

Design and Optimisation of a Mach 2.5 Wind Tunnel Nozzle

Miguel Moreno¹, Matteo Migliorini², Pavlos K. Zachos³, Anthony Haslam⁴ and David MacManus⁵

School of Aerospace, Transport and Manufacturing, Cranfield University, Bedfordshire, MK430AL, United Kingdom

The paper presents a methodology for the numerical design and optimization of a distortion-free two-dimensional Mach 2.5 nozzle based on a parametric model. The non-uniformities generated at the Mach wave reflections downstream of the nozzle throat that the Method of Characteristics only partially addresses are minimized. The spatial discretization of the domain is integrated with the boundary layer analysis for fast and robust data processing, especially in the final viscous sublayers in the transition regions within the bulk of the fluid. The flow patterns and corner flows of the supersonic nozzle are assessed via three-dimensional high-fidelity computational fluid dynamics. As a result, a fast workflow for nozzle design to meet prescribed flow quality requirements is herein illustrated.

I. Nomenclature

A	=	area (m^2)
a, b, c	=	arbitrary constants used during intermediate calculations
d_1, d_2	=	control point distance parameters
H	=	height (m)
i, j	=	linear indices
I	=	inflection point
J	=	maximum absolute derivative point
L	=	length (m)
$\dot{m}$	=	mass flow (kg/s)
M	=	Mach number
n	=	number of occurrences or curve order
O	=	origin of coordinates
P	=	total pressure (Pa)
R	=	universal gas constant ($\text{J}/(\text{kg}\cdot\text{K})$), coefficient of determination (R^2), radius
t	=	internal curve parameter
T	=	total temperature (K)
u, v, w	=	velocity components in (x, y, z)
u	=	velocity magnitude (m/s) equal to $\ u, v, w\ $
u'	=	first spatial derivative of u perpendicular to the wall distance
u''	=	second spatial derivative of u perpendicular to the wall distance

¹ Doctoral Researcher, Centre for Propulsion and Thermal Power Engineering.

² Post-doctoral Research Fellow, Centre for Propulsion and Thermal Power Engineering.

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

⁴ Senior Lecturer in Mechanical Design of Turbomachinery, Centre for Propulsion and Thermal Power Engineering.

⁵ Professor in Propulsion Integration, Centre for Propulsion and Thermal Power Engineering, AIAA Member.

x, y, z = cartesian coordinate system (m)
 y^+ = non-dimensional wall distance

Subscripts

avg = average
 max = maximum
 min = minimum
 ss = subsonic
 th = throat
 ws = working section

Greek Letters

∞ = freestream
 γ = heat capacity ratio
 δ = boundary layer thickness where $u = 0.99U_\infty$
 κ = curvature
 σ = standard deviation
 ψ = distortion metric

Abbreviations

CFD = Computational Fluid Dynamics
DSE = Design Space Exploration
MoC = Method of Characteristics
SST = Shear Stress Transport

II. Introduction

Converging-diverging nozzles have been universally used in supersonic wind tunnels for a wide range of high-speed flow applications. Supersonic nozzle design methods have been extensively presented in the literature [1][2]. A typical design approach relies on the Method of Characteristics (MoC), where a numerical solution of hyperbolic partial-differential equations produces the specification of a supersonic nozzle contour via characteristic lines [3] originating from the nozzle throat, shown in the region 2 of Figure 1. These characteristics result in reflective Mach waves or expansion fan waves generated in the expansion section of the nozzle (region 3 of Figure 1) as the flow is being locally turned in a direction parallel to the nozzle walls, and propagate further downstream along the supersonic section nozzle in the straightening section of the nozzle (region 4 in Figure 1). MoC based design methods were previously applied to two-dimensional [3] or axisymmetric [4] nozzle configurations, with and without the presence of center bodies [5].

As the contour of the supersonic nozzle segment between the throat and the inflection point is not specified through the reflection of any characteristic line, arbitrary continuous-curvature profiles are often used in wind tunnels constrained by aspects such as variable-geometry flexible plate designs that may be desired to be used, as previously shown by Sivells [6]. The expansion section of the nozzle (region 3 in Figure 1) induces the greatest magnitude of flow turning in the supersonic portion, resulting in expansion fans or Mach waves that reflect downstream [3]. The shape of this expansion section determines the locations and angularity of the Mach wave reflections. A typical nozzle design approach relies on the MoC to specify this first portion of the nozzle [1]. However, in cases where flow quality is of critical importance, further optimization of the MoC-based nozzle design is required to mitigate propagation of the Mach wave reflections through the downstream sections of the nozzle that can be detrimental to the flow quality delivered to the working section of the tunnel and are only partially addressed in MoC-based designs. Finally, throat area corrections are often required to fine adjust the Mach number at the exit of the nozzle, which is influenced by the growth of the post-throat boundary layers. Empirical boundary layer growth data can be used to displace the MoC profile to deliver reasonable-quality flows [1][7] or CFD driven design iterations have been previously used to calculate the boundary layer characteristics and produce throat area corrections based on these predictions [2].

In this paper, a supersonic nozzle design and optimization workflow is presented with CFD in the loop. The baseline design is defined via the application of the MoC. Subsequently, this baseline design is further optimized to improve the quality of the supersonic flow in terms of its uniformity and angularity at the interface plane between the

nozzle exit and the working section of a new supersonic test rig. Finally, the throat area of the optimized nozzle configuration is iteratively corrected via viscous CFD simulations to fine adjust the target onset Mach number accounting for boundary layer growth.

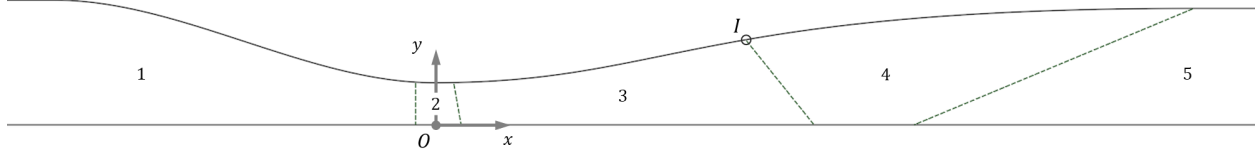


Figure 1: Nozzle regions. From left to right: 1-contraction, 2-throat, 3- expansion, 4-straightening ('Busemann') and 5-uniform flow sections. Reference system OXY and inflection point I shown for reference.

III. Methodology

A. Baseline Nozzle Design

The nozzle is part of a wind tunnel assembly with target operating Mach numbers of up to 3.0. The nozzle design presented herein targets an operating Mach number of 2.5. It is envisaged that the developed nozzle will later become part of a computational model to investigate the aerodynamics of porous surfaces as discussed by Willis [8]. It was therefore of interest to produce a supersonic nozzle to match this previous setup in terms of inlet flow conditions and dimensions of the test section. Using isentropic flow relations and the dimensions and operating conditions reported in Table 1 [8], the following mass flow value is calculated for the working section as per Equation 1 [9]. The Mach number at the subsonic portion of the nozzle was assumed to be 0.2, which is commonly used to mitigate excessive pressure losses in this section of the assembly [9].

$$\dot{m} = \frac{P \cdot A \cdot M}{\left(\frac{R \cdot T}{\gamma}\right)^{\frac{1}{2}} \left(1 + \frac{(\gamma - 1)}{2} \cdot M^2\right)^{\frac{\gamma + 1}{2 \cdot (\gamma - 1)}}} = 14.3 \text{ kg/s} \quad (1)$$

Table 1: Reference flow properties and working section dimensions for nozzle design.

Inlet total pressure	P_0	172,400 Pa
Inlet total temperature	T_0	292 K
Working section Mach number	M_{ws}	2.46
Working section cross section area	A_{ws}	$0.305 \text{ m} \times 0.305 \text{ m}$

Where the properties P_0 and T_0 are the total pressure and temperatures measured at the subsonic nozzle inlet, and the Mach number M_{ws} is measured at the working section located downstream of the supersonic nozzle outlet (Figure 2). The working section cross-section was a square with 0.305m height in y and z directions (Figure 2) [8].

For the subsonic portion, an adaptation of the cubic profile described by Hussain [10] was selected to avoid excessive pressure losses compared to other shapes. The supersonic portion of the nozzle is calculated through characteristic line reflections for a target supersonic length, L , applying a tangency condition at the throat located at $x = 0$ in Figure 2. This tangency condition is met by aligning the last control polygon segment of the subsonic nozzle with the first control polygon segment of the supersonic nozzle, which ensures continuous derivative at the throat. The supersonic length value is calculated as the closest integer length to the length of the shortest Mach 3.0 nozzle designed via the MoC, as that is the maximum Mach number the wind tunnel is designed for and therefore it would set an upper limit for the supersonic nozzle length. The subsonic nozzle length L_{ss} is selected to be $L/2$, as this value was found to be sufficient to deliver good quality flows in terms of turbulence intensity based on the nozzle contraction ratio and

centerline flow velocity [10]. However, a shorter length value is recommended for future work after completing the analysis of the flow physics of the nozzle.

