

Numerical simulation of flows in the ONERA S3Ch transonic wind tunnel

J.-L. Hantrais-Gervois* and N. Rembaut.†

DAAA, ONERA, Institut Polytechnique de Paris, 92190, Meudon, France

G. Tanguy‡

Université de Lille, CNRS, ONERA, Arts et Métiers ParisTech, Centrale Lille, Laboratoire de Mécanique des Fluides de Lille - Kampé de Fériet, 59000, Lille, France

M. Werner§

German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, 37073 Göttingen, Germany

D. Di Pasquale|| and P. Thurier.

Cranfield University, College Rd, Wharley End, Bedford MK43 0AL, The United Kingdom

A. Katirci, B. Nasuhbeyoğlu||

Roketsan Missile Industries, P.K. 30 Elmadağ 06780 Ankara, Turkey

A. Elmiligi**

NASA Langley Research Center, 100 NASA Road Hampton, VA 23681-2199, The United States of America

This paper investigates the flow field within the high-speed leg of the ONERA S3Ch wind tunnel as part of the NATO activity AVT-387, “Common Research Wind and Water Tunnel for CFD Verification and Validation.” The objective of the study is to analyze the flow in the S3Ch test section under various flow conditions and geometrical configurations in order to develop a numerical twin of the test section. The empty test section is investigated at a Mach number of 0.7. The test conditions are achieved either under fully subsonic conditions using a straight test-section geometry, or under transonic flow conditions through the use of a sonic throat, which represents a particularly challenging case for CFD solvers. The study is conducted within the framework of the NATO activity and is intended to serve as a benchmark for several CFD solvers and meshing techniques, with the aim of identifying best practices for this type of configuration.

I. Nomenclature

AoA	=	Flow incidence, deg
M	=	Mach Number
MFR	=	Mass flow rate
P	=	Static pressure, Pa
P_i	=	Stagnation pressure, Pa
$P_{i,0}$	=	Inlet stagnation pressure, Pa
P_{out}	=	Outlet static pressure, Pa
U	=	Longitudinal velocity, m/s

*Research Engineer, Department of Aerodynamics, Aeroelasticity, Acoustics, Jean-Luc.Hantrais-Gervois@onera.fr

†Research Engineer, Department of Aerodynamics, Aeroelasticity, Acoustics, Nicolas.Rembaut@onera.fr

‡Research Engineer, Department of Aerodynamics, Aeroelasticity, Acoustics, Geoffrey.Tanguy@onera.fr

§Research Scientist, Department of High-Speed Configurations, Michael.Werner@dlr.de

||Lecturer in Aerodynamics, Centre for Aeronautics, davide.dipasquale@cranfield.ac.uk, Senior AIAA member

||Research Engineer, Aerodynamics and Engineering Analysis Department, bnasuhbeyoglu@roketan.com.tr

**Research Engineer, Configuration Aerodynamics Branch, AIAA Associate Fellow

U_e = Reference velocity within the test section, m/s
 V_x = Velocity in the x direction, m/s
 Δ_t = time step, s

II. Introduction

A. Context

Computational Fluid Dynamics (CFD) has become a major complementary tool to experimental wind-tunnel testing. The simulations enable access to detailed three-dimensional descriptions of the flow field, often providing a level of spatial resolution that exceeds what can be obtained through experimental measurements, which are typically limited to discrete probes or measurement planes. Beyond traditional free-flight or infinite-domain simulations, CFD is increasingly employed to accurately reproduce actual wind-tunnel test conditions in order to improve the comparison with experimental data. Representing the wind tunnel makes it possible to assess the influence of wall confinement, test-section flow non-uniformity, and support systems on measured aerodynamic characteristics, which are not present in free-flight conditions..

In addition, once validated, wind-tunnel simulations can provide detailed insight into the flow field within regions of the facility that are difficult to instrument or physically access. Computational Fluid Dynamics can also be used to assess the aerodynamic impact of geometric modifications during the design process of a new model or measurement setup, as well as during the process of improving the facility.

However, numerical simulation of wind-tunnel facilities remains a challenging task due to the wide variety of tunnel geometries and the broad range of flow regimes encountered, which generally require the development of case-specific modeling strategies. Simulations of the entire wind-tunnel circuit remain relatively rare and usually require the use of models to represent certain components of the circuit, such as the fan or the honeycomb mesh.[1][2][3]. Most of the time, numerical studies are limited to key sections of the facility, including the settling chamber, contraction, test section, and diffuser [4]. The mesh size employed in these simulations varies significantly depending on the complexity of the test-section configuration, such as open or closed test sections [5][6], permeable or slotted walls [7] [8], or including the model mounting supports (sting, struts or floor mounted for example) [9][10].

When properly used, wind-tunnel simulation can be a powerful tool for improving the understanding of wind-tunnel measurements. Numerical simulations of the ONERA S2MA wind tunnel [11] enabled the development of wall-interference correction models for high-Mach-number flows, a non linear regime where conventional potential-flow equations are no longer accurate. These models can then be used for the correction of local flow angles and Mach number around the wind tunnel model in order to obtain the aerodynamic forces and moments coefficients. Recent work from Mouton et al. [12] also investigated the validity of the current wind tunnel correction processes based on the simulation of the ONERA F1 wind tunnel with the CRM-HL model within the framework of the High lift prediction workshop. However, increasing simulation complexity is often associated with a corresponding increase in mesh resolution requirements. High-fidelity studies have reported mesh sizes reaching up to several hundred million cells, with some simulations reaching 500 million grid elements [8].

This paper is conducted within the framework of the NATO Applied Vehicle Technology (AVT) Research Task Group AVT-387, which is dedicated to the investigation of four Common Research Wind/Water Tunnels which represent the different fluid dynamic regimes of most fluid dynamic testing of vehicles. This activity follows earlier NATO efforts including AVT-284 RSM on “Advanced Wind Tunnel Boundary Simulation” [13] and AVT-338 RSM, “Advanced Wind Tunnel Boundary Simulation II” [14].

The objective of the present study is to assess the flow field within the high-speed leg of the ONERA S3Ch wind tunnel (see figure 1) for a range of empty test section configurations. The analysis is based on computations performed using multiple CFD solvers and meshing strategies, with the goal of identifying best practices for the accurate numerical simulation of wind tunnels.

B. Previous validation of numerical tools

A previous paper describing the AVT-387 collaborative validation activity based on the S3Ch facility has been published (see [15]). This previous work enabled the comparison and validation of CFD solvers and turbulence models across multiple grid resolutions. Despite the apparent geometric simplicity of the S3Ch high-speed leg, its numerical simulation was found to be challenging. Two distinct wind-tunnel configurations were investigated at an operating Mach

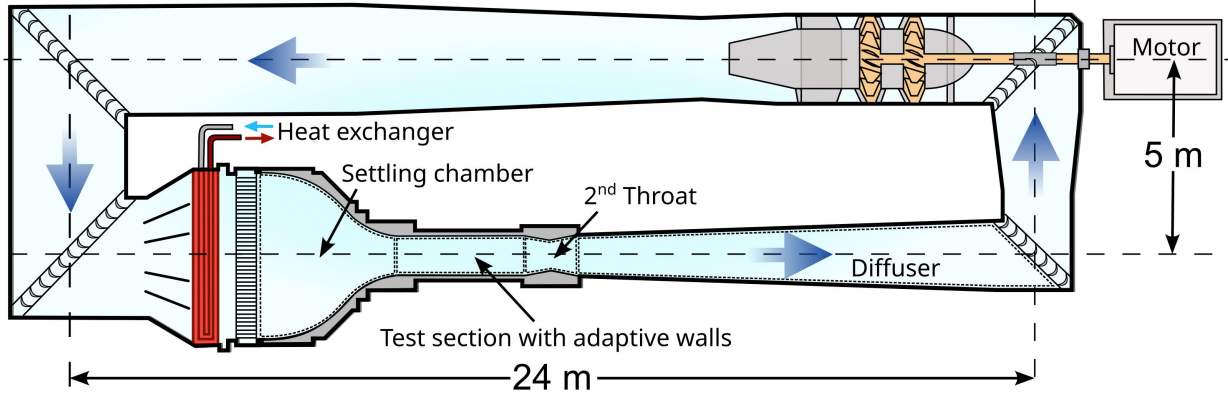


Fig. 1 Illustration of the ONERA S3Ch full circuit scheme

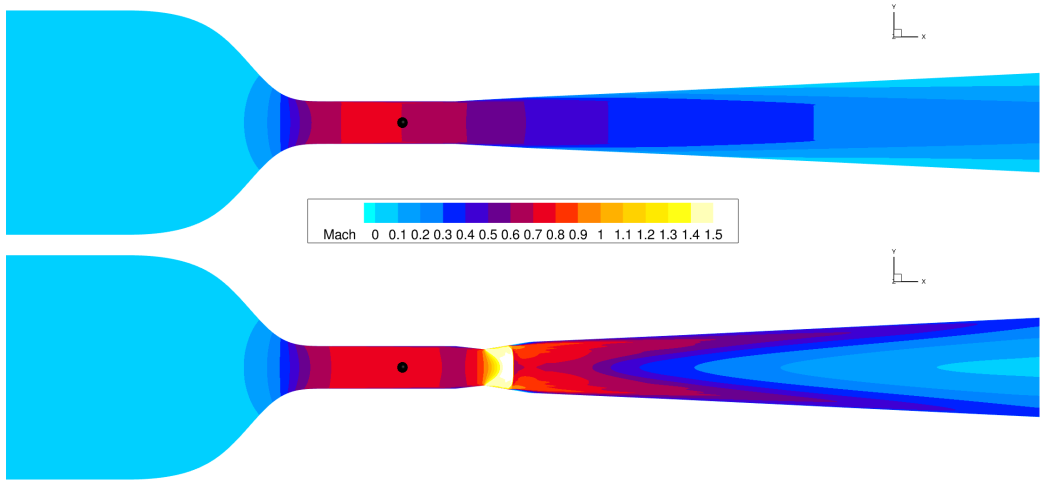


Fig. 2 S3Ch main flow regimes (CFD fields): without (top) and with sonic throat (bottom)

number of 0.7. These configurations corresponded to a fully subsonic case with a straight test section (Case A, see figure 2 top) and a transonic configuration employing a sonic throat (Case B, see figure 2 bottom).

