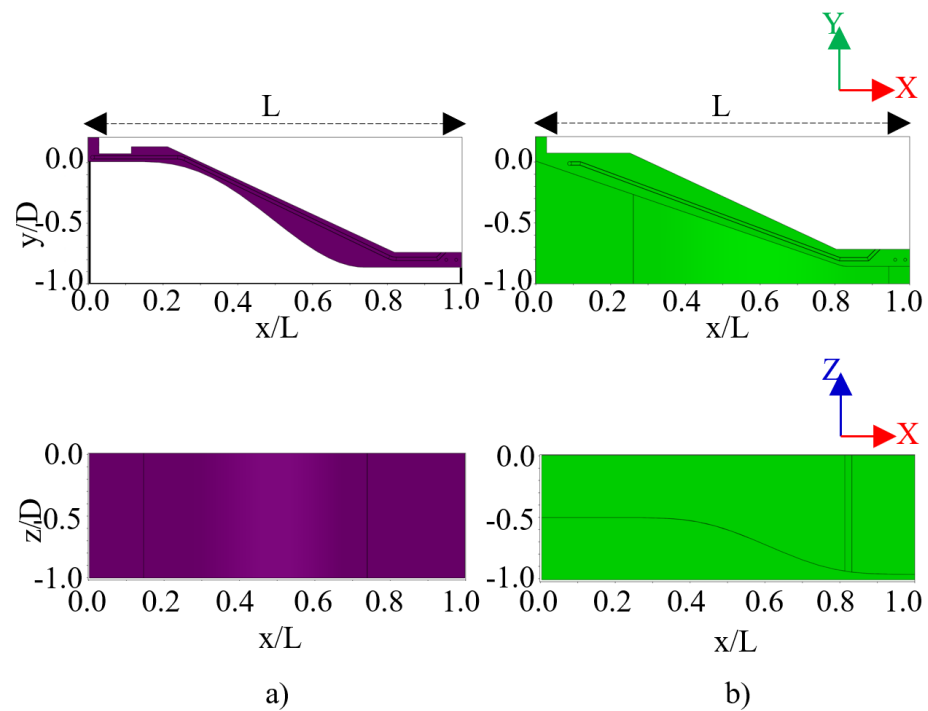


Figure 4. Nozzle profiles with common coordinate system. $D = 146.3$ mm, $L = 438.9$ mm. (a) Nozzle 1, (b) nozzle 2.

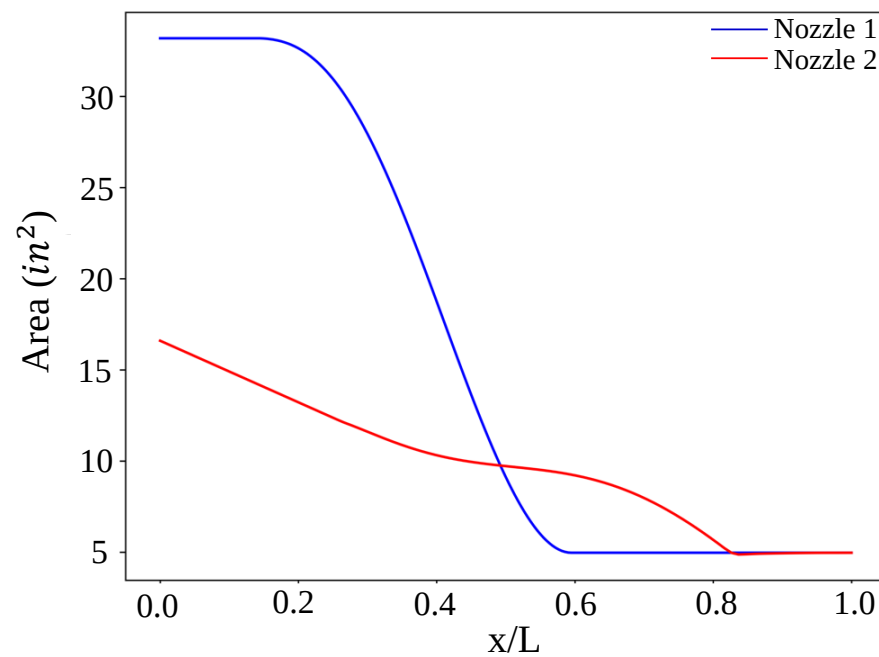


Figure 5. Area distribution of nozzle 1 and nozzle 2.

Table 1. Nozzle characteristics.

Nozzle	Area Ratio	Length Ratio, L/D_e	Exit Aspect Ratio, AR_e	Curve Definition
1	6.68:1	7.8	6.68	5th order polynomial
2	3.34:1	7.8	6.12	5th order polynomial

The chosen nozzle geometries are intended to represent, in a simplified form, the flow paths encountered by exhaust gases through shape-transitioning nozzles in next-generation aircraft. The two configurations decouple the flow problem: nozzle 1 subjects the flow to predominantly favorable pressure gradients, while nozzle 2 introduces both favorable and adverse pressure gradients within the same passage. Internal flow through both nozzles is characterized using planar Particle Image Velocimetry (PIV). Preliminary results across exit Mach numbers of 0.1 to 0.3 were reported in Onoja and Cuppoletti [20], where the general flow behavior was shown to be insensitive to Mach number over this range. The present study therefore focuses on the $M = 0.2$ case, corresponding to a Reynolds number of $Re_{D_e} \approx 2.50 \times 10^5$ for both nozzles, with the nozzle exit equivalent diameter $D_e = 2.2$ in serving as the characteristic length. The measured exit jet velocities from PIV are $V_{\text{jet}} = 70.28$ m/s and 71.61 m/s for nozzles 1 and 2, respectively. Optical access was maintained over the range $0.089 \leq x/L \leq 0.92$; beyond this range, the cross-sectional area remains invariant.

2.3. Particle Image Velocimetry

Two-component, high-speed Particle Image Velocimetry (PIV) measurements were conducted using a Photonics DM Series dual-head laser operating at 527 nm, with each head emitting pulses of approximately 30 mJ. A combination of focusing, turning, and light-spreading optics formed a light sheet of approximately 1.8 mm thickness to illuminate the seeded flow. Double-frame images were acquired with a FASTCAM Nova S12 high-speed camera (1024×1024 pixels) fitted with a Nikon 50 mm lens. The camera and focusing-spreading optics were mounted on two traverse mechanisms, enabling observation of the flow through different cutting planes from the bottom or sidewalls.

For each test case, 1000 raw image pairs were acquired at 1 kHz, yielding an approximate 10-pixel particle displacement between consecutive frames. To capture the entire flow domain, the camera was traversed vertically in three steps with a 10% overlap between the inlet and midsection fields of view (FOVs) and a 35% overlap between the midsection and exit FOVs. The time interval between laser pulses (Δt) was set based on estimated exit velocities for each flow condition and the FOV dimensions; Δt therefore varied across FOVs and nozzles, ranging from approximately 22 μs for the exit FOVs to 112 μs for the inlet FOV of nozzle 1. Details of the PIV acquisition parameters are given in Table 2.

Table 2. PIV acquisition parameters.

Nozzle	Image Rate (kHz)	Δt (μs)	Digital Resolution (px/mm)	Vector Array Size
Nozzle 1				
Inlet FOV	1	112.87	5.68	256×256
Mid FOV	1	25.74	6.20	256×256
Exit FOV	1	23.41	6.20	256×256
Nozzle 2				
Inlet FOV	1	48.29	6.56	256×256
Mid FOV	1	27.98	6.56	256×256
Exit FOV	1	22.27	6.56	256×256

The flow was seeded with olive oil particles of approximately 1 μm diameter, generated by a LaVision aerosol generator. Recorded images were processed using LaVision DaVis 10.1.2 with a multi-pass cross-correlation algorithm of decreasing interrogation window size: an initial pass of 128×128 pixels with 50% overlap, followed by a final pass of 16×16 pixels with 75% overlap, producing 256×256 vectors per FOV. A combination

of black spray paint and Rhodamine WT (water-based) was applied to the nozzle interior surfaces to improve surface smoothness and minimize laser reflections. The optical arrangement is shown in Figure 6.

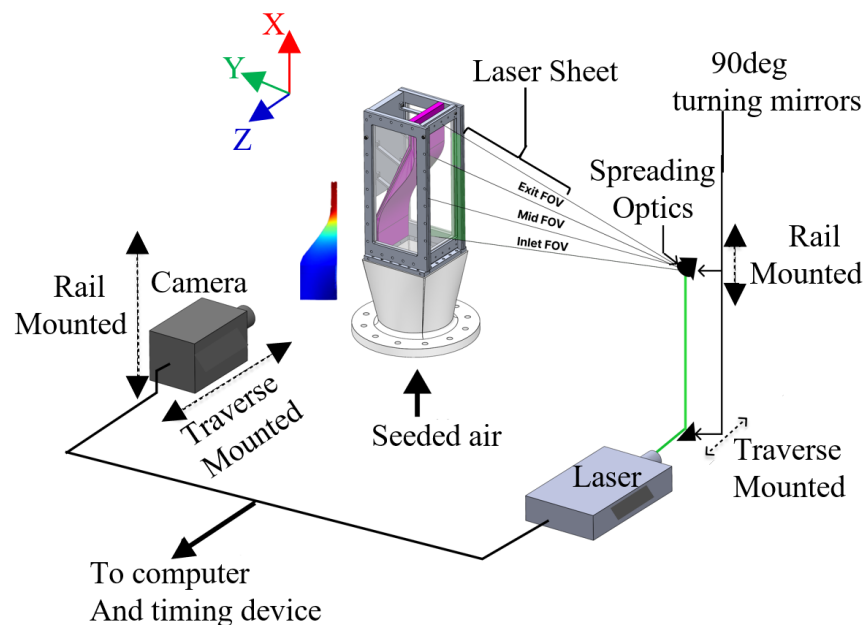


Figure 6. PIV optical setup.