B. Parametric Nozzle Geometry Definition

The nozzle contours along both the subsonic and supersonic sections were parameterized as two cubic Bezier curves b_i with tangency conditions at their intersection at the throat as shown in Figure 2. Given a set of control points a_j forming a control polygon, the curves b_i were defined as the linear interpolation between each polygonal vertex and an internal curve parameter t as shown in Equation 2 [11]:

$$b_i = \sum_{j=0}^{n=3} a_j \binom{n}{j} t^j (1-t)^{n-j}, \quad 0 \leq t \leq 1 \quad (2)$$

The subsonic portion of the nozzle includes the control points a_0 to a_3 , while the supersonic portion includes the control points a_3 to a_6 . The axial position of control points a_4 and a_5 was parameterized using the dimensionless quantities d_1 and d_2 , which will drive the numerical nozzle design. For the baseline case, via non-linear-least-squares Bezier curve fitting with an initially-uniform knot vector t to a minimum length MoC solution with tangency conditions at the throat, the parameters d_1 and d_2 take the values of 0.05 and 0.8, respectively.

If the velocity magnitude u is differentiated with regards to its normal distance from the nozzle wall, the curvature of the nozzle profile κ is defined as per Equation 3 using the first and second derivatives of the velocity magnitude u' and u'' respectively. The inflection point I (Figure 2), is defined as the point of the nozzle contour at which the curvature in Equation 3 becomes zero, which happens when the cross product in the numerator of Equation 2 becomes null. The point of maximum absolute derivative $|u'|$ or maximum contour angularity J is also shown in Figure 2 for reference:

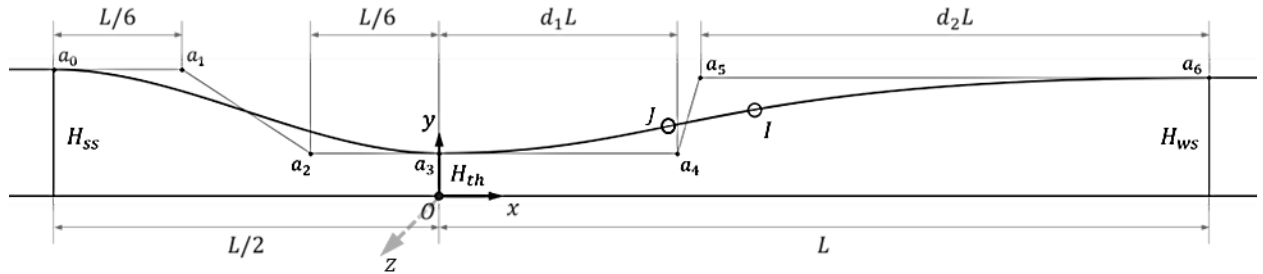


Figure 2: Parametric Nozzle Geometry at symmetry plane.

$$\kappa = \frac{|u'' \times u'|}{\|u'\|^3} \quad (3)$$

Applying mass flow conservation from the value calculated in Equation 1 and under the assumption of constant in-plane depth of the nozzle equal to the working section width H_{ws} , the nozzle inlet, throat and outlet cross-sectional areas were also derived from Equation 1. The semi-heights H_{ss} , H_{th} and H_{ws} of those sections can be therefore calculated, with values 0.562, 0.19 and 0.5 feet respectively. The control polygon specification based on the control points a_i in Figure 2 is detailed in Table 2:

Table 2: Definition of nozzle control points location.

Control Point	X-Coordinate	Y-Coordinate
a_0	$-L/2$	H_{ss}
a_1	$-L/3$	H_{ss}
a_2	$-L/6$	H_{th}
a_3	0	H_{th}
a_4	$d_1 L$	H_{th}
a_5	$L(1 - d_2)$	H_{ws}
a_6	L	H_{ws}

The design ranges for d_1 and d_2 were initially estimated via non-linear-least-squares curve fitting of existing wind tunnel nozzles [12] [13] that operate within the target operating Mach number of the wind tunnel. The supersonic portion of these nozzles was fit with a cubic Bezier curve where the initial and final control points that corresponded to the throat and working section inlet were fixed, and the y -derivative at those points was set to zero to achieve tangency between subsonic and supersonic nozzles, and between the supersonic nozzle and the working section. An initially-uniform knot vector t was used to calculate the best-fitting control polygon to the reference nozzles for simplicity using the formulation in Equation 2, and both the knot and control point values were re-adjusted until the squared distances of the nozzle contour lines between the fitted and reference models [12] [13] converged within 10^{-5} which indicates a sufficiently accurate representation of these previous designs.

C. Flow Quality Metrics

The supersonic nozzle will become part of a test rig whose flow quality at the test section is of critical importance. Therefore, flow uniformity and the ability to accurately deliver the target Mach number at the working section inlet were used as part of the nozzle's design and optimization process. Flow uniformity is influenced by the development of Mach wave reflections across the supersonic section, while the target Mach number is strongly linked with the growth of the boundary layers along the nozzle's walls whose presence decelerates the flow at the centerline causing discrepancies between the target and delivered free stream Mach number at the working section.

Flow uniformity at the bulk of the flow outside the boundary layer regions at the nozzle exit plane is defined as in Equation 4. It is expressed as a percentage in the subsequent sections of the paper.

$$\psi(x) = \left(\frac{u_{max} - u_{min}}{u_{avg}} \right)_{\delta > 1.1\delta_\kappa} \quad (4)$$

Where u_{max} , u_{min} and u_{avg} stand for the maximum, minimum and mass-flow-averaged (per unit length in the out-of-plane direction z in Figure 2) values of the velocity magnitude in the bulk of the flow defined as the stream tube above 10% of the maximum curvature boundary layer thickness δ_κ at each axial position as shown in the boundary layer markers in Figure 3.a. This boundary layer thickness is defined as the point of maximum curvature of the flow field as shown in Figure 3.b, which is calculated as the inverse of the radius of curvature to the velocity magnitude distribution u in the normal direction to the wall. Considering u is a discrete function of points, it is possible to group these points in consecutive overlapping triplets of points a , b and c as represented in Figure 4. Then, it is possible to approximate the curvature of the flow profile to the inverse of the radius of the circumscribing circle or circumradius with reference to each triplet of points [14] as shown in Equation 5:

$$R = \frac{1}{2} \sqrt{(\overline{bc} \times (\overline{ac} \times \overline{bc}) \cdot \|\overline{ac}\|^2 - \overline{ac} \times (\overline{ac} \times \overline{bc}) \cdot \|\overline{bc}\|^2) / \|\overline{ac} \times \overline{bc}\|^2} \quad (5)$$

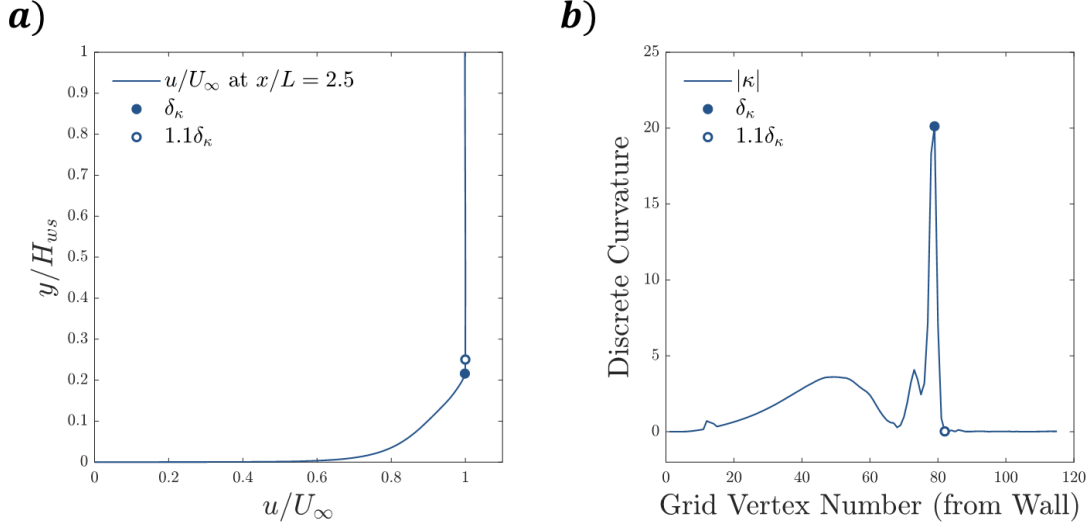


Figure 3: Boundary layer profile (a) and discrete curvature distribution, κ , of the profile (b).