In comparison with the experiments, the following findings could be obtained:

- The subsonic case behaved very well from both a numerical and a physical standpoint in the test section, thereby enabling detailed comparisons with the experiment and between the partners (case A, see following chapters).
- The configuration with the shocked second throat remained challenging from a numerical point of view, and further validation is required (case B, see following chapters).
- The main challenge was identified in the diffuser (for both the subsonic case A and the shocked second throat B). Significant differences with the experiments were observed, such as a more pronounced loss of total pressure in the real diffuser. On the numerical side, the positive pressure gradient in the diffuser led to different behaviors among the simulations, (more or less prone to thin flow separation).

Previous simulations differed from the experiments due to the absence of the return circuit, but also because of geometric simplifications that may affect the boundary-layer state (e.g., the step at the end of the adaptive walls and the precise shape of the second throat). In addition, the experimental flow state in this region is difficult to accurately measure. Therefore, the present paper further investigates the impact of the step, as well as the influence of the rest of the circuit, modeled through an inhomogeneous inlet condition.

III. The ONERA S3Ch wind tunnel

A. Wind tunnel description

The ONERA S3Ch is a closed-circuit atmospheric transonic wind tunnel featuring adaptive walls (see figure 1 and reference [16]). The air is put in motion thanks to a 3.5 MW electric motor operating at a constant rotational speed of 1500 rpm. The Mach number in the test section can be varied between approximately 0.2 and 1.3, allowing access to subsonic, transonic and supersonic regimes. The settling chamber is a central component of the wind tunnel, designed to homogenize the flow before it enters the test section. It has dimensions of approximately $4.2 \times 4.2 \times 3$ meters and contains several flow conditioning devices. A heat exchanger with a power of 2.5 MW is located at the upstream end of the settling chamber. Its role is to regulate the stagnation temperature of the flow, which typically lies between 300 K and 320 K. Downstream of the heat exchanger, a honeycomb structure and two turbulence grids are installed to reduce large-scale flow non-uniformities and turbulence intensity. Despite these conditioning devices, measurements in the settling chamber reveal residual flow heterogeneities, including spatial variations in velocity and temperature. Upstream of the test section, a convergent nozzle accelerates the flow to the desired Mach number.

B. High speed leg features

The high speed leg corresponds to the portion of the wind tunnel extending from the settling chamber to the diffuser (figure 3). In the settling chamber air velocity is low (about 6 m.s^{-1}). The flow is then accelerated through the convergent (the contraction ratio is 29.1) and reaches transonic conditions in the test section. After the test section, a second throat can be used to achieve a highly stable Mach number in the test section. In that case, the left and right walls are folded in a "V shape" to form a minimum section where the Mach number reaches 1. Beyond this throat, Mach number increases up to a point where the supersonic flow cannot be sustained anymore by the engine and a shock forms in the diffuser (figure 3b).

The tunnel can also be operated without the second throat. In this configuration, the flow beyond the test section remains subsonic and decelerates steadily in the diffuser (figure 3a).

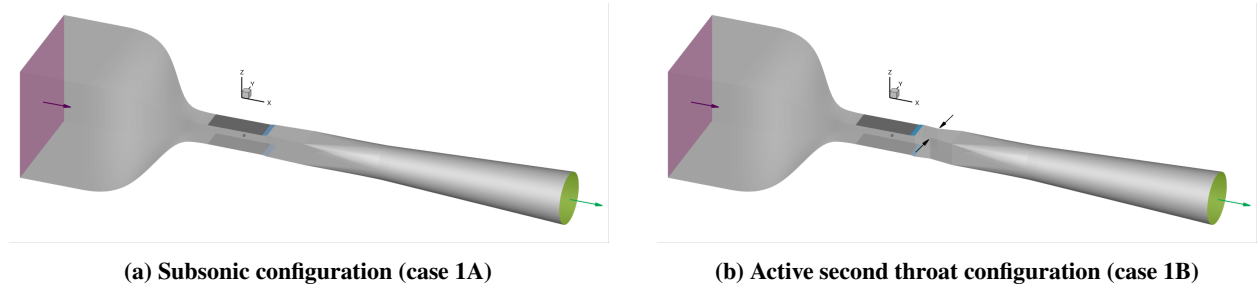


Fig. 3 S3Ch high speed leg test cases

C. Test section description

The test section itself is a rectangular test section by 0.804 m (width) and 0.764 m (height) (test section area 0.614 m^2). The test section is 2.020 m long. The wind tunnel test section can accommodate several sets of top and bottom walls for various purposes. For two-dimensional tests, wall interference need to be mitigated and top and bottom walls feature adaptive full walls. This is the usual setting and it has thus been used for this empty test section validation activity. As no model is present in the tunnel, the walls are flat. Other sets of walls can be mounted, such as flat walls without or with screens to enable more optical access for measurements and also flat porous walls. A slight divergence of the top and bottom walls is introduced to compensate for boundary-layer growth along the four walls.

D. Inlet and test section measurements

For this validation exercise, several empty test section measurements have been made available (see figure 4). These measurements are detailed in [17]. Along the full circuit, static pressure measurements are acquired on the top wall. It is worthwhile to notice that these top wall measurements are not always acquired on the top wall and may result from

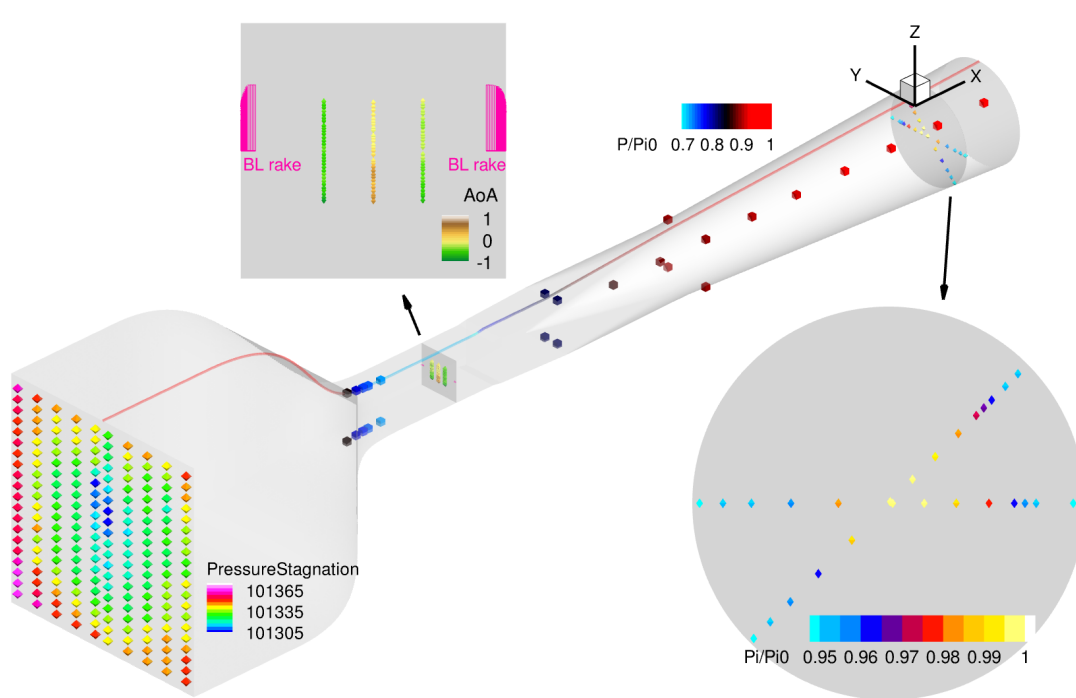


Fig. 4 ONERA S3Ch measurements used in this paper

averaging of several pressure tapes. Thus, for clarity, the numerical results will be taken strictly along the top wall center and compared to the measurements.

1. Settling chamber

Through the return circuit with the corners, the screens and vanes, flow arrives in the settling chamber with an history. Even though the circuit is designed to maximize the flow homogeneity, some non-homogeneity is present in the settling chamber. A rake featuring several probes has mapped the entrance of the settling chamber. Turbulence level, total pressure, total temperature and velocity measurements are available. These measurements show a non-uniformity of the flow (velocity deficit in the middle, and left/right asymmetry).

2. Test section

In the test section, several flow velocity and flow angularity measurements have been acquired. In this paper the flow angularity measurements at the location of the model are used. They exhibit some non-homogeneity and spanwise variation. Some boundary layer measurements on the right and left walls are also used (those measurements will only be indicative as they have been acquired at Mach number 0.5 and not 0.7). Although many static pressure measurements are spread on the adaptive walls (top and bottom), they serve for the wall adaptation process. As the walls remain flat in the empty test section, this data hasn't been used.

3. Diffuser

The diffuser can be equipped with a rake in order to perform total pressure measurements. The rake can also rotate to acquire total pressures within two azimuthal orientations.