2.4. Numerical Model and Computational Domain

The computational component of this study was performed using the ANSYS Fluent 2021 R2 commercial CFD code [22], solving the steady-state, density-based Reynolds-Averaged Navier–Stokes (RANS) equations. These equations govern the conservation of mass, momentum, and energy for the mean flow, with the effects of turbulence incorporated through additional terms. The k – ω Shear Stress Transport (SST) turbulence model [23] was employed to close the system of equations arising from Reynolds averaging.

The steady-state RANS continuity equation is:

$$\frac{\partial}{\partial x_i} (\bar{\rho} \bar{u}_i) = 0. \quad (4)$$

The RANS momentum equations, including the Reynolds stresses, are:

$$\frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\bar{\mu} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \bar{\mu} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right] - \frac{\partial}{\partial x_j} (\bar{\rho} u'_i u'_j). \quad (5)$$

The Reynolds stresses, $-\bar{\rho} u'_i u'_j$, are modeled using the Boussinesq hypothesis:

$$-\bar{\rho} u'_i u'_j = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_t \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij}, \quad (6)$$

where μ_t is the turbulent eddy viscosity. The k – ω SST model determines μ_t from two transport equations: one for the turbulent kinetic energy (k) and one for the specific dissipation rate (ω).

The turbulent kinetic energy transport equation is:

$$\frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_j k) = P_k - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right], \quad (7)$$

where P_k is the production of turbulent kinetic energy. The specific dissipation rate transport equation is:

$$\frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_j \omega) = \alpha \frac{\omega}{k} P_k - \beta \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + F. \quad (8)$$

The energy equation, accounting for turbulence, is:

$$\frac{\partial}{\partial x_j} \left[\bar{\rho} \bar{u}_j \left(\bar{h} + \frac{\bar{u}_k \bar{u}_k}{2} \right) \right] = \frac{\partial}{\partial x_j} \left(k_{\text{eff}} \frac{\partial \bar{T}}{\partial x_j} \right) + \Phi, \quad (9)$$

where $k_{\text{eff}} = \lambda + \mu_t / \text{Pr}_t$, λ is the molecular thermal conductivity, and Pr_t is the turbulent Prandtl number. Air is treated as an ideal gas:

$$\bar{p} = \bar{\rho} R \bar{T}, \quad (10)$$

and the dynamic viscosity is modeled using Sutherland's law:

$$\bar{\mu} = \mu_0 \left(\frac{\bar{T}}{T_0} \right)^{3/2} \frac{T_0 + S}{\bar{T} + S}. \quad (11)$$

Crowe [6] employed the k - ω model to study internal nozzle flows within curved geometries, while noting the need for high-resolution experimental measurements to validate the numerical predictions. The present work addresses this gap by providing detailed PIV measurements and using them to evaluate the performance of the k - ω SST model for internal nozzle applications. A broader assessment of turbulence closure sensitivity for this class of shape-transitioning nozzles, comparing the k - ω SST and Spalart–Allmaras RANS models with wall-modeled large-eddy simulation (WMLES) and validation-quality experimental data, is reported separately by the authors [24].

In a previous study [20], two-dimensional simulations of nozzle 1 showed close agreement with PIV measurements. To improve comparability in the spanwise direction, three-dimensional simulations were performed for both nozzles in the present work. The computational domains were discretized using the multi-zone meshing feature in ANSYS Mesher, producing fully structured hexahedral grids with refinement concentrated near walls, corners, and bends. The characteristic cell sizes at the finest level are 4.37×10^{-5} m for nozzle 1 and 7.52×10^{-5} m for nozzle 2. Near-wall refinement was applied at a growth rate of 1.2, targeting $y^+ = 1$ to ensure consistency with the wall-resolved requirements of the k - ω SST model.

A nozzle pressure ratio (NPR) of 1.029 was specified with ambient conditions of 14.4 psi and 295.93 K, matching the flow conditions of the PIV experiments. The boundary conditions consisted of a pressure inlet at the nozzle entrance, no-slip walls, and a pressure outlet at the exit. Figure 7 illustrates the discretized computational domain and the applied boundary conditions. Convergence was monitored through the scaled residuals of the continuity, momentum, energy, and turbulence equations, with a criterion of 10^{-6} applied to each variable. The solution was deemed converged once all residuals remained below this threshold and the mass flow rate exhibited negligible variation with continued iterations.

Grid Sensitivity Analysis

A grid convergence study was conducted using the Grid Convergence Index (GCI) method recommended by Roache [25] to assess the sensitivity of the numerical results to grid resolution. Three grid levels were employed for nozzle 1 with an effective grid refinement ratio of $r = 1.587$, as summarized in Table 3. The computed mass flow rate at

the nozzle outlet was selected as the quantity of interest. The relative error between the fine (N_1) and medium (N_2) grids is 0.0217%, and the GCI values at both levels are well below 1%, confirming that the solution is within the asymptotic range of convergence [26].

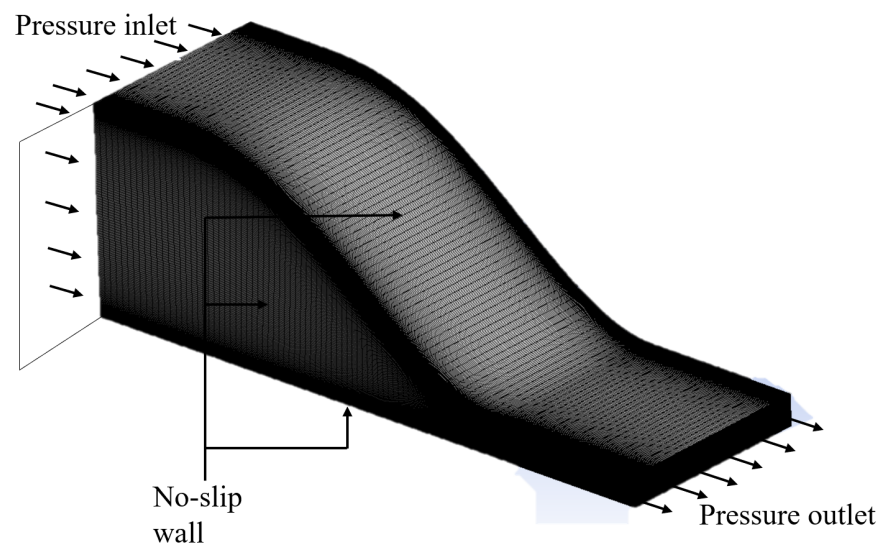


Figure 7. Computational domain of nozzle 1.

Figure 8 compares velocity magnitudes along the nozzle centreline for all grid levels and the measured PIV velocities for both nozzles. For nozzle 1, the velocity profiles show negligible variation across all three grid levels and closely approximate the PIV measurements. For nozzle 2, successive grid refinement progressively improves agreement with the PIV data, particularly in the near-exit regions. Based on these results, the fine grid was selected for all simulations reported herein.

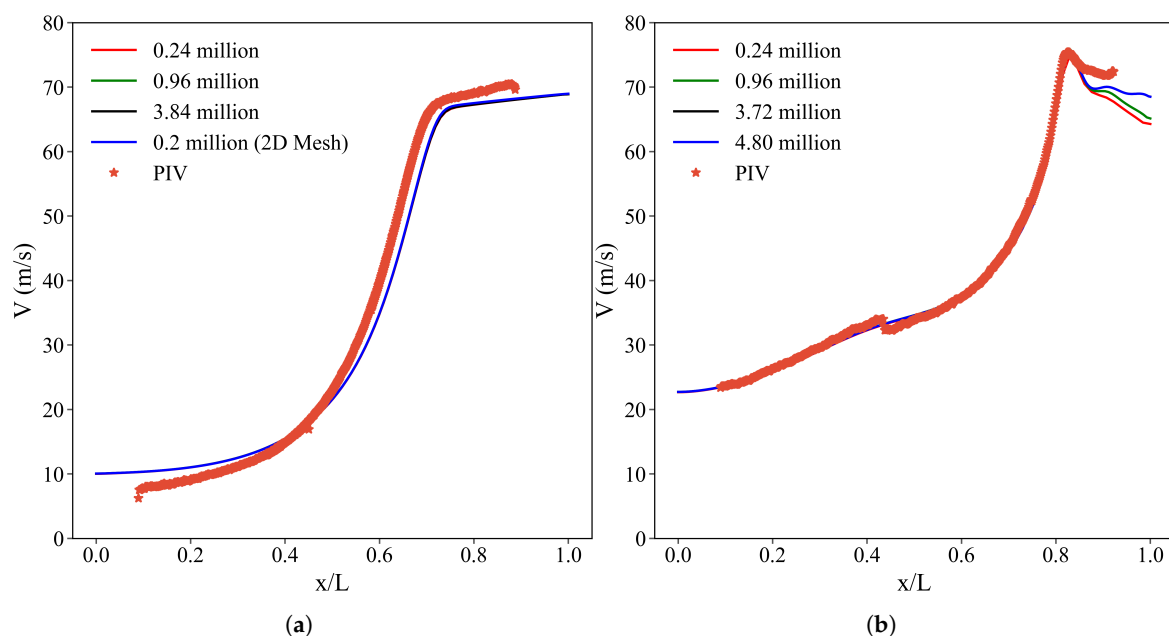


Figure 8. Grid sensitivity along nozzle centrelines at exit $M = 0.2$, $Re_{D_e} = 2.50 \times 10^5$. (a) Velocity along $y/D = -0.93$ with laser sheet at $z/D = -0.5$ for nozzle 1. (b) Velocity along $z/D = -0.25$ with laser sheet at $y/D = -0.85$ for nozzle 2. The velocity profiles for the different grid levels overlap almost entirely, indicating that the solution is grid-independent.