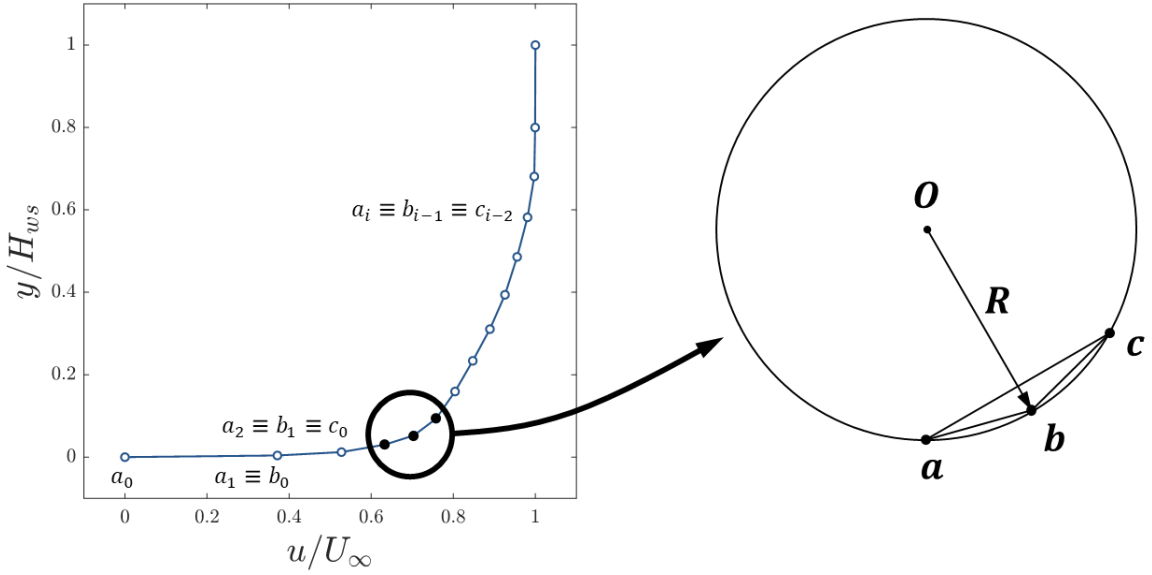


Figure 4: Circumradius to a triplet of points applied on a boundary layer profile.

Axial flow uniformity, ψ , varies with the axial position in the supersonic part of the nozzle. As part of the design space exploration process, each nozzle is characterized via its statistically maximum value defined as the average flow distortion along axial length L (Figure 2) plus the statistical deviation from the mean value σ shown in Equation 6:

$$\hat{\psi}_{max} = \text{mean}(\psi(x)) + \sigma(\psi(x)) \quad (6)$$

D. CFD Methods

The optimization of the MoC nozzle design was conducted via Euler CFD simulations using the ANSYS CFX 2021R1 solver [15]. The viscous simulations to determine throat area corrections were conducted using the compressible, pressure based, steady state Navier-Stokes solver with a second order spatial discretization scheme also part of ANSYS CFX. The air was modelled as an ideal gas. For the viscous simulations, a k-omega shear stress transport (SST) model was used to model the turbulence production, with automatic wall functions in the vicinity of the walls when y^+ is above 11, as shown in Table 3. The fluid flow through the nozzle is achieved through a pair of (static) pressure-driven boundary conditions placed upstream of the subsonic nozzle inlet and downstream the working section outlet as shown in Table 4. The bounding surface of the assembly is simulated as an adiabatic no-slip wall, and the centerline of the domain as a symmetry plane. These boundary conditions seek to replicate the conditions of the experimental campaign from Willis [8] applied to planar nozzle domain corresponding to the symmetry plane at the center of the working section depth in the z-coordinate.

Table 3: Grid characteristics applied for the CFD simulations of the viscous nozzles.

Number of elements	98,704
Maximum included angle	108°
Maximum aspect ratio	80,000
Minimum wall y^+	1
Maximum equiangular skewness	0.2

Table 4: Boundary conditions used for the nozzle CFD simulations.

Geometry	Boundary Condition	Specifications
Subsonic inlet	Inlet	$P_{ss} = 1,676,58.43 \text{ Pa}$ $T_{ss} = 289.68 \text{ K}$ $M_{ss} = 0.2$
Supersonic outlet	Outlet	$P_{ws} = 9,318.6 \text{ Pa}$
Nozzle walls	Wall	Smooth, Adiabatic, No Slip
Nozzle centerline	Symmetry	-

The spatial discretization was coupled with the assessment of the working section boundary layers, where the inflation layers of the grid cells in the vicinity of the nozzle walls were set to capture the locus of the maximum curvature boundary layer thickness at every axial station of the computational domain. A cell growth function proportional to the circumradius of each triplet of grid points is set as per Equation 5, which is constrained with a minimum wall y^+ value. This integrated approach ensures the best viscous sublayer processing with the minimum grid cell count (shown in Table 3) resulting in faster computing times. The structured planar grid is generated from the initial MoC solution flow topology following a node spacing in the normal direction from the wall conformal to the inverse boundary layer curvatures at the end of the extended working section domain used for stability of the numerical solution, which corresponds to $x/L = 2.5$.

All numerical simulations were performed on an AMD Ryzen 32-core (3.49 GHz) workstation, with the same optimized grid file for ease of solution interpolation between different cases.

E. Design Space exploration methods

The parametric definition of the nozzle enabled the exploration of a design space to determine nozzle geometries that satisfied the flow quality requirements. The design space of d_1 and d_2 was initially explored across the ranges shown in Table 5 determined empirically. These ranges were narrowed down in subsequent design iterations until a sub-region of the design space with the most promising designs based on their maximum statistical distortion value was identified.

Table 5: Initial design envelope for supersonic nozzle parameters.

Parameter	Minimum	Maximum
d_1	0.05	0.45
d_2	0.50	0.80

The design space comprises permutations of d_1 and d_2 , and it is sampled with the minimum number of design points to minimize computing time, while maximizing the coverage area. The selected sampling approach is a smart flow-distortion-driven generation of samples that seeks to maximize the minimum pointwise distance between nozzle candidates, generating a variable density of design points based on the distortion levels of the previously-generated designs. This uniform sample generation is known as the MAXIMIN design, previously shown in [16].

The strategy for nozzle design and optimization includes an initial sweep of nozzle designs using Euler CFD to identify the most-probable design region in terms of d_1 and d_2 , followed by design space sweeps using viscous CFD seeking to correct the throat to exit area to match the target working section Mach number of 2.5. A schematic of the identified solution regions found by each viscous design space iteration after the initial inviscid parameter sweep and the location of the final nozzle design is shown in Figure 5.

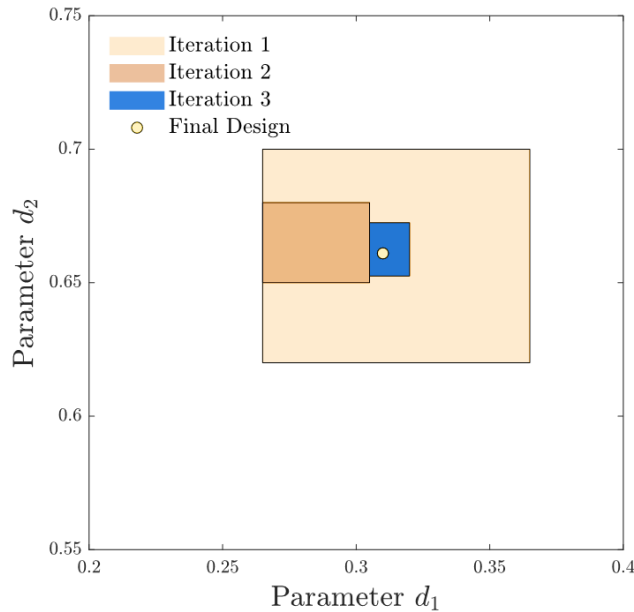


Figure 5: Design space regions of viscous iterations for throat correction.

IV. Results and Discussion

A. Mach Wave Mitigation

The first step in the nozzle design process was to reduce the design space to the regions that resulted in minimal Mach wave reflections across the supersonic part of the nozzle. Cancelling, or partially suppressing, these Mach wave reflections is important to yield acceptable flow quality characteristics at the working section downstream the nozzle. In this work, flow quality was quantified via the function ψ that should be minimized (Equations 5 and 6). The location of the Mach wave reflections can be predicted using inviscid flow models as described by Yen [17], and hence a parameter sweep based on Euler CFD models is used to reduce the nozzle design regions from the initial set where d_1 ranges between (0.05, 0.45) and d_2 ranges between (0.5, 0.8), to the 'Iteration 1' zone in Figure 5 where d_1 ranges between (0.265, 0.365) and d_2 between (0.62, 0.7).

In Figure 6.a, the variation of the flow distortion metric ψ across the working section is shown for the nozzle designs tested. The statistical maximum value of each profile is represented as a contour map in the d_1 and d_2 design space in Figure 6.b to highlight the promising design region for the next viscous DSE iteration in Figure 5. The design points 'A', 'B', 'C' and 'D' are shown for reference combined with the other nozzle candidates in Figure 6.a and Figure 6.b, as well as their flow distortion characteristics in terms of density gradients in Figure 7, which highlights the reduction of maximum statistical distortion values that reflect in the uniformity levels of the flow field.