IV. High speed leg test case descriptions

In order to assess the capabilities of RANS models for the modeling of the S3Ch transonic tunnel, two test cases have been prepared. They correspond to the fully subsonic high-speed leg (case A, see figure 3a) and the case with the second throat active (case B, see 3b). For both geometries, the aerodynamic conditions correspond to a Mach number M

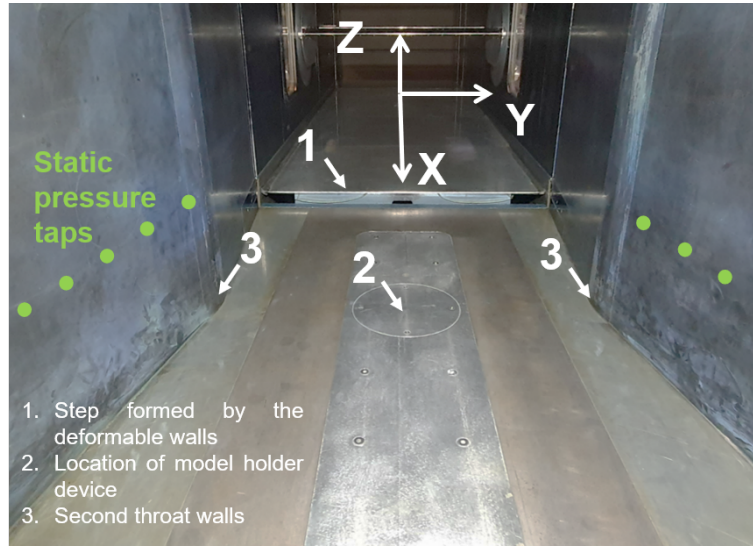


Fig. 5 ONERA S3Ch detailed view of the geometry of the end of adaptive wall and of the second throat

of 0.700 in the test section:

- Atmospheric total conditions with total pressure = 101325.0 Pa and total temperature = 300.0 K
- A numeric turbulence level of 10^{-4}
- A Mach number of 0.700 is imposed at the tunnel center located at 1.2 m along the tunnel axis (this corresponds to the model rotation center, see for instance figure 3a)

A. High speed leg in subsonic setting (case A)

In that case (see figure 3a), the pressure at the end of the diffuser must be adjusted to achieve the target Mach number at the reference position. Experimentally, this is done by setting the engine to the appropriate operating condition. Numerically, this is achieved by setting the appropriate pressure on the boundary condition. This pressure needs to be set iteratively, either by an automatic procedure or manually.

B. High speed leg with the sonic second throat (case B)

In that case (see figure 3b), the experimental procedure consists of choking the second throat at a small cross-sectional area. The throat area is then progressively increased to reach the target Mach number at the reference position. The engine settings are adapted so as to locate the shock between two pressure measurements on the ramp downstream of the throat (see the green pressure taps in figure 5). The throat section is measured.

The corresponding numerical case features the geometry of the throat based on the measured throat section in the experiment. The numerical outlet condition must then be adjusted to position the foot of the shock in the appropriate region. At this point, it is important to indicate that the throat section has been simplified for the numerical exercise. The real geometry is smooth, whereas the CAD of the proposed test case is sharp (see point of interest 3 in figure 5). Also, some downstream details are not modeled in the CAD (e.g. cavities at the foot of the moving walls that create the throat).

C. End of test section geometry

It is important to indicate that in the wind tunnel, the adaptive-wall section ends with a step (see point of interest 1 in figure 5). The height of this step depends on the wall divergence and the wall adaptation. For the empty test section, the adaptable walls are flat and end with a deviation by 30mm. The step is not a solid step as some fluid can circulate below the adaptable walls (cavity flow).

In previous numerical studies, this feature was simplified by smoothing the step using an inclined wall segment (see [15]). In the present work, the impact of the step is explicitly investigated.

V. Numerical investigation

A. CFD solvers

Various CFD solvers have been used in this study as well as in the previous validation effort [15]. This section presents the list of solvers employed and briefly describes their main numerical features.

1. *elsA numerical features*

The computations are based on RANS modeling using the elsA software, co-owned by ONERA and SAFRAN [18]. The solver relies on a finite-volume formulation (cell-centered) and supports both structured and unstructured grids.

In the present study, structured schemes are used together with an implicit LU scheme. A second-order central-difference scheme is applied for the spatial discretization of the mean flow (Jameson–Schmidt–Tukel scheme with second- and fourth-order dissipation terms), while a second-order upwind scheme (Roe scheme) is used for the turbulent quantities. The system of equations is solved in a fully coupled manner. Time integration is performed using a first-order backward Euler scheme. Local time stepping and multigrid techniques are employed to accelerate convergence. Regarding turbulence modeling, based on previous validation work [15] simulations are performed with the $k-\omega$ Menter SST (with Zheng limiter) in combination with the QCR correction (2000 version) [19].

2. *CODA numerical features*

The second solver used in this study is CODA, a CFD software developed jointly by ONERA, DLR, Airbus, and their European partners [20]. CODA is based on a cell-centered finite-volume approach for solving the Reynolds-averaged Navier–Stokes equations on unstructured meshes. In the present work, convective fluxes of the mean flow are discretized using a second-order Roe scheme with a spline-quintic limiter, while turbulent fluxes are discretized using a first-order Roe scheme. Cell gradients are computed using a least-squares approach. Turbulence was modeled using the one-equation Spalart–Allmaras turbulence model in the negative formulation [21], in combination with the QCR extension [19]. Both steady (RANS) and unsteady (URANS) simulations are performed. For steady simulations, a strongly implicit backward Euler scheme is used, while unsteady simulations rely on a second-order Rosenbrock scheme. The time step for unsteady simulations is set to $\Delta t = 5 \times 10^{-6}$ s.

3. *CFD++ numerical features*

CFD++ is a general-purpose CFD solver developed by Metacomp Technologies. It is capable of simulating compressible and incompressible flows, including turbulence, heat transfer, and multi-physics phenomena. The solver supports both structured and unstructured grids, a wide range of turbulence models (RANS, LES, and hybrid approaches), and high-performance computing for large-scale simulations. It is used here as one of the reference solvers for comparison.

4. *STAR-CCM+ numerical features*

STAR-CCM+ is a commercial CFD software developed by Siemens Digital Industries Software. It provides an integrated environment for the simulation of fluid flow and multi-physics problems on complex geometries. The solver includes advanced meshing capabilities and supports a wide range of turbulence models and physical phenomena. In this study, it is used as a reference industrial CFD tool for comparison purposes.

5. *Roketsan's in-house numerical feature*

Roketsan's in-house sustained CFD software (ROKA) is a powerful and customized flow solver to support the aerodynamic design, optimization, and database processes of air vehicles. Covering a wide range of flow regimes from low subsonic to hypersonic speeds, the software is capable of simulating incompressible and compressible turbulent flow problems using the finite volume method for compressible Navier–Stokes equations. As a density-based ideal gas solver, it includes various boundary conditions for internal and external flow analyses, a rich set of turbulence models, and both implicit and explicit time integration methods, enabling time-dependent and steady-state simulations. The software is highly scalable in high-performance computing environments to ensure efficiency in large-scale simulations. It provides the flexibility to handle unstructured meshes with various types of element topology coexisting.

6. *Fluent numerical features*

Simulations were also performed using the commercial CFD software ANSYS Fluent. The solver employs a finite-volume discretization of the Navier–Stokes equations. In this study, a density-based implicit solver is used with a second-order central-difference scheme for spatial discretization. Pressure–velocity coupling is achieved using the SIMPLE algorithm. Four turbulence models were investigated: three eddy-viscosity models—Spalart–Allmaras, $k-\omega$ SST (Menter), and GEKO—augmented with the Quadratic Constitutive Relation (QCR) correction, as well as the stress- ω Full Reynolds Stress Model.

7. *FUN3D numerical features*

FUN3D is an unstructured, three-dimensional, implicit Navier-Stokes solver that is nominally second-order spatially accurate. The simulations employ a node-based finite-volume method [22] where the flow variables are stored at the mesh vertices. At the interfaces delimiting neighboring median dual control volumes, inviscid fluxes are computed using the Roe approximate Riemann solver. To achieve second-order accuracy, interface values are extrapolated from the control volume centroids using gradients computed at the mesh vertices via an unweighted least-squares technique. The Barth and Jespersen reconstruction limiter [23] is applied and frozen after 3,000 iterations. Full viscous fluxes are discretized using a finite-volume formulation, with the necessary velocity gradients on the median dual faces computed via the Green-Gauss theorem. The solution is updated at each time step using a backward Euler time-integration scheme, and the linear system of equations is approximately solved using a point-implicit procedure. Local time-step scaling is employed to accelerate convergence to a steady state. Turbulence is resolved using the Spalart–Allmaras model with the negative turbulence variable provision (SA-neg) [21] and the SA-QCR2000 model [19].

8. *USM3D numerical features*

The USM3D flow solver, developed at NASA Langley Research Center is a cell-centered Reynolds-averaged Navier–Stokes (RANS) solver integrated into the NASA Tetrahedral Unstructured Software System (TetrUSS). USM3D and TetrUSS have been widely used for configuration aerodynamic research within NASA (see [24]), [25]) and for product development in major aerospace companies (see [26]), [27]). The version used for this paper is the extended to mixed-element grids (USM3D-ME, see [28], [29] and [30]). Additionally, algorithmic improvements have been implemented to enhance solver robustness and reduce computational time.

A new solution methodology, the Hierarchical Adaptive Nonlinear Iteration Method (HANIM), has been implemented in USM3D-ME (see [31] and [32]). HANIM is a robust nonlinear method that improves the efficiency and automation of Reynolds-averaged Navier-Stokes (RANS) solutions. HANIM introduces two additional hierarchies to the simple preconditioner of USM3D-ME, which is based on a linearized discrete formulation and a point-implicit Gauss-Seidel relaxation scheme. The first hierarchy is a matrix-free linear solver that employs Fréchet derivatives and a generalized conjugate residual method to solve the exact linearization of the RANS equations. The second hierarchy is a nonlinear control mechanism that automatically adapts the operational pseudo-time step to accelerate convergence and overcome transitional instabilities and limit cycles. Turbulence is resolved using the Spalart–Allmaras model with the negative turbulence variable provision (SA-neg) [21] and the SA-QCR2000 model [19].