Table 3. Grid convergence study for nozzle 1. GCI computed using mass flow rate as the quantity of interest, with safety factor $F_s = 1.25$.

	N_1	N_2	N_3	ϵ_{12} (%)	ϵ_{23} (%)	GCI_{12} (%)	GCI_{23} (%)
$N_1 = 3.84 \times 10^6$, $N_2 = 0.96 \times 10^6$, $N_3 = 0.24 \times 10^6$, $r = 1.587$							
$\dot{m}$ (kg/s)	0.2475	0.2474	0.2468	0.0217	0.2380	0.00272	0.02983

3. Results

This section presents results from experimental and numerical investigations of the two nozzle configurations. Figure 8 shows the overall acceleration of the flow in both nozzles, driven by the globally favorable pressure gradient. In both cases, the acceleration is gradual, with peak streamwise velocities occurring toward the exit. In nozzle 2, the velocity peaks at approximately $x/L = 0.83$, a consequence of the diverging sidewalls that slightly increase the effective nozzle area beyond this point. The higher inlet velocities in nozzle 2 are consistent with its smaller inlet area (Figure 5).

Streamwise velocity contour plots for nozzle 1 are shown in Figure 9, with the PIV measurements capturing the flow development through the duct. The data were obtained with the laser sheet positioned at the midpoint in the z -direction and PIV images acquired with the camera placed orthogonally. The contours show a stratification of streamwise velocity beginning at approximately $x/L = 0.5$, where the flow acceleration becomes pronounced. In the PIV measurements, regions near the inlet and portions of the wall surface toward the exit exhibit excess laser reflections that prevent full resolution of the near-wall boundary layer. These reflections are characteristic of the challenges associated with internal flow PIV measurements.

Figure 10 shows the near-wall resolution capacity within the midsection FOV of nozzle 1. The Gaussian intensity distribution of the laser sheet produces a peak saturation zone with a thickness of approximately $0.0116D_e$. Beyond this zone, flow vectors can be resolved to within half the saturation thickness from the wall. The application of Rhodamine WT and black paint to the nozzle surfaces effectively reduced excess surface reflections, improving the near-wall resolution.

Wall-normal velocity profiles extracted between $x/L = 0.65$ and 0.92 are shown in Figure 11 and reveal the boundary layer development along the curved upper wall of nozzle 1. Within this region, the nozzle reaches its peak streamline curvature, and the flow negotiates a convex surface toward the exit beyond $x/L = 0.75$. The resulting flow turning skews the velocity profiles toward the curved wall, producing a clear distortion relative to the upstream profiles (red box region). The comparison between PIV and CFD data shows close agreement in the wall-normal profiles, with a mean absolute error of approximately 1% within $0.6 < x/L < 0.75$. Approaching the exit ($0.8 < x/L < 1.0$), the error increases, which is attributed to limitations of the steady RANS turbulence model in regions of higher flow gradients.

Calculating boundary layer thickness from skewed velocity profiles in curved flow fields presents significant challenges. Previous studies [27–29] have shown that defining integral boundary layer metrics using the conventional 99% freestream velocity is inadequate in curved velocity fields. In the present work, the boundary layer edge is identified using the Kneedle algorithm [30], which locates the point of maximum curvature in the velocity profile. This approach is necessary because the combined effects of streamwise pressure gradients and wall curvature produce profiles for which a unique edge velocity is not readily identifiable using traditional definitions such as δ_{99} (where $U = 0.99U_\infty$) [31].

At each streamwise station, the algorithm is applied to the velocity profile extracted normal to the wall, with search bounds prescribed based on the local duct geometry to restrict detection to the near-wall region. An example of the detected knee point on a velocity profile at $x/L = 0.71$ for nozzle 1 (CFD data) is shown in Figure 12. The corresponding integral boundary layer quantities are defined as:

$$\delta^* = \int_0^\delta \left(1 - \frac{u(y)}{U_k}\right) dy, \quad (12)$$

$$\theta = \int_0^\delta \frac{u(y)}{U_k} \left(1 - \frac{u(y)}{U_k}\right) dy, \quad (13)$$

where U_k is the velocity at the detected knee point. The shape factor is defined as $H = \delta^*/\theta$.

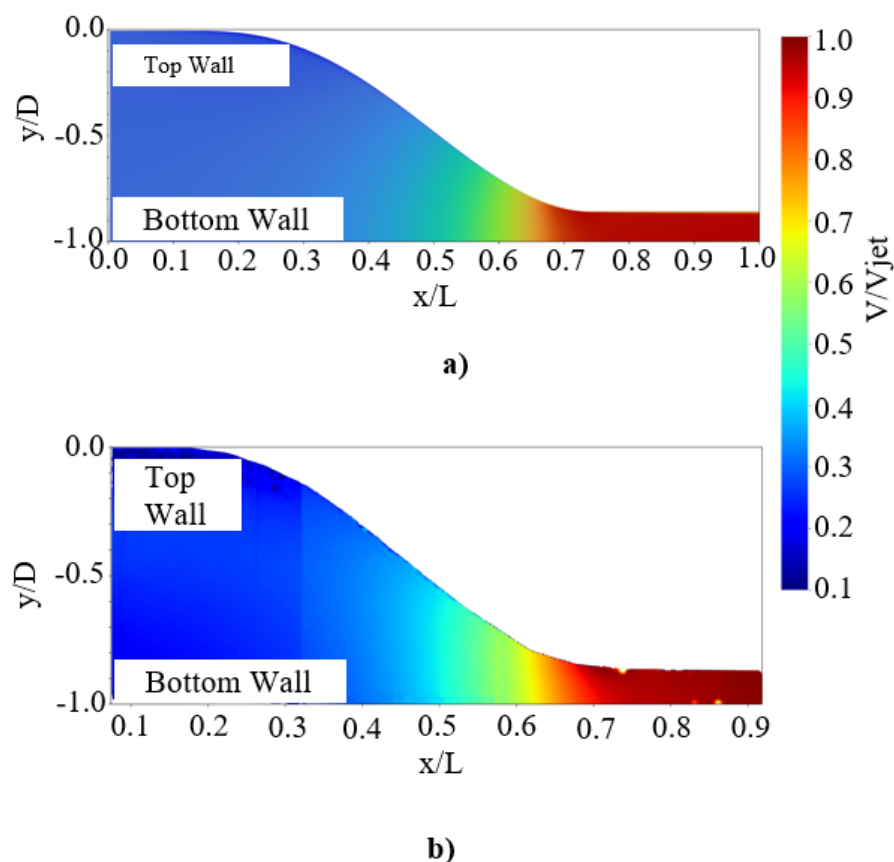


Figure 9. Streamwise velocity contours at exit $M = 0.2$, $Re_{D_c} = 2.50 \times 10^5$ for nozzle 1, $V_{jet} = 70.28$ m/s. (a) CFD velocity contours at $z/D = -0.5$. (b) PIV velocity contours with laser sheet at $z/D = -0.5$.

The boundary layer thicknesses obtained from this analysis at discrete locations are shown in Figure 13. Upstream of the curvature region ($x/L < 0.25$), the boundary layer thickens in a manner consistent with flow over a flat surface. Between $x/L = 0.25$ and 0.68 , the boundary layer thins under the favorable pressure gradient. Beyond $x/L = 0.75$, the boundary layer thickens toward the exit, coinciding with the region of peak wall curvature. The thinning-to-thickening transition between $x/L = 0.68$ and 0.75 corresponds to the region of local adverse pressure gradient ($dp/dx > 0$) identified in the CFD wall-pressure distribution (Figure 14). The corresponding static pressure contour (Figure 14c) shows the spatial structure of the pressure field, including the low-pressure region associated with the flow acceleration through the contraction and the localized adverse pressure gradient near the curved wall. The calculated shape factors in this region are $H < 3$, which,

compared to the two-dimensional laminar separation criterion reported by Miao et al. [32], suggests that the boundary layer remains attached at the present Reynolds number. Should separation occur at higher Reynolds numbers, the adverse pressure gradient region between $x/L = 0.68$ and 0.75 represents the most likely location for its onset.