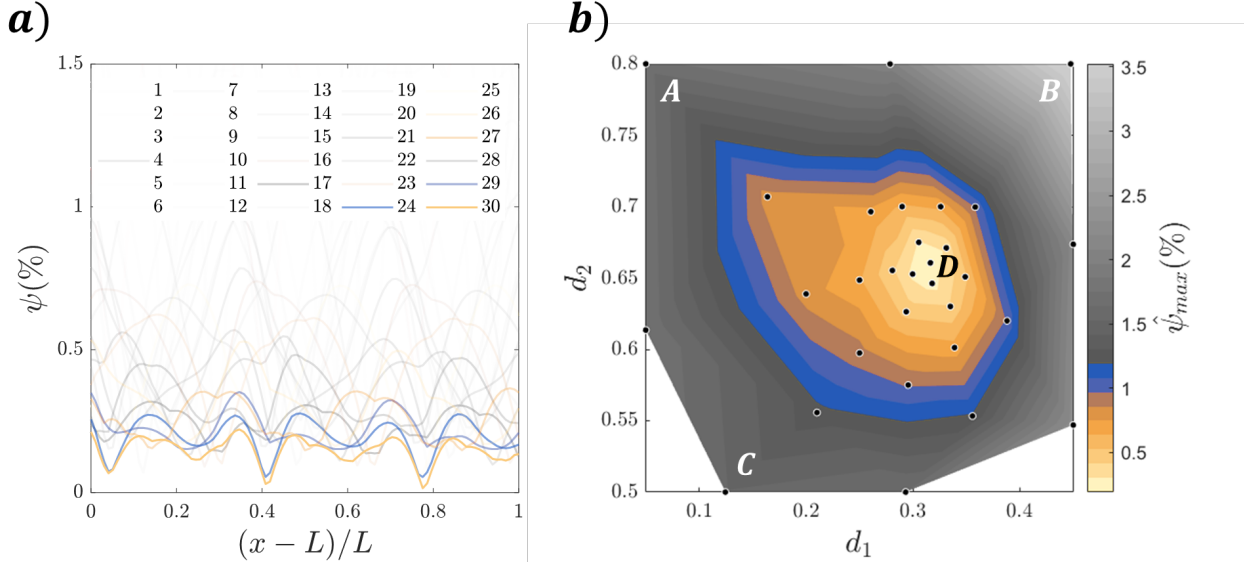


Figure 6: Design space exploration via Euler (inviscid) CFD for wave reflection mitigation. (a) Flow uniformity variation across the supersonic section of the nozzle. (b) Design space for best flow uniformity based on the parameters d_1 and d_2 .

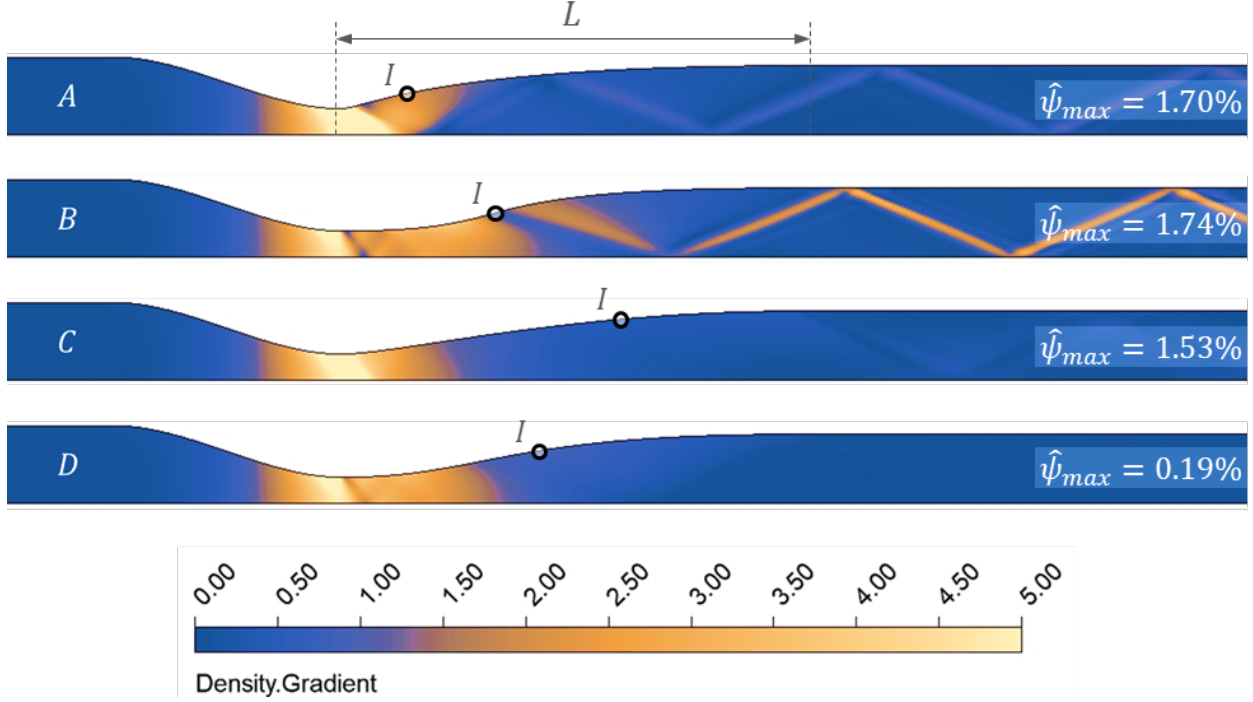


Figure 7: Expansion wave reflections for nozzle designs ‘A’, ‘B’, ‘C’ and ‘D’ and their maximum statistical distortion value $\hat{\psi}_{max}$. Mitigation of Mach wave reflections is achieved in design ‘D’.

B. Throat Corrections via Viscous CFD Simulations

When the viscosity of the air is considered, the flow properties delivered by the nozzle will vary from the values predicted via the Euler based simulations. This is because the developing boundary layer along the nozzle walls causes a Mach number reduction at the working section proportional to its displacement thickness [1]. Slightly increasing the target Mach number of the supersonic nozzle is an indirect approach that can address this issue, as previously discussed by Vallabh [2]. The implication of this change is a nozzle design with a smaller throat cross section than the baseline model calculated through critical area ratios and mass-flow conservation. This means that a new pair of optimal d_1 and d_2 values in the vicinity of the previously calculated d_1 and d_2 must be determined so that the maximum statistical flow distortion value is minimized in the working section.

The throat correction process was conducted iteratively, where the required centerline Mach number at the working section inlet is estimated based on how far the previous nozzle design is from the target operating conditions when the flow has viscous effects as per the following sequence of steps. For simplicity, the behavior of the nozzle assembly is assumed to be uniform in its spanwise direction and equal to its symmetry plane (two-dimensional or planar domain). However, this is not fully representative as there are corner effects, boundary layer growth in the side walls and spanwise turbulence production that are not captured in the two-dimensional simulations. Therefore, the mass-flow averaged Mach number per unit length in the z -direction in the working section inlet is selected as the planar nozzle design Mach number since it is lower than the freestream value because of the boundary layer presence. This seeks to alleviate the effect of the three-dimensional effects that may bring the freestream Mach number down during its three-dimensional performance. The following sequence of steps summarizes the iterative throat correction process:

1. A target Mach number was iteratively estimated for the supersonic nozzle based on the working section mass-flow averaged Mach number (Equation 7) of a previous generation of nozzles.
2. A new generation of nozzles is created in terms of d_1 and d_2 using the previous isentropic Mach number
3. All designs are simulated using viscous CFD and evaluated in terms of maximum statistical flow distortion
4. The mass-flow averaged Mach number at the nozzle exit section is calculated as per Equation 7 including boundary layers, and this value is used to iteratively correct the target Mach number that is being used in step 1 of the next throat correction stage.
5. A new optimal design region is identified for the next iteration based on maximum distortion values.

$$\langle M \rangle_{ws} = \frac{\sum M_i \cdot \rho_i \cdot |\vec{A}_i \cdot \vec{v}_i|}{\sum \rho_i \cdot |\vec{A}_i \cdot \vec{v}_i|} \quad (7)$$

Where the terms $\vec{A}$ and $\vec{v}$ refer to the area and velocity vectors per unit length in the stream-wise direction of the symmetry plane of the nozzle shown in Figure 1, and the index i refers to the face or cell number in the target cross-flow plane in which the integral calculation takes place. In the case of interest, this quantity is calculated at the working section inlet or supersonic nozzle outlet, which matches the axial position of a_6 in Figure 2. This value will be lower than the centerline Mach number aiming to minimize the overprediction of the nozzle center line Mach number during the subsequent three-dimensional simulations, as shown in a following section.

In the first DSE iteration, the nozzle design ‘D’ that resulted from the inviscid parameter sweep is evaluated using viscous simulations, resulting in a working section Mach number slightly below 2.45 using Equation 7. Hence, the baseline nozzle model in the step 1 of the previous list was re-adjusted to deliver Mach 2.6 in the working section. A new range for the parameters d_1 and d_2 is estimated based on the optimal distortion region of the map in Figure 6.b, where 19 equally spaced samples are generated in a region with center $(d_1, d_2) = (0.315, 0.660)$ and radius $(r_1, r_2) = (0.05, 0.04)$.