B. Grids

Various meshes were employed in this study, including both structured and unstructured grids. The structured meshes were extracted from a previous study [15], whereas the unstructured meshes were newly generated for the present work. To investigate the effect of the terminating step of the adaptive wall, two mesh configurations were defined in the vicinity of the step located at the downstream end of the test section.

For the structured mesh, the step is added on the existing former meshed (figure 6) thanks to a Chimera patch (figure 7). The step mesh refinement is adapted to correspond to the underlying mesh refinement. For the coarse mesh ($3.7 \cdot 10^6$ elements) the step mesh adds $1.6 \cdot 10^6$ elements, whereas for the superfine mesh ($78.7 \cdot 10^6$ elements), the step mesh adds $43.9 \cdot 10^6$ elements.

A set of unstructured mesh with and without the step (for both the subsonic and the shocked configurations) has also been made for this study. This mesh corresponds to the coarse level used in the previous validation exercise. Two unstructured grids were generated using the Fidelity Pointwise meshing software: one without the step and one with the step. To ensure consistency with the structured mesh results, similar boundary-layer parameters were applied. The initial surface spacing was set to an average value of 60 mm, with local refinement applied in the test section and second

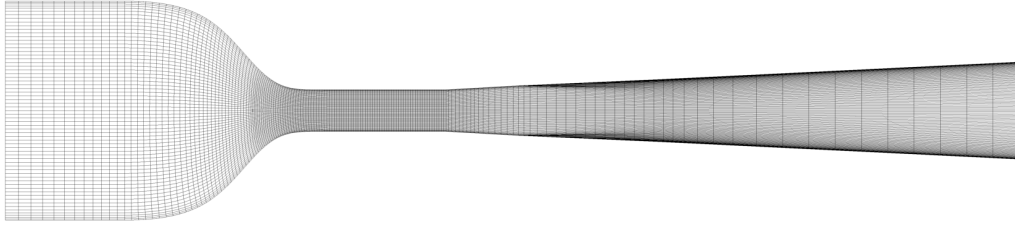


Fig. 6 Structured coarse mesh

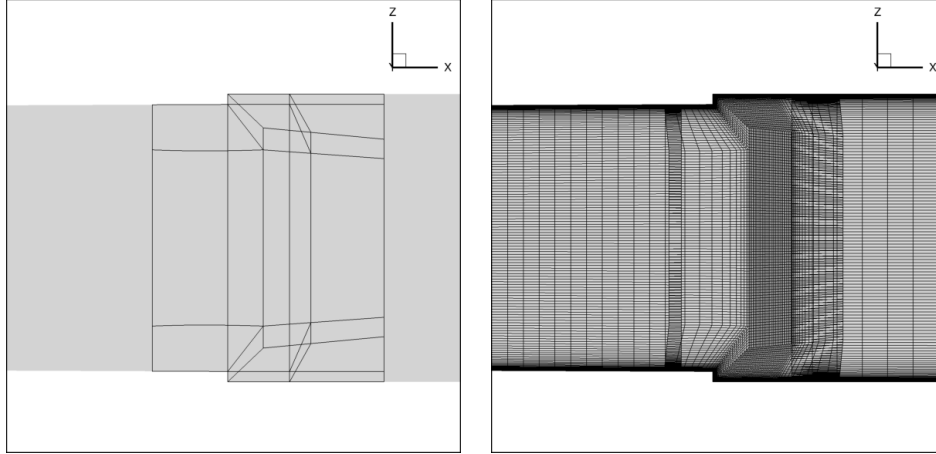


Fig. 7 Chimera step meshing approach (coarse mesh)

throat regions, where the spacing was reduced to 45 mm. Additional refinement was introduced in corner regions, with a further local reduction of 5 mm, to better capture corner-flow effects. The boundary layer mesh was generated using the T-Rex algorithm, with wall-normal parameters consistent with the structured mesh: a first cell height of $3.5\mu\text{m}$, a growth rate of 1.15, and a minimum of 50 layers. The resulting unstructured meshes are therefore more refined in terms of cell count than their structured counterpart. The coarse structured mesh contains approximately 4 million cells, whereas the unstructured meshes contain approximately 9 million cells. Figure 8 and figure 9 show the unstructured meshes without and with the step, respectively. Once the nodes are created, the walls can be meshed and refined to minimize the average cell area.

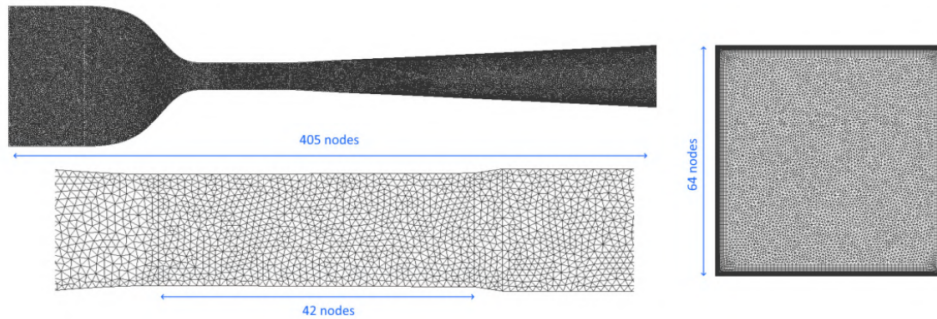


Fig. 8 Unstructured mesh without step

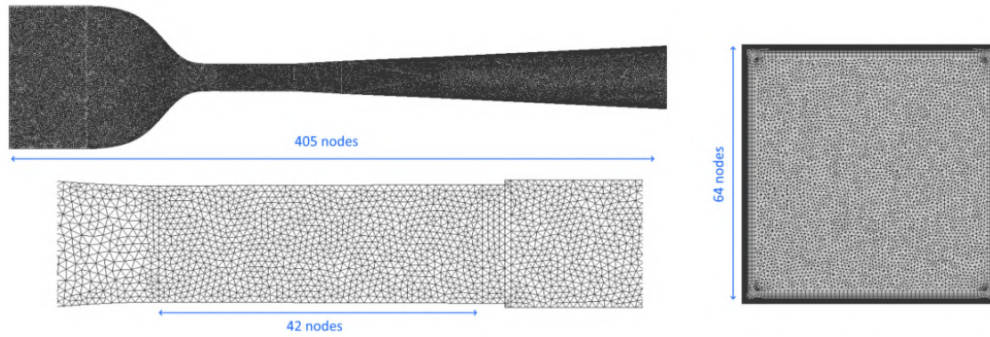


Fig. 9 Unstructured mesh with step

VI. Numerical investigation

A. Assessment of the flow for subsonic conditions (case A)

1. Overall Case A simulation (reference configuration)

A complete account on the Case A simulations can be found in [15]. Many partners compared their solutions on the high speed leg with a setting of the Mach number in the middle of the test section at $M = 0.700$. The simulations have been carried out on a common set of meshes (hexas meshes with four levels of mesh density to allow for mesh convergence analysis). Different RANS solvers have been used in combination with various turbulence models (ranging from one equation Spalart-Allmaras to seven equation RSM).

The main findings are summarised in this section. All simulations are crossplotted in figure 10. In this figure, the evolution of pressure and Mach number along the high speed leg is presented. A zoom in the test section (right) exhibits the test section Mach number achieved. The range for all simulations around the targeted $M = 0.700$ is within ± 0.001 which enables precise comparisons. The Mach number gradient is very coherent within the simulations. This coherence is stressed in figure 11 (left) where a comparison of the boundary layers on a lateral wall is provided. The experiments are not exactly matched, but they are indicative only (they have been acquired for another test condition, $M = 0.500$). The main difference between the simulations consists in the blending between the linear law close to the wall and the log law, owing to the different turbulence models (mainly Spalart-Allmaras versus $k-\omega$ Menter SST behaviour).

Beyond the test section, the comparison to the experiments (figure 10, left picture) shows a discrepancy for all simulations in the diffuser. The simulations all tend to conserve better total pressure than the experiment. This is detailed at the end of the diffuser (see figure 11, right). In the diffuser, the boundary layers are well developed and are comparable in size to the inviscid flow in the middle. The simulations exhibit very contrasted results in this part of the circuit and the comparison with the experiment is poor. It can be noticed that the experiments are not symmetrical.

Thanks to the contributors of this validation exercise, some numerical settings could be identified:

- The common mesh family allowed for a mesh sensitivity analysis and the coarse mesh proved already pertinent for this validation exercise (about $5 \cdot 10^6$ elements).
- The Spalart-Allmaras simulations exhibited a good coherence among the various software implementations. For some models, the software implementation plays a significant role that may override the turbulence model expression. Some completely different solvers could reach excellent agreement with different turbulence models. The comparison enabled to identify undesirable behaviours in some simulations. The points of singularity between the solutions could be seen on the boundary layer profiles, the turbulence intensity (in the corners, but also in the mean flow) and in the diffuser behaviours (core flow shape and tendency to separate).
- The quadratic constitutive relation (QCR[19]) mitigates the slight flow separation behaviours present in some simulations (as in the diffuser).

Building on the numerical parameters identified by each partner to carry out the simulations, the modelisation aspects mentioned previously can be analysed.