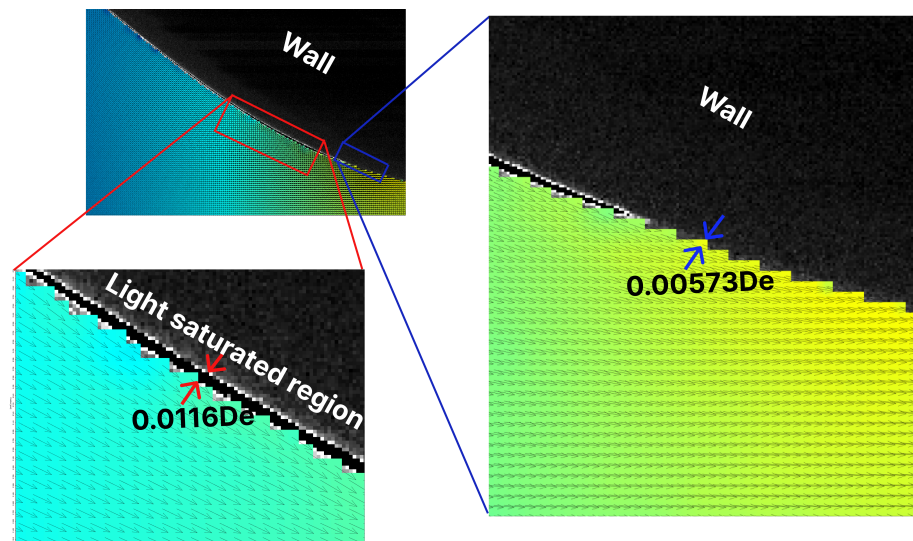


Figure 10. Near-wall PIV resolution for nozzle 1, showing the laser saturation zone and the closest resolved velocity vectors near the wall.

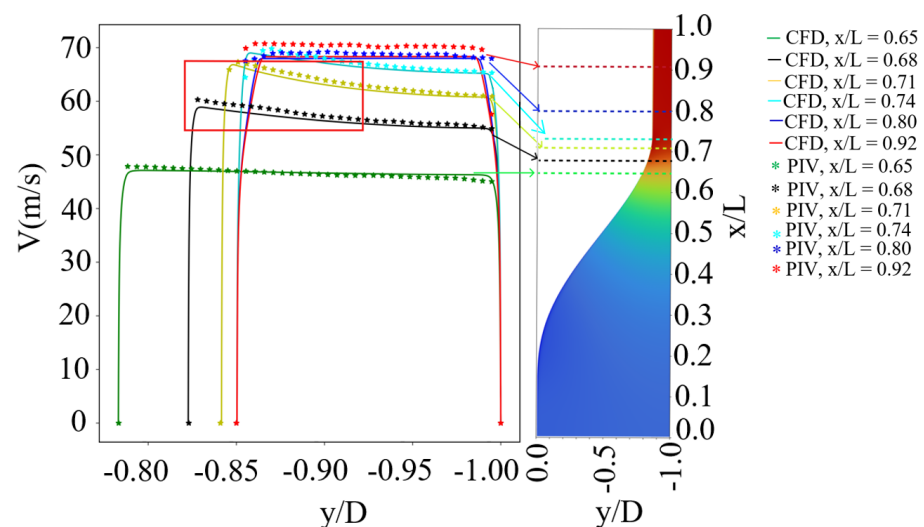


Figure 11. Wall-normal velocity profiles at $x/L = 0.65$ to 0.92 , exit $M = 0.2$, $Re_{De} = 2.50 \times 10^5$ for nozzle 1. The arrows link each velocity profile to its corresponding streamwise location on the contour, identified by color according to the legend.

Although the PIV and CFD boundary layer thicknesses do not agree exactly in magnitude, both methods display consistent trends in the streamwise variation of thinning and thickening (Figure 13). The PIV-derived thicknesses are smaller than the CFD values across both nozzles. Two factors are believed to contribute to this offset. First, the wall location cannot be established with the same precision in the PIV measurements as in the CFD, where the wall is a defined boundary. The near-wall laser saturation zone noted earlier (Figure 10) obscures the exact surface position in the PIV datasets, and because the boundary layer thickness is computed relative to this inferred wall location, any offset in the reference position propagates directly into the calculated thickness. Second, the Kneedle algorithm identifies the boundary layer edge as the point of maximum curvature in the velocity profile; the PIV profiles contain measurement noise that is absent in the

smooth CFD profiles, which can shift the detected edge location even when the underlying velocity distributions are in close agreement. The latter is consistent with the close agreement observed in the wall-normal velocity profiles themselves (Figure 11). Despite these magnitude offsets, the streamwise trends, including the locations of boundary layer thinning and thickening, are captured consistently by both methods. The boundary layer thicknesses along the converging upper wall of nozzle 1 are notably smaller than those along the diverging wall of nozzle 2, consistent with the observations of Bradshaw and Young [33] that streamline curvature affects boundary layer growth differently depending on whether the surface is convex or concave.

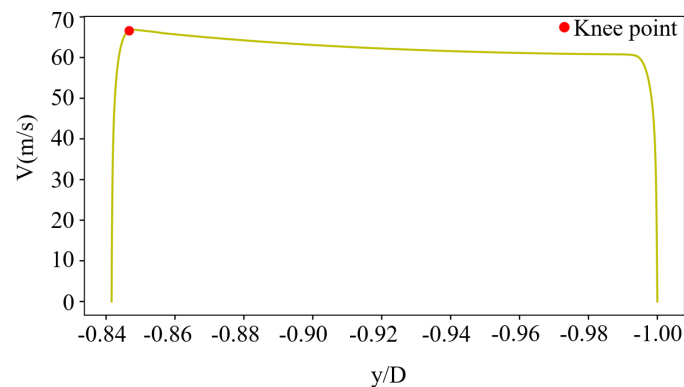


Figure 12. Demonstration of velocity profile knee-point detection using nozzle 1 CFD data at $x/L = 0.71$.

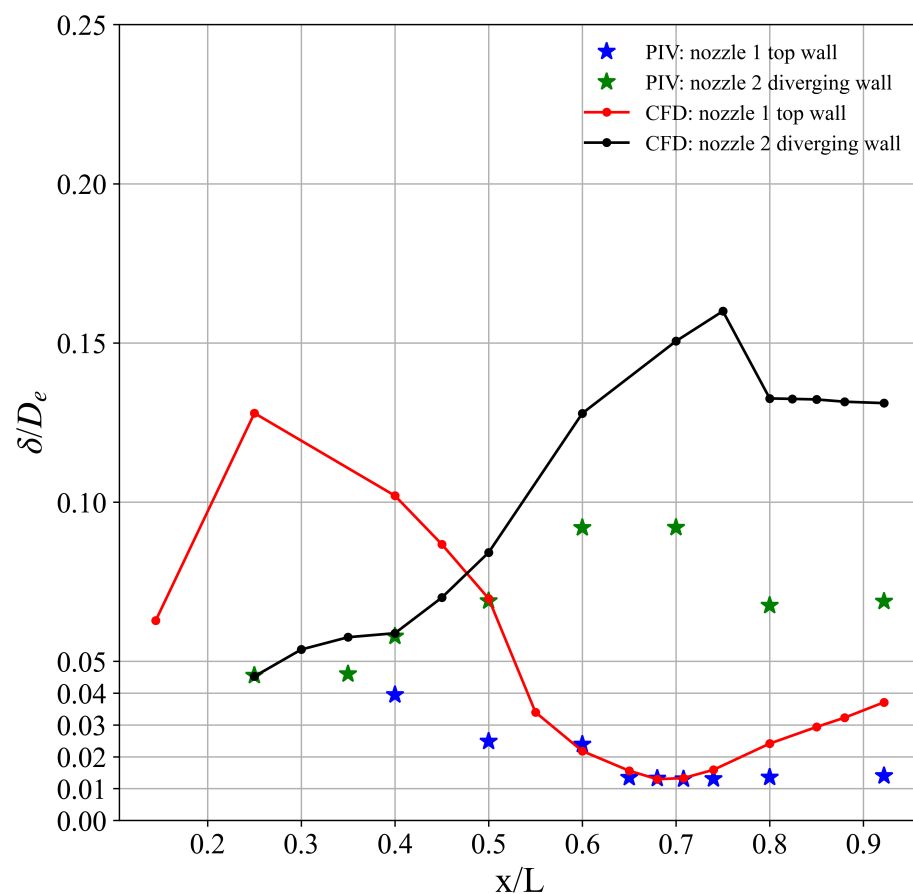


Figure 13. Boundary layer thicknesses for nozzle 1 and nozzle 2, taken at $z/D = -0.5$ and $y/D = -0.86$, respectively.