The distortion coefficient of the newly generated nozzles is evaluated in the working section as shown in Figure 8.a. The maximum statistical values of each design are shown as a contour plot in the (d_1, d_2) design space in Figure 8.b, highlighting the most promising design region for the next generation of nozzle designs. The area averaged Mach number delivered by the supersonic nozzle in this iteration was equal to 2.55 using Equation 7.

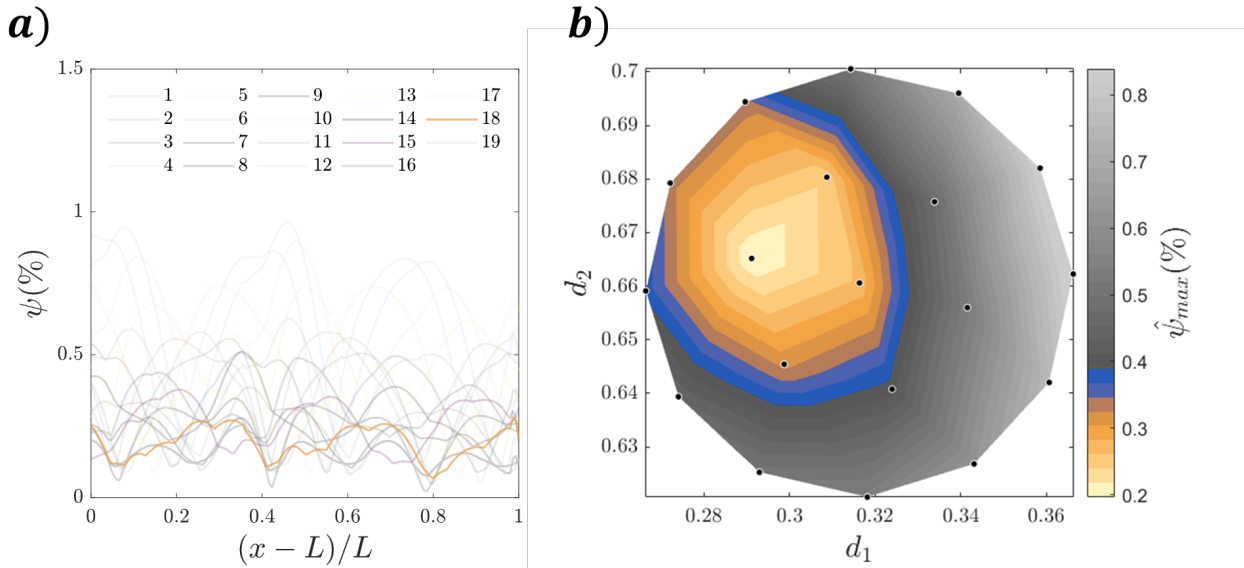


Figure 8: Design space exploration via viscous CFD for throat corrections at target Mach 2.5. (a) Flow uniformity variation across the supersonic section of the nozzle. (b) design space for best flow uniformity defined via d_1 and d_2 parameters.

For the second design space iteration, the isentropic baseline nozzle design was adjusted to deliver Mach 2.55 in the working section. Twelve equally spaced samples are generated in a polar region with center $(d_1, d_2) = (0.285, 0.665)$ and radius $(r_1, r_2) = (0.02, 0.015)$. The distortion parameter of these nozzles is shown in Figure 9.a, where a lower degree of scatteredness and magnitude in the ψ profile was found compared to Figure 8.a and Figure 6.a. The achieved Mach number at the working section inlet was slightly above 2.5 in all these nozzles.

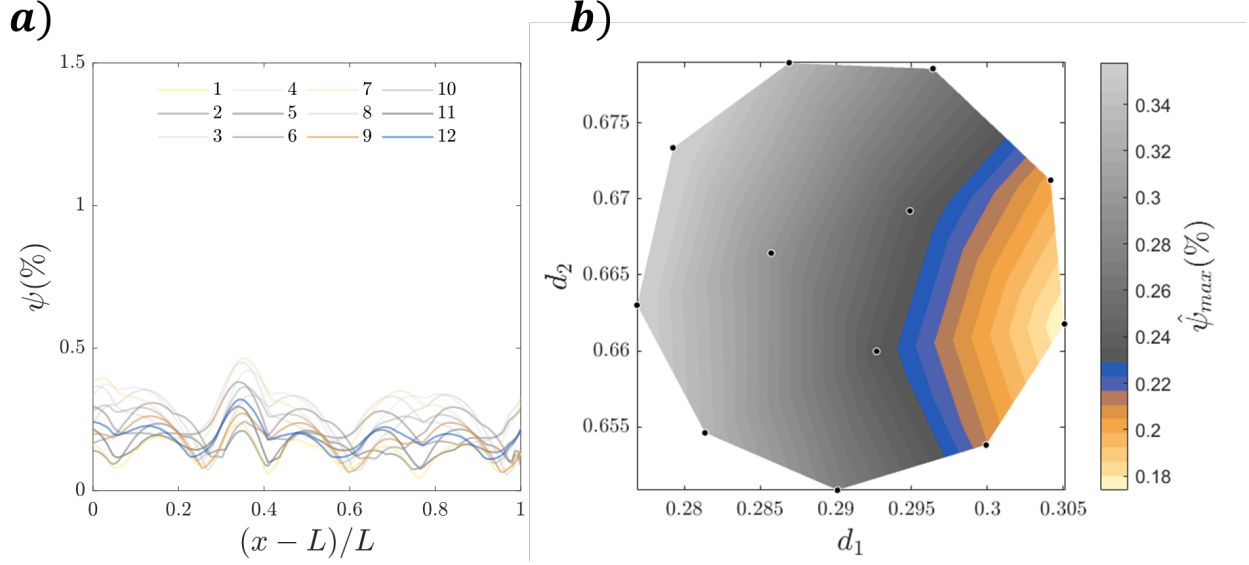


Figure 9: Design space exploration via viscous CFD for throat corrections at target Mach 2.55. (a) Flow uniformity variation across the supersonic section of the nozzle. (b) design space for best flow uniformity defined via d_1 and d_2 parameters.

The contour map of the maximum distortion values in the design space shown in Figure 9.b shows that the previously estimated ranges for d_1 and d_2 do not contain the region of optimal flow uniformity nozzles. A final design space viscous iteration with throat corrections is done for baseline nozzle models designed for an isentropic Mach number of 2.5525, which was found to match the mass-flow averaged Mach 2.5 value in the working section inlet. A total of 13 nozzle samples are generated via the previously described smart design exploration method in the d_1, d_2 range of center $(0.3125, 0.6625)$ and radius $(r_1, r_2) = (0.0075, 0.01)$, all of which indicate distortion lower than 0.3% (Figure 10.a). The contour view of the design space and the maximum distortion values in Figure 10.b shows an optimal nozzle design ‘E’ with a final d_1 and d_2 values of 0.31 and 0.66, respectively.

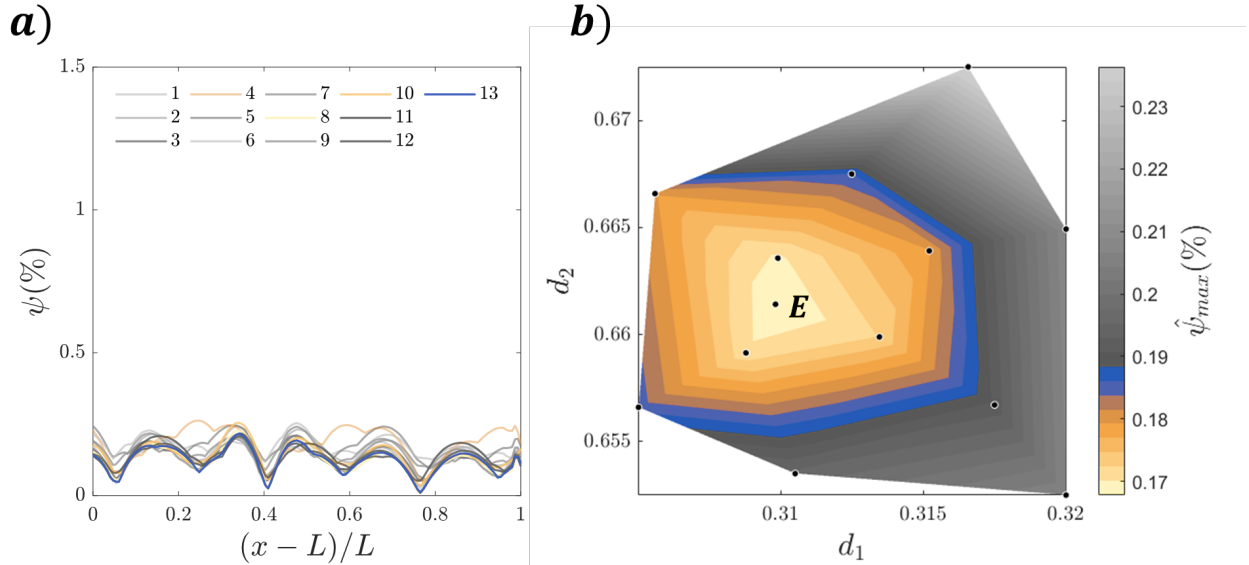


Figure 10: Design space exploration via viscous CFD for throat corrections at target Mach 2.5525. (a) Flow uniformity variation across the supersonic section of the nozzle. (b) design space for best flow uniformity defined via d_1 and d_2 parameters.