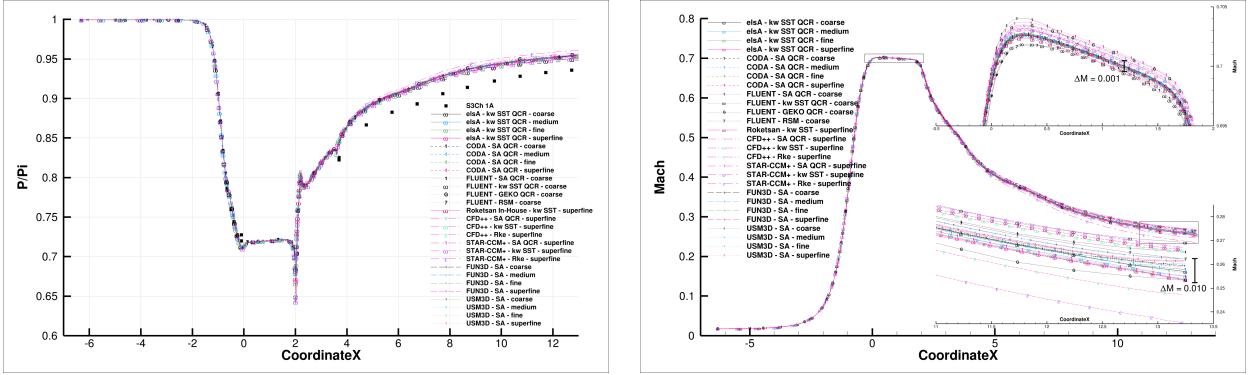


Fig. 10 S3Ch top wall pressures (left) and centerline Mach number (right) for all simulations

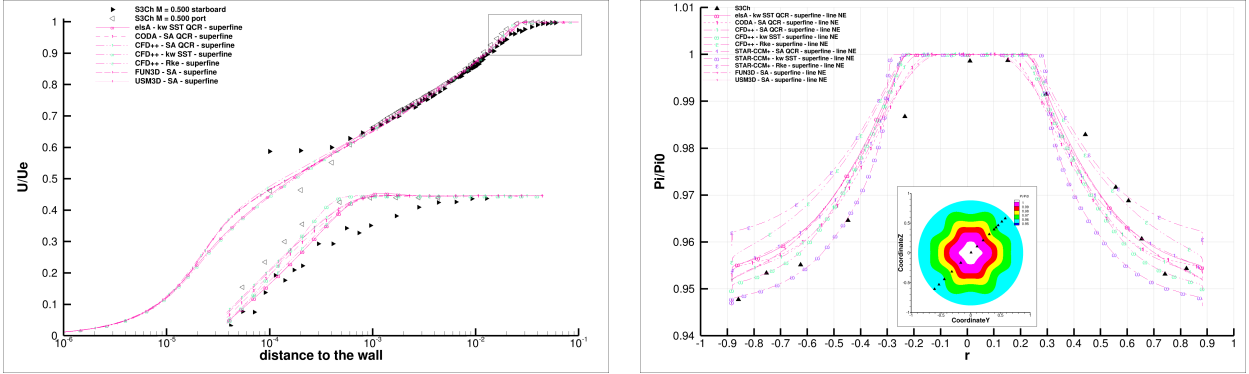


Fig. 11 comparison of simulations on the superline mesh level on the tunnel center boundary layer (lateral wall, see left) and on the diffuser exit plane (see right)

Mesh Type	Turbulence Model	$P_{0,in}$ (Pa)	P_{out} (Pa)	Target M	CFD M	ΔM	MFR (kg/s)
Structured	SA-QCR	101325	96895	0.7	0.6995	0.0005	131.49
Unstructured	SA-QCR	101325	96740	0.7	0.6999	0.0001	131.56

Table 2 Case A simulation parameters (without the step)

2. Impact of test section geometry

The impact of the test-section geometry is assessed by comparing configurations with and without the step located at the end of the adaptive walls. This geometric feature introduces a local discontinuity that may influence the downstream development of the flow.

As a preliminary step, the solution obtained on the newly generated unstructured mesh is compared with that obtained using the reference structured mesh for the baseline configuration without the step. The comparison is performed using the Spalart–Allmaras turbulence model with QCR correction. The boundary conditions applied in the simulations, as well as the resulting Mach number within the test section, are reported in Table 2.

Between the two meshes, the outlet pressure was slightly adjusted to achieve the target Mach number of 0.7, with an accuracy of approximately 0.0005. The CFD results show that both meshes provide essentially identical flow solutions for the configuration without the step. As illustrated in Figure 12, the unstructured mesh provides a level of accuracy comparable to that of the structured mesh, with nearly identical distributions of Mach number along the centerline and static pressure along the top wall. The boundary-layer profiles upstream of the step, extracted at a streamwise location of $x = 2.0$ m, also show excellent agreement between the structured and unstructured meshes (Figure 13).

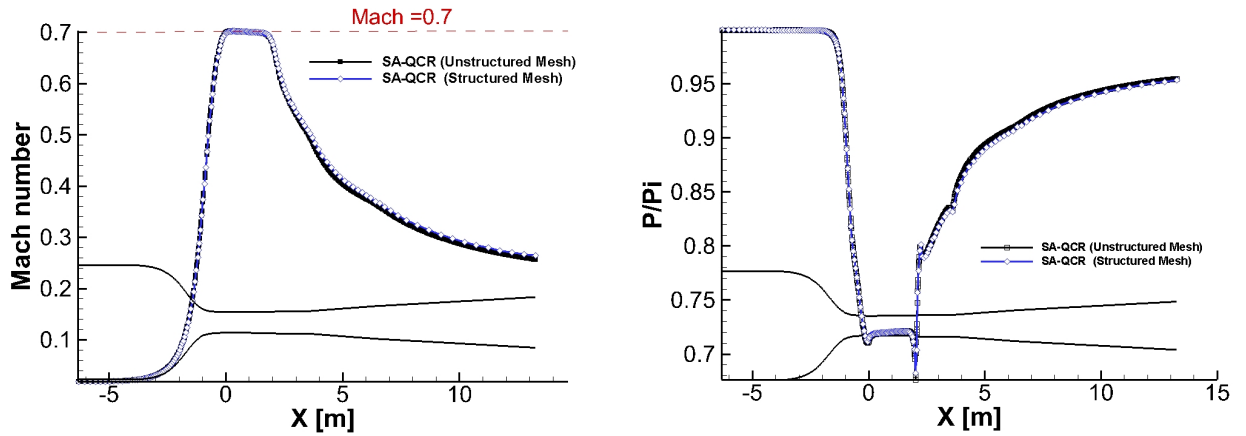


Fig. 12 Static pressure comparison along the top wall and mach number along the centre line

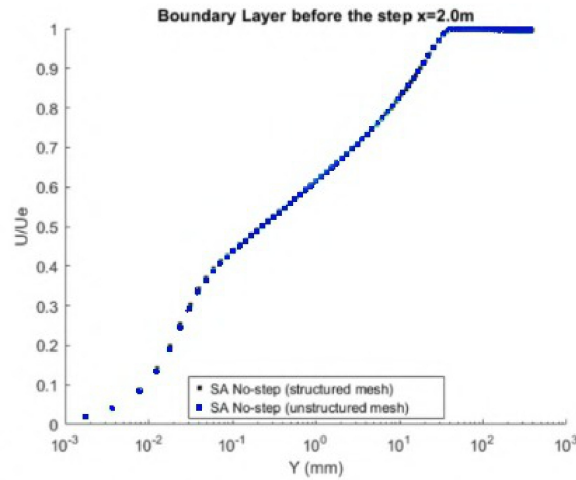


Fig. 13 Boundary layers

The impact of the step is then evaluated using two turbulence models: Spalart–Allmaras with QCR correction (SA-QCR) and $k-\omega$ SST with QCR correction. The corresponding boundary conditions and target Mach number are

Turbulence Model	$P_{0,in}$ (Pa)	P_{out} (Pa)	Target M	CFD M	ΔM	MFR (kg/s)
SA-QCR	101325	95300	0.7	0.6997	0.0003	131.65
k- ω -SST-QCR	101325	94600	0.7	0.7004	-0.0004	131.53

Table 3 Case A simulation parameters (with the step)

summarized in Table 3. Both simulations achieve Mach numbers in close agreement with the target value, and their mass flow rates are sufficiently consistent to indicate satisfactory convergence.

As shown in Figure 14b, the step induces a recirculation region downstream of the geometric discontinuity, with an extent comparable to that obtained with the smoothed-wall representation. However, more pronounced differences are observed in the boundary-layer development. The presence of the step leads to a thickening of the boundary layer, which in turn increases total pressure losses in the diffuser, particularly along the top and bottom walls. As a results, the distribution of the static pressure on the top wall is impacted by the presence of the step. The level of pressure ratio decrease within the diffuser which provide a better agreement compared with the experimental data (figure 15). The relative difference between the CFD results and the experimental data for the pressure within the diffuser is below 1% when the step is represented for both turbulence models.

Within the diffuser, the loss of total pressure close to the wall modifies the shape of the central inviscid. In the configuration with the step, total pressure loss contours are compressed toward the center, indicating a stronger influence of viscous effects near the walls. The in-flow measurements at the end of the diffuser can be compared with these new results (see figure 16 presenting the results obtained on the structured meshes, with or without the step). The central inviscid core is narrower when accounting for the step (especially on the right picture presenting a rake getting closer to the top and bottom walls). However, agreement with experimental data is not improved in this region. The experiment shows a large asymmetry that cannot be reproduced in the simulations. This asymmetry may find its origin in the rest of the circuit as the first corner is just downstream of this instrumented section.

Overall, the step is shown to have a significant impact on the downstream flow. A simulation of the whole circuit would need to represent the step in order to model properly the total pressure losses in the circuit. Nevertheless, if one is interested in the test section flow only, as can be the case for wall interference simulation, the step introduces a numerical complexity that is not necessary.

3. Impact of non-uniform inlet conditions

It has been seen that the S3Ch measurements are not symmetric (diffuser, for instance, but this is also the case in the test section). Due the return circuit with the corners, the screens and vanes, this is only natural. The simulations of the high speed leg (fully symmetric geometry) cannot show anything but a symmetric flow. The purpose being to mimic the tunnel flow field with CFD, modeling asymmetry is of importance. Modelling the full tunnel comes with a cost and bring some modelling and validation challenges. Another option is to represent the non-symmetric features at the inlet of our computational domain in order to account for the integration of the history of the flow in the circuit. S3Ch benefits from measurements in the inlet (total pressure, total temperature and some velocity measurements, see 17). As the inhomogeneous flow is subtle, preceding comparisons on the tunnel top wall or at the centerline won't exhibit much effect. Thus, another set of measurements in the flow field are used: some local angle of attack in the test section. A five-hole probe has acquired 3 vertical lines of measurements at the model position (see reference [17] and figure 17). These measurements exhibit noticeable variations despite the relative proximity of the measurements.