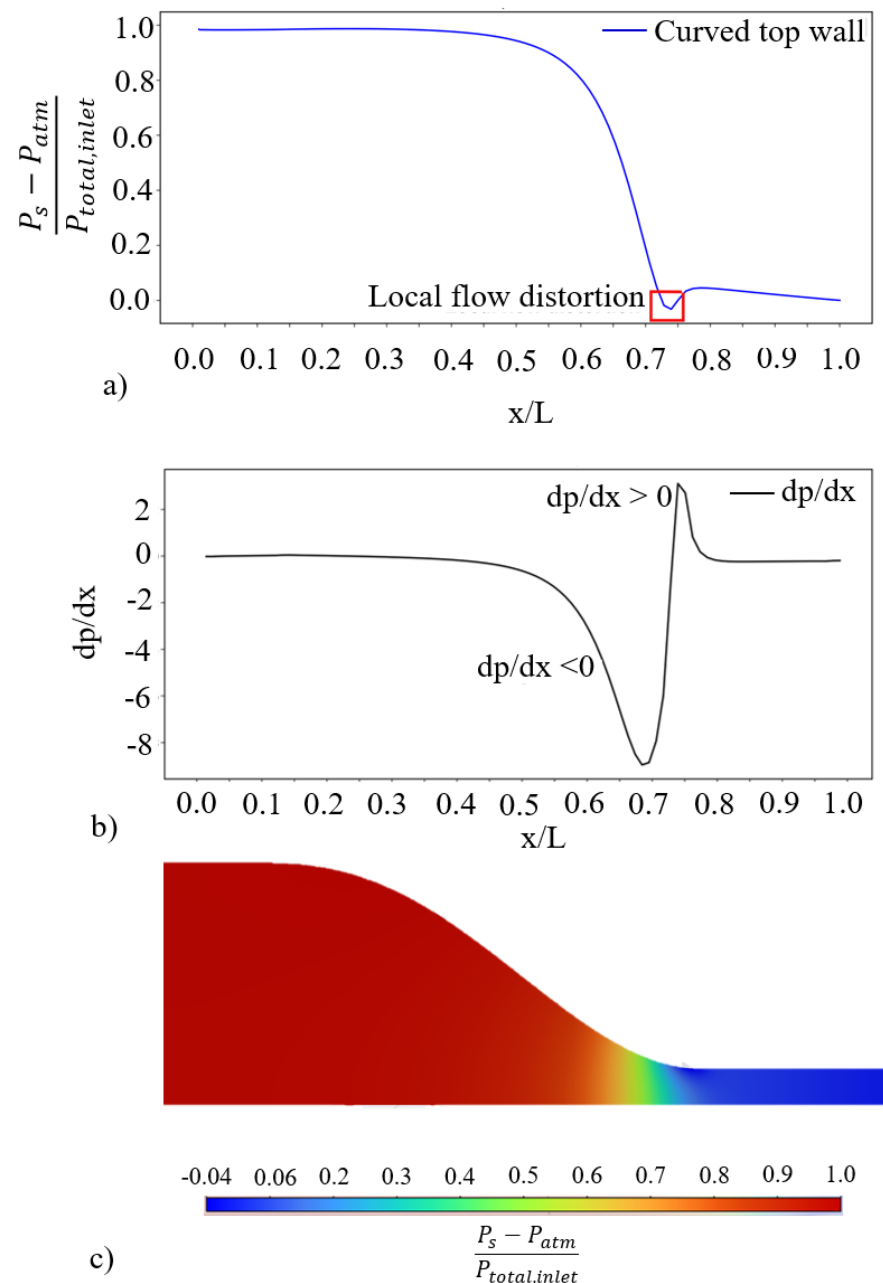


Figure 14. Curved wall static pressure variation from CFD, $Re_{D_e} = 2.50 \times 10^5$ for nozzle 1. (a) Curved wall static pressure variation, (b) streamwise pressure gradient and (c) static pressure contour on the $z/D = -0.5$ midplane.

To further assess the near-exit discrepancy between CFD and experiment, additional measurements were acquired using a pitot-static probe positioned 0.2145 in downstream of the nozzle exit ($x/L = 1.012$). Although this location is downstream of the last PIV measurement plane at $x/L = 0.92$, the pitot probe velocity profile is more consistent with the PIV data than with the CFD prediction at $x/L = 1.0$, with a mean absolute error exceeding 2% relative to the pitot probe data (Figure 15). This agreement between two independent experimental techniques, combined with their shared departure from the CFD prediction, indicates that the RANS model does not fully capture the flow development in the high-gradient region beyond the flow turn near the nozzle exit.

Streamwise velocity contour plots for nozzle 2 are shown in Figure 16, with the PIV measurements again capturing the flow development through the duct. These represent side (Figure 16a,c) and bottom (Figure 16b,d) views of the flow. In Figure 16c, the laser

sheet is positioned at $z/D = -0.25$, and in Figure 16d, it is positioned at $y/D = -0.86$. Both views were acquired to capture the three-dimensional nature of the flow through nozzle 2, with PIV images taken with the camera placed orthogonally (as in Figure 6). Both views show the stratified nature of the velocity field, with peak values occurring in the $x/L \approx 0.8$ region. The bottom views of the PIV and CFD contours reveal a region of locally reduced velocity, labeled as the low-velocity pocket in Figure 16. This pocket forms as the higher-velocity fluid migrates toward the diverging side wall: the lower wall static pressure along the diverging surface accelerates the flow toward that wall and shifts the velocity peak in its direction, leaving a region of comparatively lower velocity on the opposite side of the passage. The pocket is therefore a manifestation of the skewed velocity field produced by the cross-stream pressure imbalance, rather than a region of locally stagnant flow. A similar response of the velocity field to diverging walls was observed in the spatially limited measurements of Whitelaw and Yu [34] within curved diffusers.

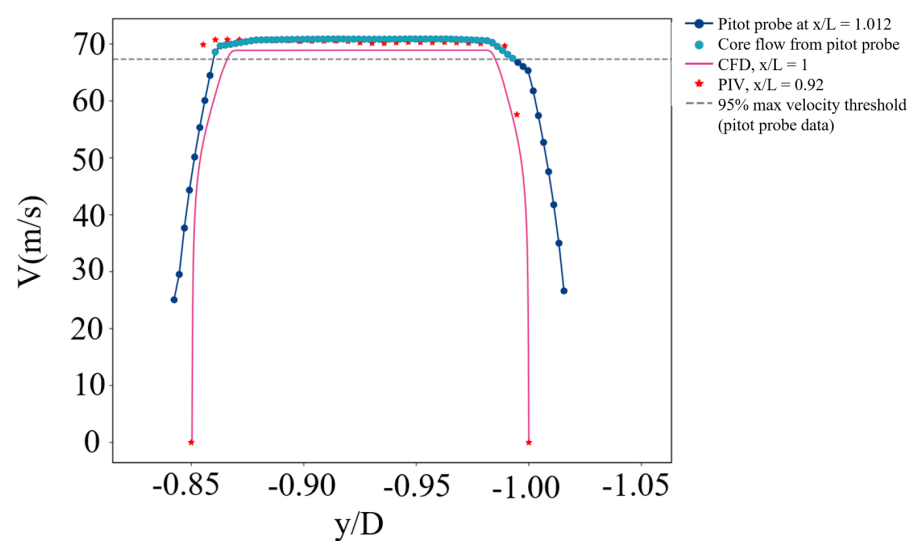


Figure 15. Comparison of velocity profiles at $0.92 < x/L < 1.012$, exit $M = 0.2$, $Re_{D_c} = 2.50 \times 10^5$ for nozzle 1.

The side views reveal flow behavior comparable to the curved converging walls in Figures 8a and 9. Line plots extracted along planes in the bottom and side views (Figure 17) confirm this similarity: there is almost no distinction between the side and bottom view velocity profiles until the exit, where the converging top wall meets the diverging sidewall. In this region, the effective area of nozzle 2 reaches a minimum at $x/L = 0.83$ before the diverging side surface produces an expansion to match the slightly larger exit area. This demonstrates the three-dimensional complexity that the diverging section adds to the overall flow development in nozzle 2. The flow in the side-view plane experiences a predominantly favorable pressure gradient, as in nozzle 1, albeit with a straight converging wall rather than a curved one.

The wall pressure distributions from CFD (Figure 18) provide insight into the flow features observed in the contour plots. Similar local troughs in static pressure are observed for the converging walls of nozzle 1 and nozzle 2, although the trough in nozzle 2 is located further downstream, near $x/L = 0.83$, where the diverging surface returns to the primary flow direction. Two regions of increasing pressure are identified in nozzle 2, each associated with a different wall. The first appears along the diverging side wall at approximately $0.4 < x/L < 0.7$, while the second appears along the converging top wall near $x/L = 0.83$, where the contraction ends. In the first region, the bulk flow is primarily driven by favorable pressure gradients imposed by the straight converging top wall. Along the diverging

surface, however, the streamwise pressure gradient is locally adverse ($dp/dx > 0$), which decelerates the near-wall fluid and promotes boundary layer growth. This region along the diverging surface coincides with the location of peak boundary layer thickness and will be discussed further. Separately, the wall static pressures along the diverging surface are lower than those along the converging wall at the same streamwise locations (Figure 18), producing a cross-stream pressure imbalance that shifts the velocity peak in the wall-normal profiles toward the diverging wall. Upstream of $x/L = 0.83$, the static pressures along the bottom flat wall (in green) and straight converging walls decrease monotonically, with the converging wall pressures (in black) remaining slightly higher. These two adverse pressure gradient regions distinguish nozzle 2 from nozzle 1, which exhibits only a single such region associated with the peak curvature of its converging top wall (Figure 14). The presence of an adverse pressure gradient on the diverging side wall, in addition to that on the converging top wall, represents the primary distinction in the pressure development between the two configurations.