The contours of density gradients and Mach number are shown in Figure 12 and Figure 12 respectively. The nozzle design shows a maximum statistical distortion level of $\hat{\psi}_{max} \cong 0.16\%$ in the working section, which achieves maximum levels of flow uniformity for the selected configuration of supersonic nozzle contour in Equation 2.

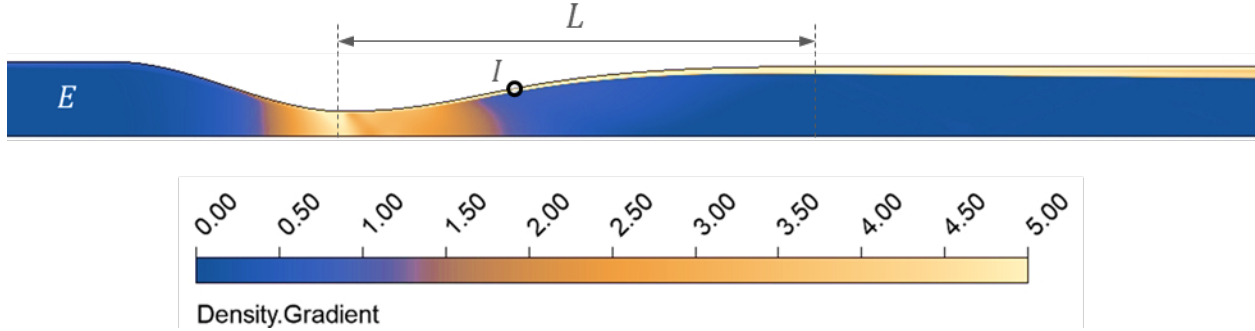


Figure 11: Density gradient distribution in the symmetry plane for design ‘E’

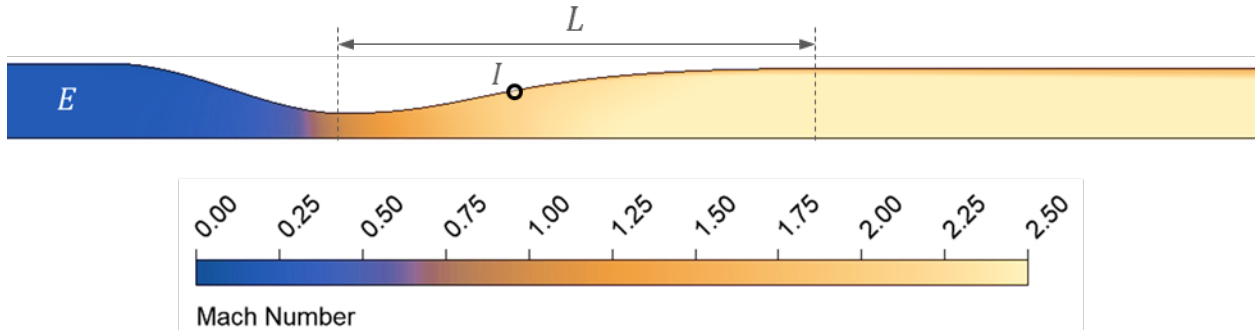
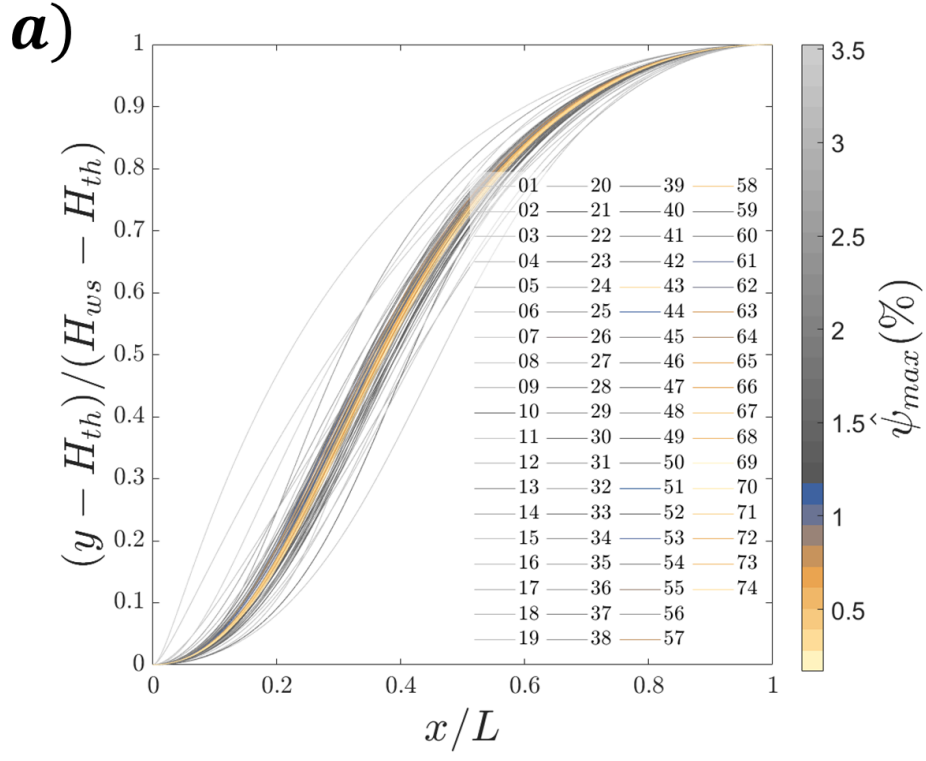
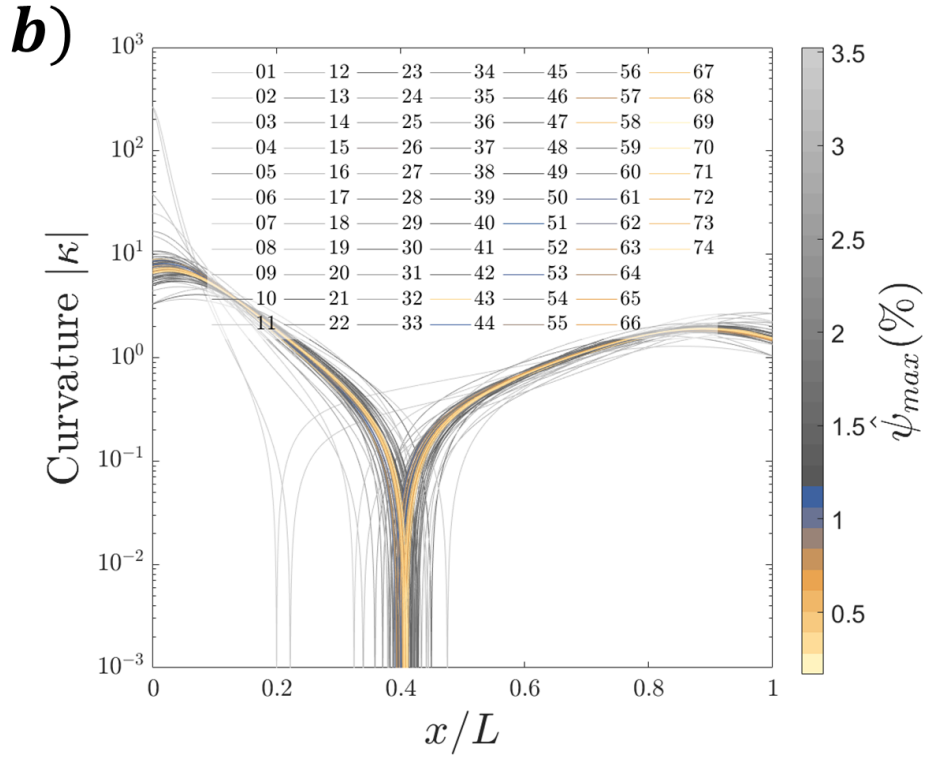


Figure 12: Mach number distribution in the symmetry plane for design ‘E’.

A graphical summary of the geometrical specifications for the analyzed supersonic nozzles is provided in Figure 13. All 74 design points of the supersonic nozzles are shown as legend entries, where the nozzle designs ‘A’, ‘B’, ‘C’, ‘D’ and ‘E’ correspond to the design points ‘01’, ‘03’, ‘06’, ‘30’ and ‘74’ respectively. The different samples carry a color code associated with their maximum statistical distortion value, highlighting the designs with $\hat{\psi}_{max} \leq 0.3\%$.