Again, the simulations have been carried out on both the coarse and the superfine meshes. The results of the introduction of the inhomogeneous total pressure is presented in figure 18. On the left, the situation without inhomogeneity is symmetric and exhibits no angularity. When inhomogeneity in total pressure is introduced on the inlet, a significantly different result is achieved in the middle of the test section. Pockets of flow angularity are present. The experimental map is not recovered, but the order of magnitude of the effect is well present. The superfine mesh produces a more defined result than the coarse mesh, but with the same flow topology (more pronounced peaks and valleys, not shown here). The total temperature does not affect much the flow field (it behaves like a passive scalar).

As the injection of total pressure variation is very effective in modifying the test section flow field, attempts of inlet total pressure map tuning have been carried out. An optimisation method has been used to act on total pressure on the inlet, so as to obtain the experimental values in the middle of the test section. A simple parametrisation has been

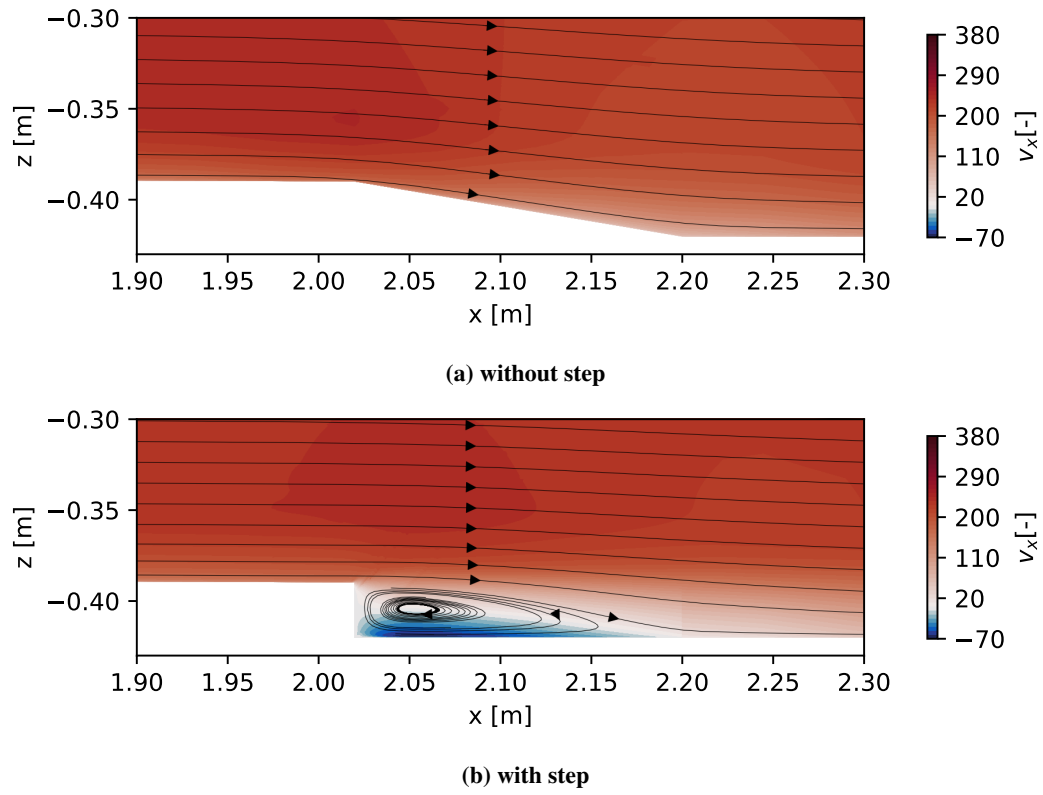


Fig. 14 Distribution of the x-normal velocity component near the location of the step, CODA

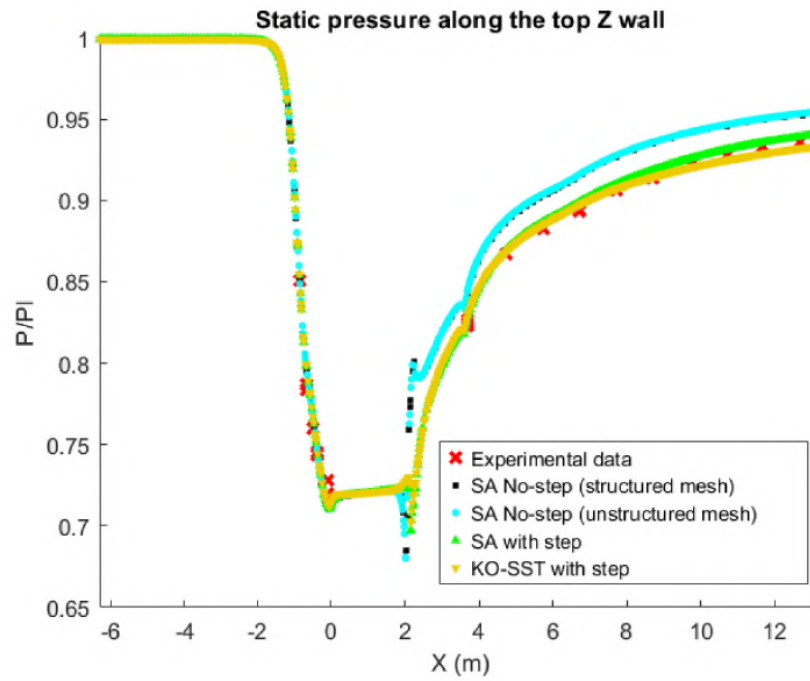


Fig. 15 Static pressure comparison along the top Z wall

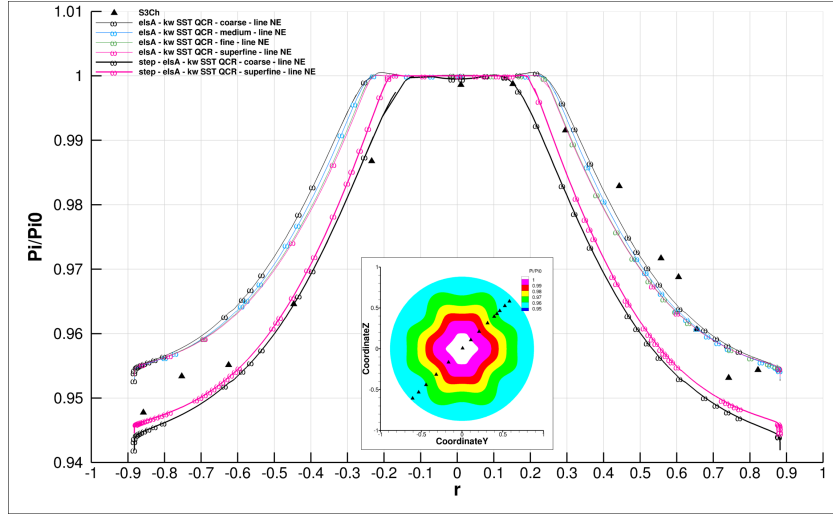


Fig. 16 S3Ch total pressure losses at the end of the diffuser

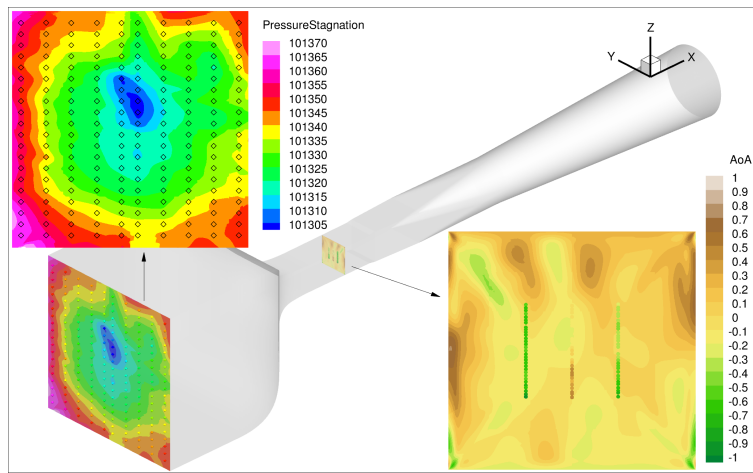


Fig. 17 S3Ch measurements in the settling chamber at the inlet of the computational domain and S3Ch flow angularity measurements in the test section superposed to the CFD simulation.

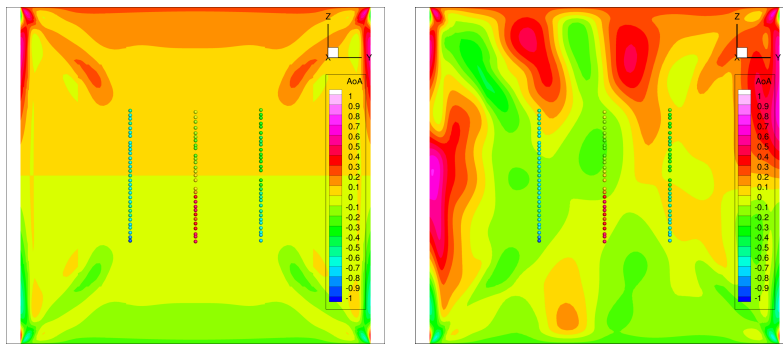


Fig. 18 Numerical flow angularity in the test section compared to the five holes measurements (symbols) (left: CFD without inhomogeneous inlet, coarse mesh, right: CFD with inhomogeneous inlet, coarse mesh).