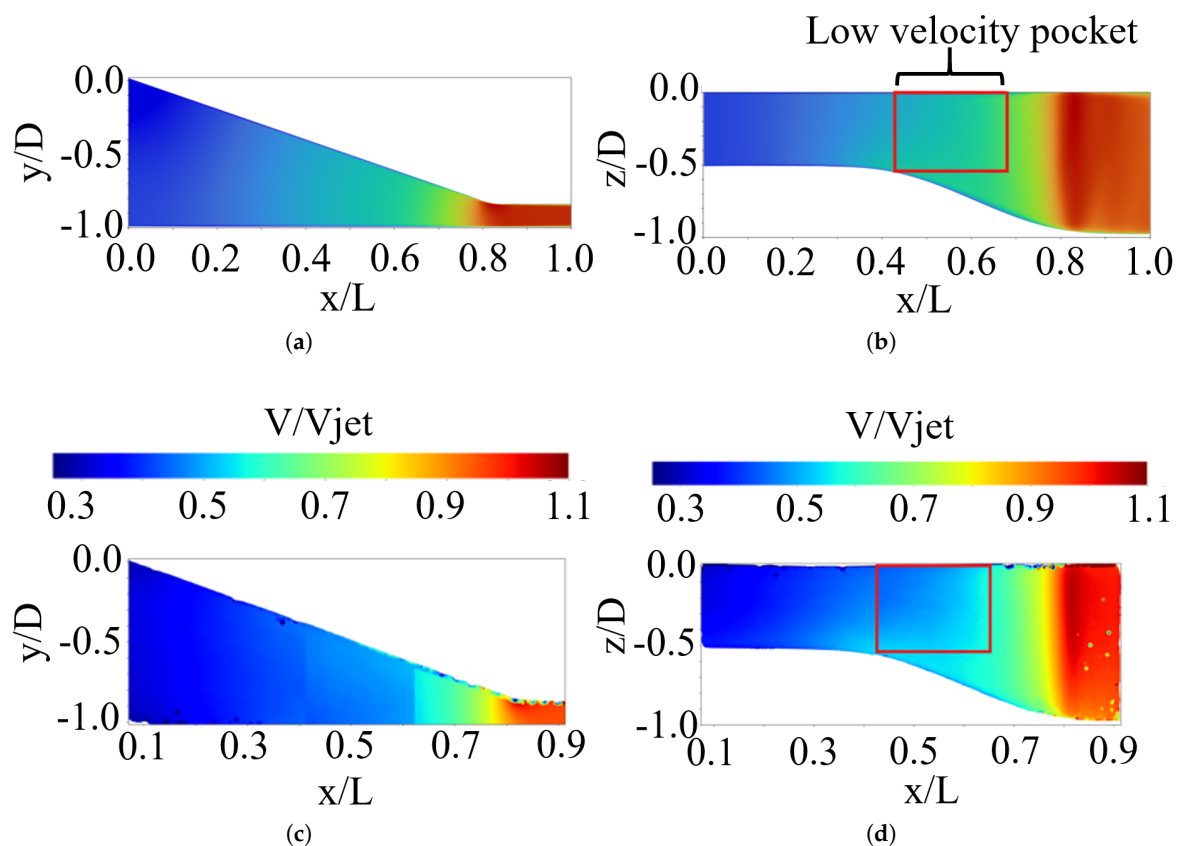


Figure 16. (a) CFD velocity contours at $z/D = -0.25$. (b) CFD velocity contours at $y/D = -0.86$. (c) PIV velocity contours with laser sheet at $z/D = -0.25$. (d) PIV velocity contours with laser sheet at $y/D = -0.86$, for nozzle 2, $V_{jet} = 71.61$ m/s.

Wall-normal velocity profiles at $x/L = 0.25$ to 0.92 in the $y/D = -0.86$ plane are shown in Figure 19 for nozzle 2. At the $0.35 < x/L < 0.7$ locations, the velocity peak in the wall-normal profiles shifts toward the diverging wall in response to the onset of the diverging surface. Collectively, this produces the lower velocity pocket regions shown earlier in Figure 16. Beyond $x/L = 0.7$, the shift reverses as the curved side surface returns to the primary flow direction. The PIV data also captures a flow distortion (in the black box) at $x/L > 0.8$ ($z/D = -0.9$), where the diverging surface fully re-aligns with the primary flow direction. The CFD data does not capture this distortion, consistent with the near-exit RANS limitations observed for nozzle 1.

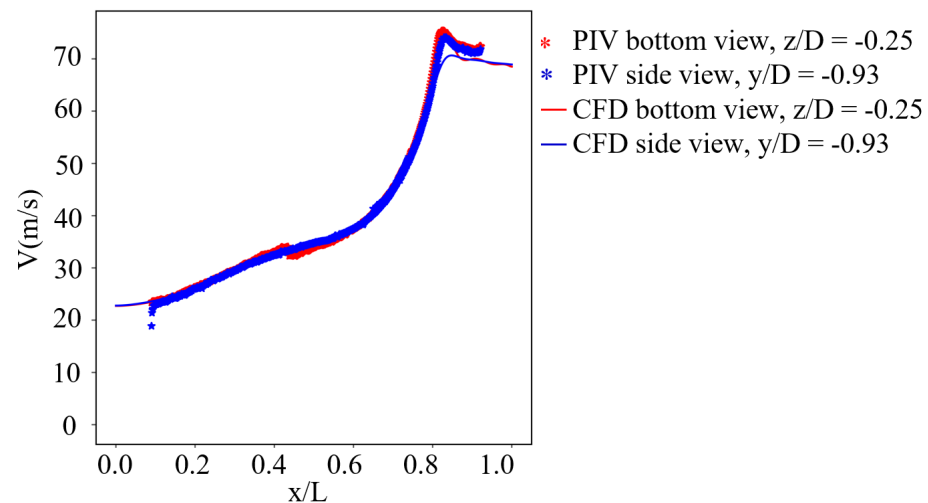


Figure 17. Line plots along $z/D = -0.25$ and $y/D = -0.93$ for nozzle 2.

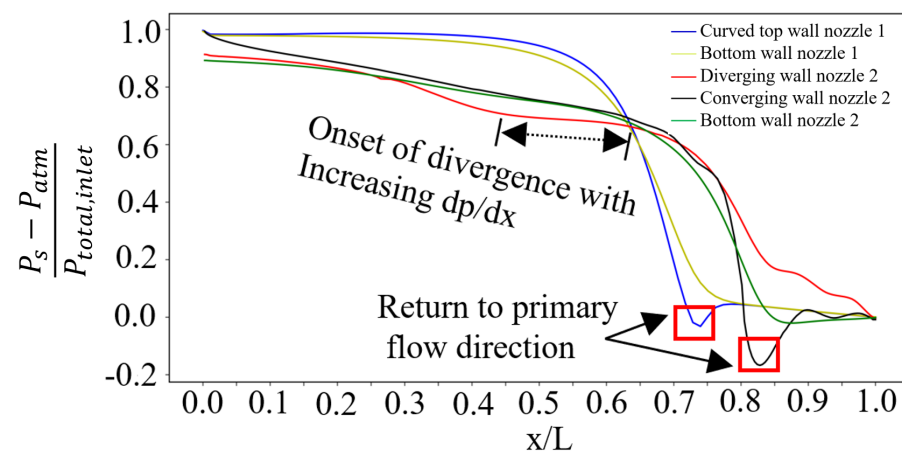


Figure 18. Wall static pressure variation for nozzle 1 and nozzle 2 from CFD.

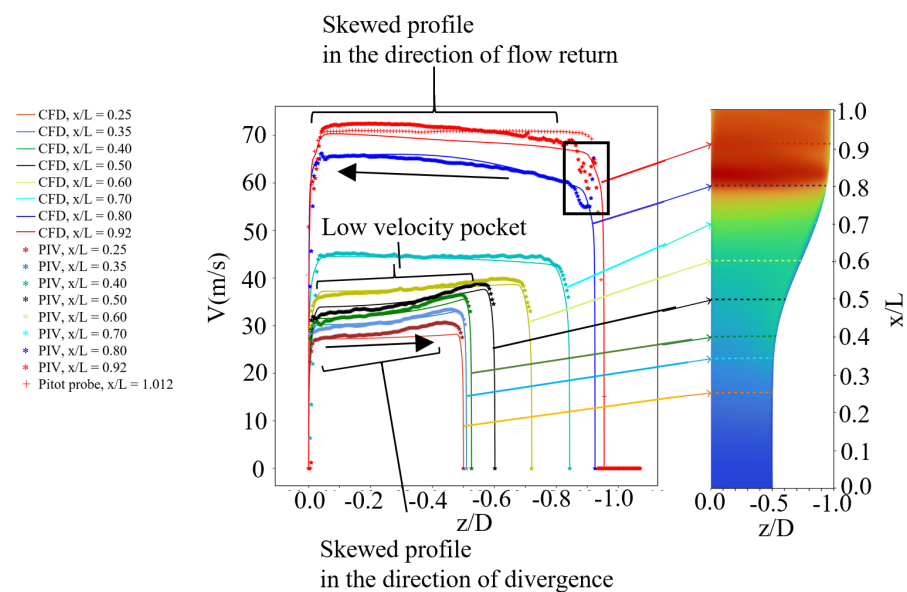


Figure 19. Wall-normal velocity profiles at $x/L = 0.25$ to 0.92 , exit $Ma = 0.2$, exit $Re_{De} = 2.50 \times 10^5$ for nozzle 2.

The boundary layer development captured in Figure 19 indicates a growing boundary layer along the diverging surface from $x/L = 0.4$ to 0.74 . Beyond $x/L = 0.74$, the boundary

layer thickness decreases (Figure 13), consistent with the return of the curved surface to the primary flow direction. In the PIV measurements, this peak in boundary layer thickness occurs at $x/L = 0.7$, with a measured value of approximately $\delta/D_e = 0.092$. Given the test section length of $L = 439.4$ mm and the exit flow Reynolds number, the boundary layer along the diverging surface of nozzle 2 is significantly thicker than that along the curved converging surface of nozzle 1. The boundary layer thickness measurements from both CFD and PIV exhibit similar trends, although the CFD results indicate an overestimation of thickness values beyond $x/L = 0.5$.