Figure 13.a shows the non-dimensional contours for all the supersonic nozzles, normalizing the x-dimension with the supersonic nozzle length L and the y-dimension with the working section and throat heights H_{ws} and H_{th} . As all curves are cubic Bezier interpolations, it is possible to obtain the analytical expression of their curvatures κ as in Equation 3. Figure 13.b captures, in logarithmic scale, the absolute curvature values using the same nomenclature as in Figure 13.a. The inflection point of nozzle designs with $\hat{\psi}_{max} < 0.5\%$ remains on a locus of the 44% of the nozzle length downstream the throat. The clustering of design points in this region is justified by the aim of the design space sampling method of seeking distortion-free nozzles, which indirectly produces more nozzle samples in the design space regions with the greatest flow uniformity potential. In Figure 13.c the previous nozzle designs have their maximum profile angularity point J on a locus of the 37% of the supersonic nozzle length.



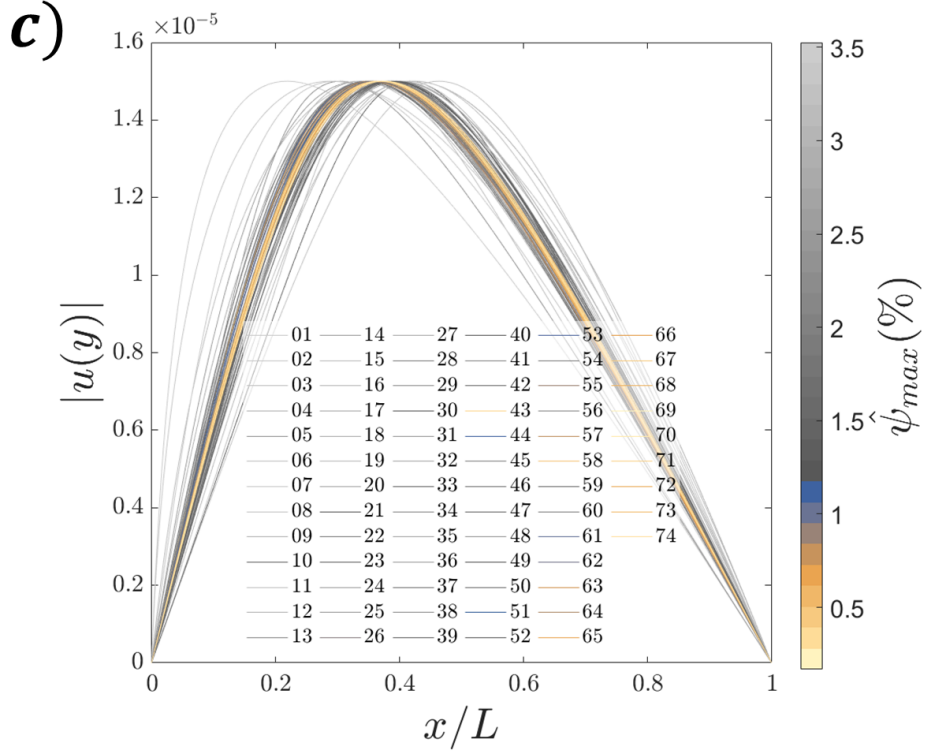


Figure 13: Geometrical specification of the analyzed nozzles. The color scale indicates the distortion values. (a) Normalized supersonic nozzle contouring. (b) Absolute curvature distributions of the previous supersonic nozzles as a function of the axial position downstream the throat. (c) Absolute derivative value of the supersonic nozzle contouring to highlight the location of the maximum profile angularity J .

Both inflection point and intermediate control point locations can be used as initialization for nozzle contouring optimization using other nozzle geometries or operating conditions. Further refinement of a minimum distortion nozzle contouring using the mathematical formulation in Equation 2 would be possible via degree elevation or knot insertion [11] at the cost of extra degrees of freedom to the suggested design space by increasing the curve order.

C. 3D Nozzle Design Verification

The three-dimensional behaviour of the two-dimensional nozzle design differs from the planar design because of the development of a spanwise layer in the side walls and corner flows [13], which are characterized by strong pressure. At the corners of the nozzle, the different viscous sublayers form a capture stream tube of variable thickness that changes the overall flow properties in the bulk flow. It is of interest to quantify the deviation of the flow characteristics from the nozzle design parameters.

The previously generated nozzle design ‘E’ was simulated in 3d CFD by repeating the grid resolution in the out-of-plane z direction of the upper half of the nozzle ‘E’. As a result, a symmetric quadrant of the nozzle with two symmetry planes is modelled with approximately 22 million hexahedral cells to capture the boundary layers normal to each symmetry plane. The same boundary conditions in Table 4 also apply to the 3d model.

The results of the simulation confirm the hypothesis freestream Mach number being lower than calculated in 2d because of the side wall boundary layer and the corner flows of the nozzle. In nozzle ‘E’, the working section mass-flow averaged Mach number in the 2d CFD was 2.50124 and the freestream Mach number was 2.53519. In the 3d CFD, those values were 2.4604 and 2.51646 respectively. The three-dimensional viscous effects in the vicinity of the walls reduce the capture stream tube of the bulk of the flow proportionally to the boundary layer displacement thickness in each viscous sublayer region compared to an idealized isentropic solution. Therefore, the flow is decelerated in the same order of magnitude [7].

The Mach number distribution across a range of axial stations post working section is shown in Figure 14. Moreover, the simulated boundary layers at the side walls have a different cross-sectional distribution to the upper and lower surfaces of the working section as shown in Figure 14. This is due to the presence of vertical velocities within the side wall boundary layers, a typical secondary flow that arises from the corners of the two-dimensional nozzle [13], which are captured in Figure 15.

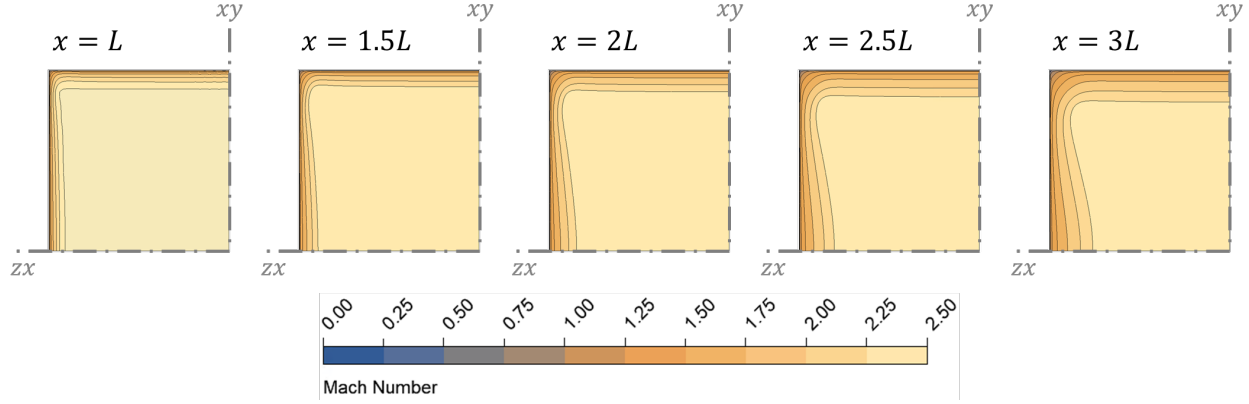


Figure 14: Mach number contours in the three-dimensional ‘E’ nozzle to highlight boundary layer thickness growth

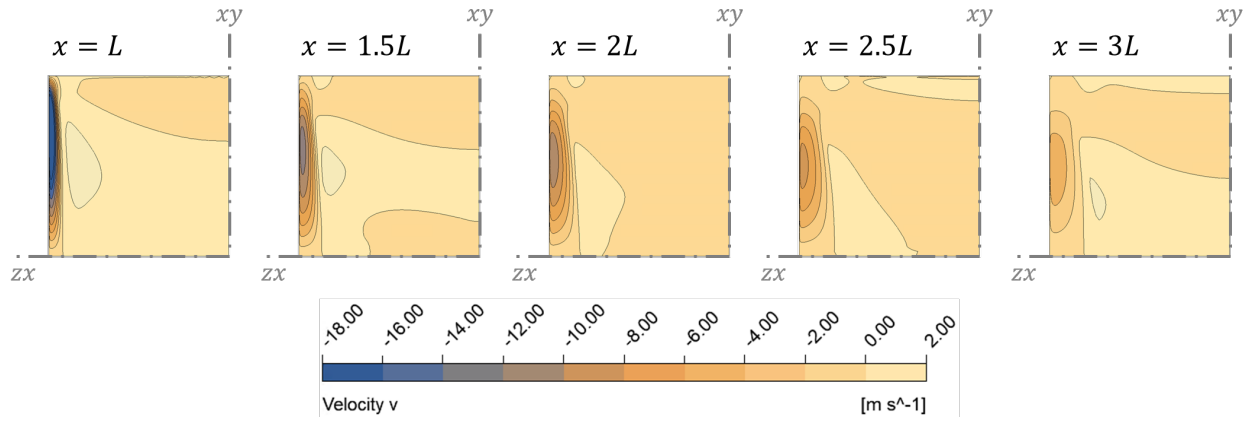


Figure 15: Vertical v -velocity contours in the three-dimensional ‘E’ nozzle to highlight secondary flows

Finally, it was desirable to benchmark the performance of the designed nozzle with the experimental setup of Willis [12] in terms of boundary layer matching downstream of the working section and Mach number comparison. The available data is a set of boundary layer flow measurements at an unknown axial station downstream the supersonic nozzle outlet and the freestream Mach number at that location. Via least-squares comparison, the reference plane for experimental measurements is estimated to be located at $x = 1.344L$ as shown in Figure 16, where the boundary layer displacement thickness was reported to be 2.64 centimeters. The reported freestream Mach number in the experimental campaign and the calculated CFD value are both identical and equal to 2.46 at that axial position. This indicates that the designed nozzle is suitable for a validation study based on the reference setup [12] and will therefore be used for further studying the aerodynamic behaviour of porous surfaces at the flow conditions delivered by the nozzle.