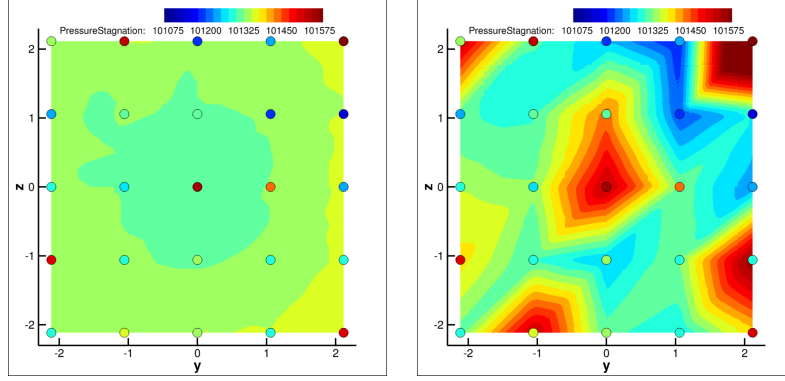


Fig. 19 Inhomogeneous inlet parameterisation (total pressure measurements with parameters on the left and adapted numerical inlet total pressure).

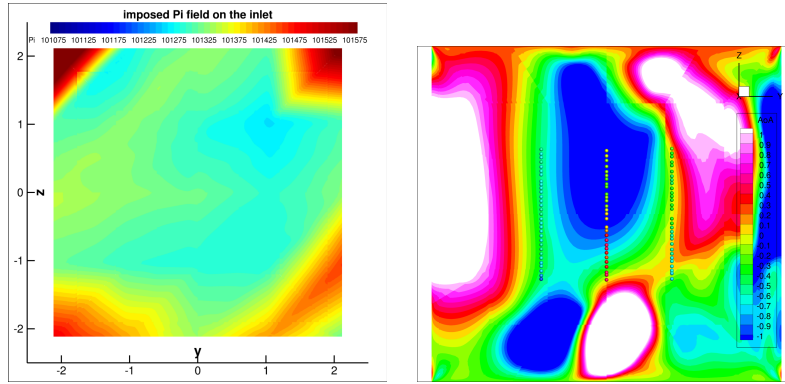


Fig. 20 Result of the optimisation of the inlet total pressure (left) to achieve the test section flow angularity (right).

applied (a 5 by 5 mesh adds deltas in total pressure on the underlying total pressure measurements, see figure 19). The optimisation method relies on a geometric approach described in [33]. It requires several simulations to achieve an inlet total pressure field (about 35). Thus, the coarse mesh has been used to tune the inlet. Parameters have been allowed to change by ± 500 Pa (about 10 times the experimental range of variation). Several results have been achieved with various inlet patterns. An example of inlet optimisation is shown in figure 20. Of course, these results are too extreme in order to get closer to the angularity map and the inlet total pressure map obtained as a result can't be used directly for future in-tunnel simulations. Nevertheless, this result is quite encouraging towards the use of a synthetic total pressure map to mimic inhomogeneities in the tunnel.

B. Assessment of the flow for transonic conditions (case B)

1. Steady flow assessment

The steady assessment for case B is presented in the following section. Due to the characteristics of this case, achieving full convergence in steady simulations proved to be challenging, however, the residual error associated with incomplete convergence remains below 1%. All steady-state simulations were conducted using a fixed outlet pressure of 80 kPa; This pressure was selected in order to match the correct shock position for both the Spalart–Allmaras QCR and the $k-\omega$ SST QCR turbulence models. In both cases, the shock foot is correctly located between the two wind-tunnel pressure taps, which define the experimentally observed shock location and are indicated by the two black lines. Figure 21 shows the different shock locations with the turbulence models with the outlet pressure of 80 kPa.

Steady simulations without the step were performed with the structured mesh to maximize convergence. Four different turbulence models were used on Fluent for this case (those being Spalart–Allmaras QCR, $k-\omega$ SST QCR,

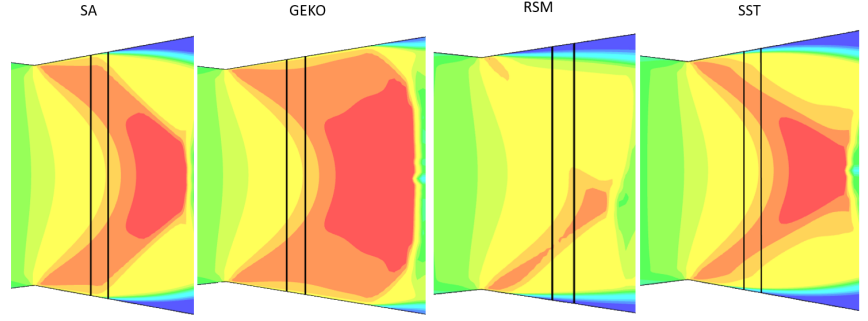


Fig. 21 Mach number distribution on the wind tunnel centerline for case 1B with and without step

GEKO—augmented QCR, and the stress- ω Full RSM). Meanwhile, steady simulations with the step were realized with the unstructured mesh. Only Spalart-Allmaras QCR on Fluent was used for this case as it is the most stable model to get converging results with those conditions.

The Mach number distribution on the wind tunnel centerline for case 1B is shown for all Fluent simulations in figure 22 and figure 23 for the test section. In the test section, the presence of step seems to stabilize the Mach number. The Mach number stays closer to the experimental value of 0.7 and it does not drop after $x = 2$ m as observed in the case without the step.

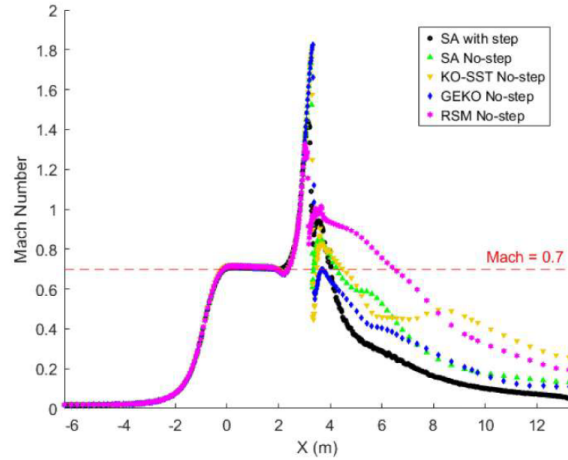


Fig. 22 Mach number distribution on the wind tunnel centerline for case B with and without step

The static pressure distribution along the upper wall for case B is shown for all Fluent simulations in figure 24. A notable difference between experimental and CFD appears in the static pressure distribution in the diffuser section of the wind tunnel. This may indicate that the CFD over-predicts boundary layer separation within the diffuser. Although the shock position seems to be in agreement with the experimental data, the level of outlet pressure at the end of the diffuser does not match the experimental data.

2. Unsteady flow assessment

In the following, the results of the CODA simulations for case B with and without step are discussed. The simulations did not converge in steady RANS mode, therefore the results shown here are based on unsteady URANS simulations. However, it has to be noted that for an outlet pressure of 80 kPa, the simulations converged to an almost steady solution. Some minor pressure oscillations were still visible, but their effect didn't extend into the test section itself. This is shown in Fig. 25, where the pressure fluctuations at $x = 2$ m, just upstream of the step location, are shown for the case with and without step. It can be seen that initially there are large pressure fluctuations, at around $t = 0.5$ s, but they are quickly damped as the fully established flow field develops. To skip the time consuming initialization of the flow field,

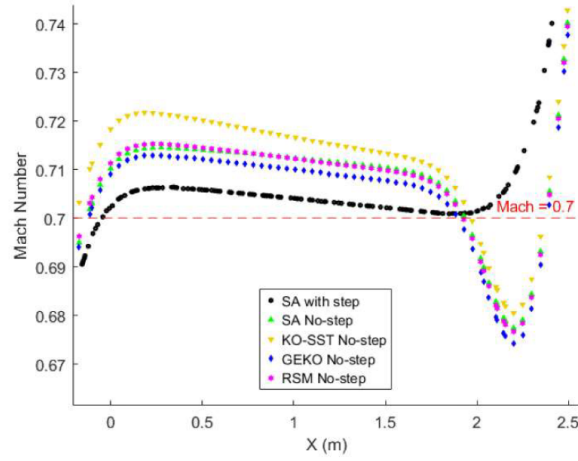


Fig. 23 Mach number distribution on the wind tunnel centerline for case B with and without step at test section

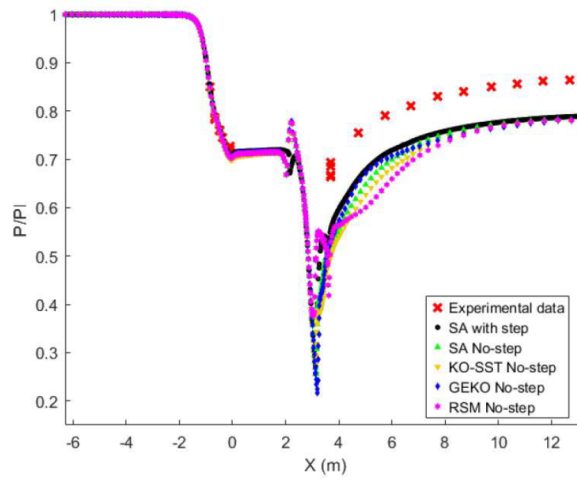


Fig. 24 Comparison of the static pressure along the upper wall for case B with and without step

the simulations with step are restarted from the simulations without step, and therefore quickly reach fully established conditions without any large scale pressure fluctuations in the test section.