Figure 20 illustrates the near-wall resolution capacity within the midsection FOV of nozzle 2. As with nozzle 1, the laser saturation zone is small (approximately $0.011D_e$), and flow vectors are resolved close to the wall beyond this zone. The PIV and CFD velocity profiles are in close agreement in the upstream regions, with mean absolute errors of approximately 2%, increasing to approximately 6% toward the exit.

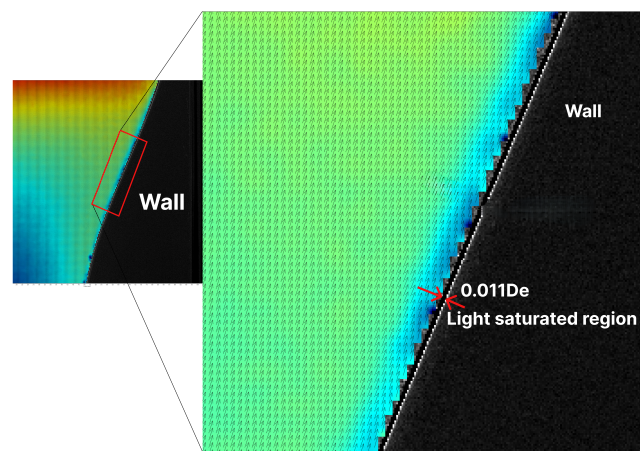


Figure 20. Near-wall PIV resolution for nozzle 2, showing the laser saturation zone and the resolved near-wall velocity vectors.

Nozzle Performance Characteristics

The preceding results show that both nozzles exhibit local adverse pressure gradients in the presence of a globally favorable pressure gradient, with nozzle 2 featuring two such regions. To assess the impact of these non-ideal flow paths on overall nozzle performance, total pressure recovery and discharge coefficient are evaluated for both nozzles using CFD and experimental pitot probe measurements at the nozzle exit. Nozzle total pressure recovery is defined as the ratio of the area-weighted average of total pressure at the outlet plane to the inlet total pressure:

$$\pi_n = \frac{P_{t,\text{outlet}}}{P_{t,\text{inlet}}} \quad (14)$$

The discharge coefficient is defined as the ratio of the actual exit mass flow rate (measured or computed) to the ideal mass flow rate for the given nozzle exit area:

$$C_D = \frac{\dot{m}_{\text{outlet}}}{\dot{m}_{\text{outlet, ideal}}} \quad (15)$$

The discharge coefficient quantifies the reduction in effective exit area due to boundary layer displacement thickness. The experimental mass flow rate was measured using the orifice plate located upstream of the test facility, as described previously.

The resulting values are shown in Table 4 for both nozzles. The experimental pitot probe measurements yield identical total pressure recovery values of approximately 99%

for both nozzles, while the CFD predicts notably lower values of approximately 92%. This discrepancy is attributed to the spatial resolution of the pitot probe grid, which does not resolve the thin near-wall and corner loss regions that are captured in the CFD. As shown in the exit total pressure contours (Figure 21), the CFD reveals total pressure deficits concentrated along the walls and in the corners, particularly in nozzle 1, where the contours exhibit greater distortion with some symmetry about the minor axis. The pitot probe measurements, by contrast, primarily sample the core flow region where total pressure is near its inlet value, and only partially capture the corner losses (indicated by the blue boxes in Figure 21b). Consequently, the experimental π_n reflects predominantly the core flow recovery and underestimates the integrated wall and corner losses.

The discharge coefficient provides a more reliable comparison because it is based on the orifice plate mass flow rate, which accounts for the entire flow rather than depending on local probe resolution. Both CFD and experiment show lower discharge coefficients for nozzle 1 than for nozzle 2 (Table 4), with agreement within approximately 1%. Despite nozzle 2 exhibiting thicker boundary layers at the measured plane (Figure 13), its higher discharge coefficient indicates that the integrated losses across the full exit area are lower than in nozzle 1, reflecting the three-dimensional nature of the loss distribution.

Table 4. Nozzle performance parameters.

Parameter	CFD	Experimental
Nozzle 1		
π_n	91.88%	99.04%
C_D	95.40%	94.39%
Nozzle 2		
π_n	92.23%	99.04%
C_D	95.63%	96.22%

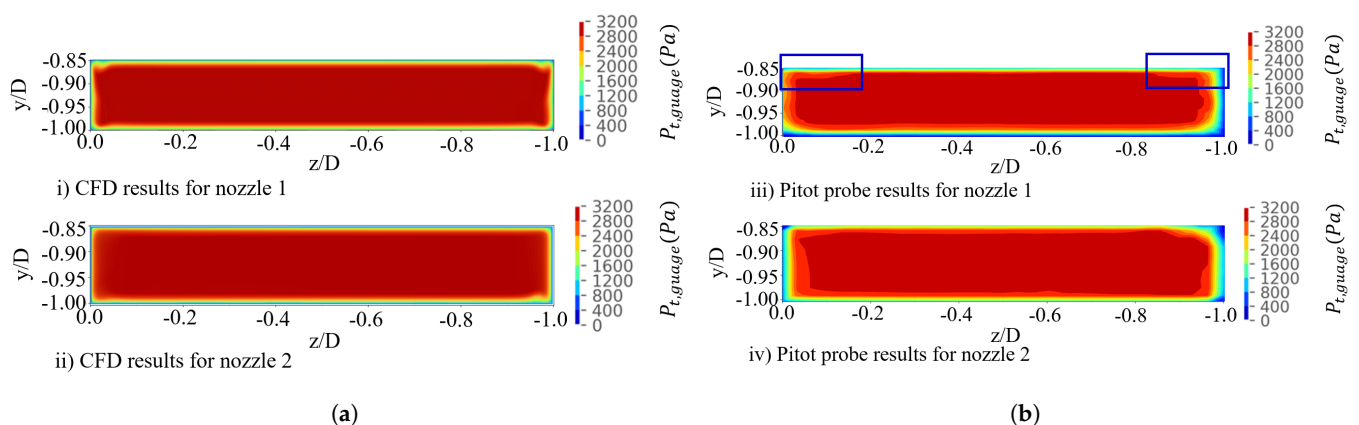


Figure 21. Exit total pressure variation for nozzle 1 and nozzle 2. (a) CFD results. (b) Pitot probe results.

The results allow a direct comparison of the two nozzle designs. The two configurations isolate distinct flow regimes: nozzle 1, with its curved converging wall, subjects the flow to a predominantly favorable pressure gradient and a single localized adverse pressure gradient region near the peak curvature, while nozzle 2, with its diverging sidewall, introduces two adverse pressure gradient regions and a strongly three-dimensional flow field. These differences are evident in the centreline velocity development (Figure 8), where nozzle 1 accelerates gradually through a late contraction and nozzle 2 reaches its velocity peak earlier before the diverging surface re-aligns with the primary flow direction. The

wall pressure distributions (Figure 18) and boundary layer thicknesses (Figure 13) further distinguish the two: nozzle 2 develops substantially thicker boundary layers along its diverging wall, and its diverging sidewall produces a distinct low-velocity pocket through the cross-stream migration of the bulk flow, a three-dimensional feature not present in nozzle 1. Despite these stronger local flow distortions, nozzle 2 exhibits the higher discharge coefficient, indicating that its integrated exit losses are lower than those of nozzle 1, whose curved converging wall produces greater total pressure distortion at the exit plane. Both designs maintain attached flow and discharge coefficients above 94% under the present conditions, reflecting the relatively benign degree of shape transition in each.

4. Conclusions

Internal flow measurements were obtained using planar PIV in two shape-transitioning nozzle ducts and compared with steady RANS $k-\omega$ SST simulations. Nozzle 1 transitions from a square inlet to a rectangular exit, while nozzle 2 transitions from a rectangular inlet of approximately half the area to a similar rectangular exit. Both flows are driven by globally favorable pressure gradients, with nozzle 1 featuring a curved converging wall and nozzle 2 incorporating a diverging sidewall that introduces localized adverse pressure gradients.

Both nozzles exhibit gradual flow acceleration with peak streamwise velocities near the exit. In nozzle 2, the velocity peaks at $x/L \approx 0.83$ due to the diverging sidewall, which locally increases the effective area beyond this point. In nozzle 1, the boundary layer thins between $x/L = 0.25$ and 0.68 under the favorable pressure gradient, then thickens beyond $x/L = 0.75$ coinciding with the region of peak wall curvature and local adverse pressure gradient. The calculated shape factors suggest the boundary layer remains attached at the present Reynolds number; should separation occur at higher Reynolds numbers, the $x/L = 0.68$ to 0.75 region represents the most likely location.

In nozzle 2, the adverse streamwise pressure gradient along the diverging wall promotes boundary layer growth from $x/L = 0.4$ to 0.74 , producing thicker boundary layers than those along the converging wall of nozzle 1. A cross-stream pressure imbalance shifts the velocity peak toward the diverging wall, collectively creating the low-velocity pocket observed in the contour plots. The PIV data captures a flow distortion at $x/L > 0.8$ where the diverging surface re-aligns with the primary flow direction, which the RANS simulation does not reproduce.

Wall-normal velocity profiles show agreement between PIV and CFD within approximately 1–2% mean absolute error in the upstream and mid-duct regions for both nozzles. This error increases toward the exit, reaching approximately 6% for nozzle 2, attributed to limitations of the steady RANS model in regions of high flow gradients. Independent pitot probe measurements at the nozzle 1 exit confirm the PIV trends over the CFD prediction in this region. Discharge coefficients from CFD and experiment agree within approximately 1% for both nozzles, with nozzle 1 exhibiting greater integrated losses than nozzle 2 despite thinner boundary layers at the measured plane, indicating that the loss distribution is three-dimensional.