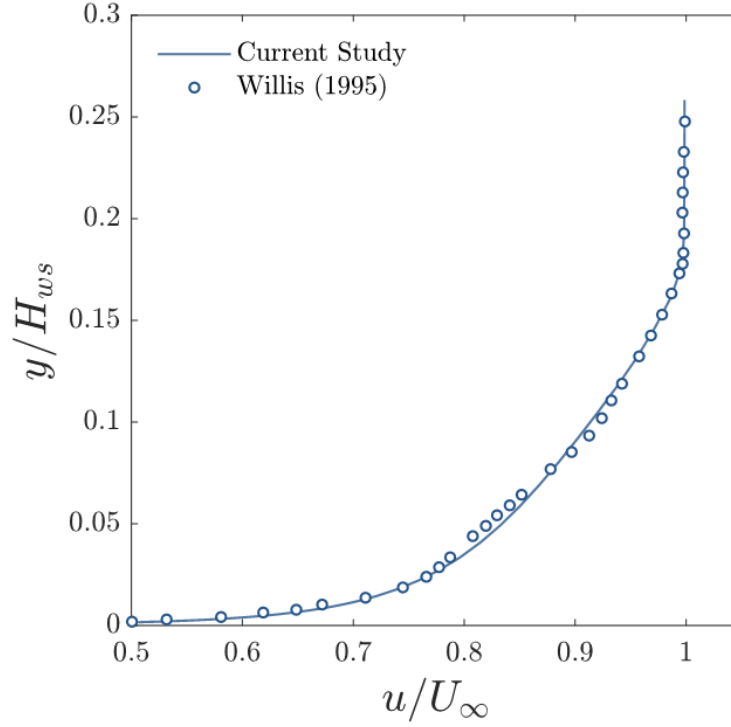


Figure 16: Boundary layer profile comparison between Willis [12] and the CFD results at the reference plane $x/L = 1.344$ downstream of the supersonic nozzle outlet.

V. Conclusions

A methodology for distortion-free supersonic nozzle design and optimization is herein presented, using computational fluid dynamics. The method relies on the Method of Characteristics to produce a baseline nozzle contour that delivers the target Mach number at the exit of 2.5. This baseline design is then fine-tuned to mitigate Mach wave reflections that develop across the supersonic section and penalized flow quality at the exit. The fine-tuning relies on a flexible, parametric definition of the nozzle contour which is used as part of a design space exploration process with Euler-based CFD in the loop. The outcomes of the work suggest that it is possible to almost eliminate the Mach wave reflections generated in the vicinity of the inflection point downstream the nozzle throat via Euler-based computational models. Subsequently, viscous CFD are conducted to determine the nozzle throat area that delivers the desired exit Mach number when viscous boundary layers are present across the domain. Therefore, the optimized nozzle geometries have reduced throat heights compared to the theoretical isentropic values to account for the presence of the viscous layers. Finally, the 3d CFD of the final contour suggests that the influence of the corner flows across the square cross-section of the nozzle is only detrimental to the exit Mach number at axial positions $x > 2L$ hence no further throat corrections were made to account for this.

It is recommended that the target supersonic nozzle is designed for an isentropic mass-flow averaged Mach number that is 2 to 3% greater than the design target Mach number at the working section inlet, so that the viscous three-dimensional design Matches the target design values. If a two-dimensional nozzle is studied or in the case whereby two-dimensional viscous simulations can be afforded during parameter sweeps, it may be more computationally-efficient to firstly evaluate the right viscous throat height before launching the throat correction iterations. It is finally recommended that the subsonic portion of the nozzle is made shorter than in the present case for greater flow uniformity. For the selected nozzle inlet Mach number, a nozzle length L_{ss} less than 40% the supersonic portion length L is suggested.

Acknowledgments

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

References

- [1] McCabe, A. "Design of a Supersonic Nozzle," University of Manchester, R. & M. No. 3440, March 1964.
- [2] Vallabh, B., "Investigation of Nozzle Contours in the CSIR Supersonic Wind Tunnel," University of Witwatersrand, MSc thesis, Johannesburg, 2016.
- [3] Anderson, J. A., "Numerical Techniques for Steady Supersonic Flow," *Modern Compressible Flow*, 3rd ed., McGraw-Hill, Maryland, 2003, pp. 383-392.
- [4] Foelsch, K., "The Analytical Design of an Axially Symmetric Laval Nozzle for a Parallel and Uniform Jet," *Journal of the Aeronautical Sciences*, Vol. 16, No. 3, 1949, pp. 161-188.
- [5] Flock, A. K. and Gülhan, A. "Design of Converging-Diverging Nozzles with Constant-Radius Centrebody," *CEAS Space Journal*, Vol. 12, 2020, pp. 191-201.
- [6] Sivells, J. C., "Design of Two-Dimensional Continuous-Curvature Supersonic Nozzles," *Journal of the Aeronautical Sciences*, 1955, pp. 685-732.
- [7] Kogan A., "Boundary-Layer Correction in Supersonic Nozzle Scaling," *Journal of the Aeronautical Sciences*, p. 64, 1958.
- [8] Willis, B. P., Davis, D. O. and Hingst, W. R., "Flow Coefficient Behaviour for Boundary Layer Bleed Holes and Slots," 33rd Aerospace Sciences Meeting and Exhibit, January 1995.
- [9] Pope, A. G and Goin, K. L., "Design of Intermittent Blowdown Tunnels", *High-Speed Wind Tunnel Testing*, John Wiley and Sons, Inc., New York, 1965, pp. 66-134..
- [10] Hussain, A. K. M. F., and Ramjee, V. "Effects of the Axisymmetric Contraction Shape on Incompressible Turbulent Flow," *Journal of Fluids Engineering*, Vol. 98, No. 1, 1976, pp. 58–68. <https://doi.org/10.1115/1.3448210>.
- [11] Prautzsch, H., Boehm, W. and Paluszny, M. "Bezier and B-Spline Techniques" *Mathematics and Visualisation*, Springer, 2002.
- [12] Willis, B. P., Davis, D. O. and Hingst, W. R., "Flowfield Measurements in a Normal-Hole-Bled Oblique Shock-Wave and Turbulent Boundary-Layer Interaction," AIAA/SAE/ASME/ASEE 31st Joint Propulsion Conference and Exhibit, July 1995.
- [13] Sabnis, K. et al. "Flow Characterisation for a Validation Study in High-Speed Aerodynamics," AIAA Aviation 2019 Forum, Texas, 2019.
- [14] Weisstein, E. W., "Circumradius," [Online]. Available: <https://mathworld.wolfram.com/Circumradius.html>.
- [15] ANSYS, "CFX," [Online]. Available: <https://www.ansys.com/en-gb/products/fluids/ansys-cfx>.
- [16] Mak, S. and Roshan, J. V., "Minimax and Minimax Projection Designs Using Clustering," *Journal of Computational and Graphical Statistics*, Atlanta, 2016, pp. 166-178.
- [17] Yen, J. C. and Martindale, W. R., "An Inviscid Supersonic Nozzle Design Approach to Perfect Flow Uniformity for Wind Tunnel Applications," 26th AIAA Applied Aerodynamics Conference, Hawaii, 2008.



DSPACE

<https://dspace.org/>

Design and optimisation of a Mach 2.5 wind tunnel nozzle

Moreno, Miguel; Migliorini, Matteo; Zachos, Pavlos K.; Haslam, Anthony; MacManus, David G.

2023-01-19

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2023-1468>

Moreno M, Migliorini M, Zachos PK, et al., (2023) Design and optimisation of a Mach 2.5 wind tunnel nozzle. In: AIAA SciTech Forum 2023, 23-27 January 2023, National Harbor, Maryland, USA. Paper number AIAA 2023-1468