The relatively benign degree of convergence and divergence in both nozzle geometries results in discharge coefficients above 94% and attached flow throughout the operating conditions tested. Future work will focus on nozzle pathways with more aggressive shape transitions to determine the onset of flow separation and its impact on nozzle performance.

Author Contributions: Conceptualization, V.O. and D.C.; Data curation, V.O. and K.G.; Formal analysis, V.O.; Investigation, V.O.; Methodology, V.O. and D.C.; Project administration, D.C.; Resources, D.C.; Software, D.C.; Supervision, D.C.; Validation, V.O.; Visualization, V.O.; Writing—Original draft, V.O.; Writing—Review and editing, V.O. and D.C. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The raw data supporting the conclusions of this article will be made available by the authors on request.

Acknowledgments: Special thanks to Russell DiMicco, whose expertise in facility design and assembly was helpful in setting up the facility used in this body of work. We also acknowledge Ryan O'Rorke, Ben Rupe, Piero Paialunga and Neil Castelino for their valuable help during the testing and data acquisition phase of this work.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

L	Length of Nozzle from Entrance to Exit
D_e	Equivalent Diameter based on nozzle exit, equal for nozzle 1 and 2
AR_e	Exit Aspect Ratio
L/D_e	Length-to-Diameter ratio
D	Inlet Hydraulic Diameter of nozzle 1
AR	Area Ratio
FOV	Field of View
PIV	Particle Image Velocimetry
dt	time between laser pulses

References

1. Ferrar, A.M. Measurements of Flow in Boundary Layer Ingesting Serpentine Inlets. Ph.D. Thesis, Virginia Tech, Blacksburg, VA, USA, 2011.
2. Wu, C.; Farokhi, S.; Taghavi, R. A computational study of advanced exhaust system transition ducts with experimental validation. *AIAA J.* **1986**, *24*, 1872, 1873. [[CrossRef](#)]
3. Abbott, J.M.; Anderson, B.H.; Rice, E.J. *Inlets, Ducts, and Nozzles*; Technical Report N88-15791; NASA Lewis Research Center: Cleveland, OH, USA, 1987.
4. Patrick, W.P.; McCormick, D.C. Circular-to-Rectangular Duct Flows A Benchmark Experimental Study. *SAE Trans.* **1987**, 963–978.
5. Davis, D.O.; Gessner, F. Experimental investigation of turbulent flow through a circular-to-rectangular transition duct. *AIAA J.* **1992**, *30*, 367–375. [[CrossRef](#)] [[PubMed](#)]
6. Crowe, D.S.; Martin, C.L. Effect of geometry on exit temperature from serpentine exhaust nozzles. In *Proceedings of the 53rd AIAA Aerospace Sciences Meeting*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2015; p. 1670. [[CrossRef](#)]
7. Shan, Y.; Zhou, X.; Tan, X.; Zhang, J.; Wu, Y. Parametric design method and performance analysis of double S-shaped nozzles. *Int. J. Aerosp. Eng.* **2019**, *2019*, 4694837. [[CrossRef](#)]
8. Sun, X.L.; Wang, Z.X.; Zhou, L.; Shi, J.W.; Liu, Z.W. Experimental and computational investigation of double serpentine nozzle. *Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng.* **2015**, *229*, 2035–2050. [[CrossRef](#)]
9. Sun, P.; Zhou, L.; Wang, Z.; Shi, J. Influences of geometric parameters on serpentine nozzles for turbofan. *Aerosp. Sci. Technol.* **2023**, *136*, 108224. [[CrossRef](#)]
10. Hui, Z.; Shi, J.; Zhou, L.; Wang, Z.; Liu, Y. Experimental investigation of serpentine nozzles for turbofan. *Aerosp. Sci. Technol.* **2021**, *117*, 106892. [[CrossRef](#)]
11. Brear, M.J.; Warfield, Z.; Mangus, J.F.; Braddom, S.; Paduano, J.D.; Philhower, J.S. Flow separation within the engine inlet of an uninhabited combat air vehicle (UCAV). *J. Fluids Eng.* **2004**, *126*, 266–272. [[CrossRef](#)]
12. Tanguy, G.; MacManus, D.G.; Zachos, P.; Gil-Prieto, D.; Garnier, E. Passive flow control study in an S-duct using stereo particle image velocimetry. *AIAA J.* **2017**, *55*, 1862–1877. [[CrossRef](#)]

13. Zachos, P.K.; MacManus, D.G.; Prieto, D.G.; Chiereghin, N. Flow distortion measurements in convoluted aeroengine intakes. *AIAA J.* **2016**, *54*, 2819–2832. [[CrossRef](#)]
14. Taylor, A.; Whitelaw, J.H.; Yianneskis, M. *Turbulent Flow in a Square-to-Round Transition*; Technical Report; NASA: Washington, DC, USA, 1981.
15. Sobota, T.H.; Marble, F.E. Swirling flows in an annular-to-rectangular transition section. *J. Propuls. Power* **1989**, *5*, 334–340. [[CrossRef](#)] [[PubMed](#)]
16. Crowe, D.S.; Martin, C.L. Hot streak characterization in serpentine exhaust nozzles. In *Proceedings of the 52nd AIAA/SAE/ASEE Joint Propulsion Conference*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2016; p. 4502. [[CrossRef](#)]
17. Crowe, D.S.; Martin, C.L., Jr. Hot streak characterization of high-performance double-serpentine exhaust nozzles at design conditions. *J. Propuls. Power* **2019**, *35*, 501–511.
18. Crowe, D.S.; Depaola, R. A Method to Compute Thermal Distortion in Non-Circular Ducts. In *Proceedings of the AIAA Scitech 2019 Forum*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2019; p. 1449. [[CrossRef](#)]
19. Gil-Prieto, D.; Zachos, P.K.; MacManus, D.G.; McLelland, G. Unsteady characteristics of S-duct intake flow distortion. *Aerosp. Sci. Technol.* **2019**, *84*, 938–952. [[CrossRef](#)]
20. Onoja, V.A.; Cuppoletti, D.R. PIV Measurements of Internal Flows through Nozzles with Curvature and Shape Transition. In *AIAA SciTech 2023 Forum*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2023; p. 0715. [[CrossRef](#)]
21. Bell, J.H.; Mehta, R.D. *Contraction Design for Small Low-Speed Wind Tunnels*; Technical Report; NASA: Washington, DC, USA, 1988.
22. ANSYS, Inc. *ANSYS Fluent Theory Guide, Release 2021 R2*; ANSYS, Inc.: Canonsburg, PA, USA, 2021.
23. Menter, F.R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* **1994**, *32*, 1598–1605. [[CrossRef](#)] [[PubMed](#)]
24. Onoja, V.A.; Ganeshan, K.; Cuppoletti, D.R.; Hyun Ryu, Y.; Darmofal, D.L.; Galbraith, M.C.; Allmaras, S.R. Validation Quality Measurements and Computations of Flow in Shape Transitioning Nozzles. In *Proceedings of the AIAA Aviation Forum and Ascend 2025*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2025; p. 3333.
25. Roache, P.J. Perspective: A method for uniform reporting of grid refinement studies. *J. Fluids Eng.* **1994**, *116*, 405–413. [[CrossRef](#)]
26. NPARC Alliance CFD Verification and Validation. Examining Spatial (Grid) Convergence. 2021. Available online: <https://www.grc.nasa.gov/www/wind/valid/tutorial/spatconv.html> (accessed on 20 May 2026).
27. Meroney, R.; Bradshaw, P. Turbulent boundary-layer growth over a longitudinally curved surface. *AIAA J.* **1975**, *13*, 1448–1453. [[CrossRef](#)] [[PubMed](#)]
28. Gibson, M.; Jones, W.; Younis, B. Calculation of turbulent boundary layers on curved surfaces. *Phys. Fluids* **1981**, *24*, 386–395. [[CrossRef](#)]
29. Barlow, R.S.; Johnston, J.P. Structure of a turbulent boundary layer on a concave surface. *J. Fluid Mech.* **1988**, *191*, 137–176. [[CrossRef](#)]
30. Satopaa, V.; Albrecht, J.; Irwin, D.; Raghavan, B. Finding a “kneedle” in a haystack: Detecting knee points in system behavior. In *Proceedings of the 2011 31st International Conference on Distributed Computing Systems Workshops*; IEEE: Piscataway, NJ, USA, 2011; pp. 166–171.
31. Griffin, K.P.; Fu, L.; Moin, P. General method for determining the boundary layer thickness in nonequilibrium flows. *Phys. Rev. Fluids* **2021**, *6*, 024608. [[CrossRef](#)]
32. Miao, J.J.; Leu, T.S.; Chou, J.H.; Lin, S.; Lin, C. Flow distortion in a circular-to-rectangular transition duct. *AIAA J.* **1990**, *28*, 1447–1456. [[CrossRef](#)]
33. Bradshaw, P.; Young, A. *Effects of Streamline Curvature on Turbulent Flow*; Agard: Paris, France, 1973.
34. Whitelaw, J.; Yu, S. Turbulent flow characteristics in an S-shaped diffusing duct. *Flow Meas. Instrum.* **1993**, *4*, 171–179. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.