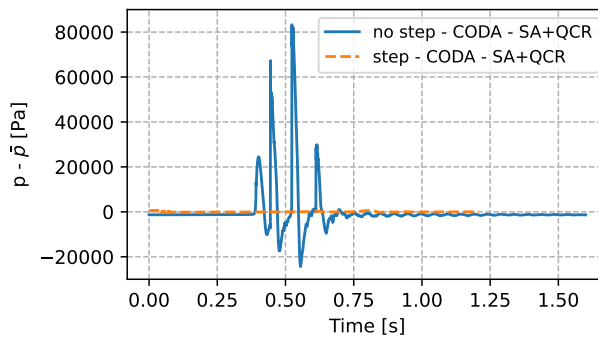


Fig. 25 Time series of the pressure fluctuations on the tunnel centerline at $x = 2$ m

The Mach number distribution on the xz - and xy -symmetry planes of the wind tunnel for the cases without and with a step are shown in figure 26 and figure 27, respectively. In both cases, the shock is located at approximately the same position, at $x \approx 3.1$ m, a bit further downstream than in the experiment. Also, both cases show large regions of separated flow that develop in the diffuser. In contrast to the steady simulations shown above, the region of separated flow is strongly asymmetric. Interestingly, the separation occurs at different walls for the case with and without step. The exact reason for this is not clear, but it seems likely that the flow topology inside the diffuser is a bi-stable system, with small changes determining where exactly the separation occurs. And once it does occur, its location seems to be fixed, as the current simulations do not show noticeable unsteadiness.

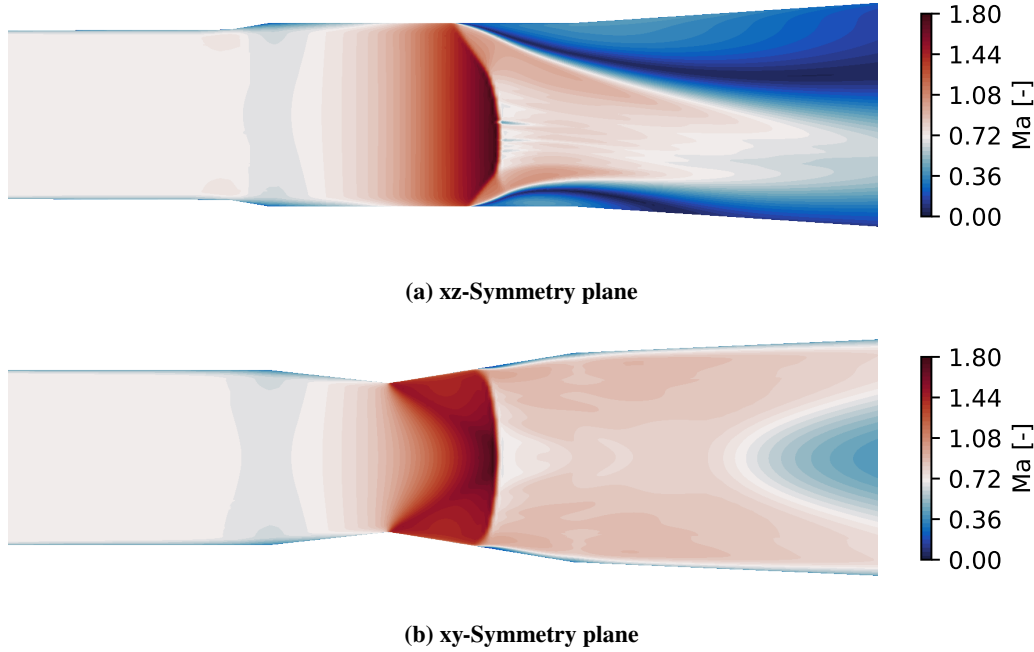


Fig. 26 Mach number distribution in the xz - and xy -symmetry planes for case B without step, CODA

To better visualize the structure of the separation, figure 28 shows the distribution of the x -normal velocity component in an x -normal plane located at $x = 4$ m. It can be seen that the extent of the main separation bubble is very similar for both cases, only at different walls. However, while the simulations with step only show one large separation bubble, the simulations with step also feature a second smaller separation at the opposite wall.

VII. Conclusion

The subsonic configuration (Case A) demonstrated robust numerical behavior and good physical accuracy within the test section, enabling reliable comparisons with experimental data and between partners. However, the diffuser remains the primary challenge for both subsonic and transonic configurations. Significant discrepancies were observed between simulations and experiments, particularly in terms of total pressure losses and flow asymmetry for transonic conditions (Case B).

The investigation confirmed that geometric simplifications, specifically the step at the end of the adaptive walls, have a significant impact on the flow. The step thickens the boundary layer and alters the central inviscid core, leading to increased total pressure losses in the diffuser, in better agreement with the experimental data for Case A.

The study also shows that inlet flow non-uniformities within the settling chamber play a key role in reproducing experimentally observed asymmetries and flow angularities. Incorporating inhomogeneous inlet conditions allows the simulations to capture the correct order of magnitude of these effects, although the exact experimental patterns are not fully recovered.

For the transonic configuration with a second throat (Case B), the simulations remain more challenging. While general flow features such as shock position and Mach number levels are reasonably captured, convergence difficulties

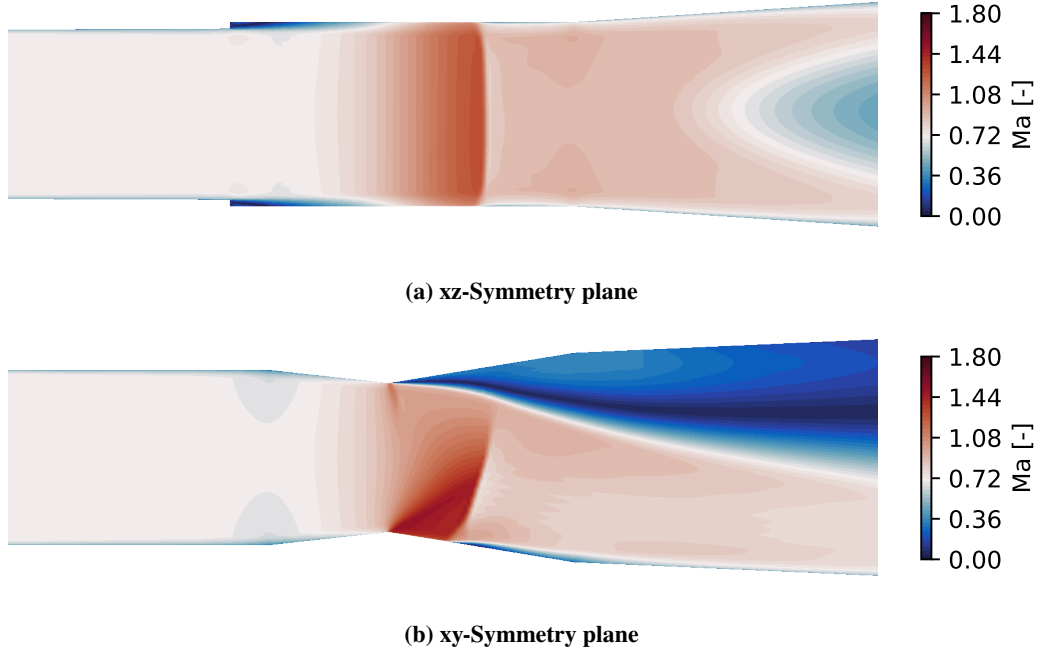


Fig. 27 Mach number distribution in the xz- and xy-symmetry planes for case B with step, CODA

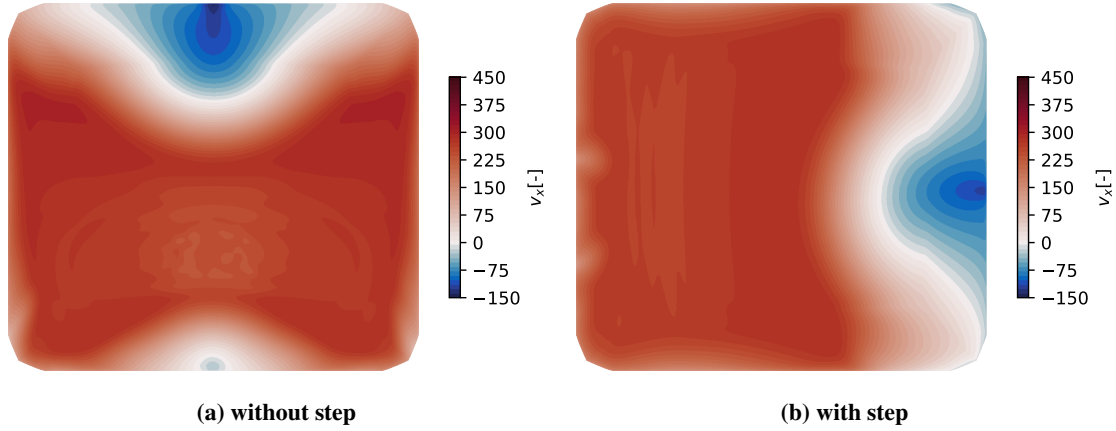


Fig. 28 Distribution of the x-normal velocity component in an x-normal plane at $x = 4$ m for case B with step, CODA

and discrepancies in the diffuser persist, and the simulations reveal complex and sometimes asymmetric separation patterns. Unsteady RANS calculations did not lead to a significantly unsteady solution for the selected outlet pressure (80 kPa); however, they consistently show asymmetric flow separation, with its location depending on the test-section geometry.

Overall, the study confirms that accurate numerical simulation of wind tunnel flows requires careful consideration of geometric details, inlet conditions, and modeling strategies. While reliable predictions can be achieved in the test section, further work is needed to improve the representation of the diffuser and complex transonic flow features.

VIII. Acknowledgment

The authors would like to acknowledge the development teams around the elsA, CODA and Roketsan softwares. The author M. Werner also gratefully acknowledges the scientific support and HPC resources provided by the German

Aerospace Center (DLR). The HPC system CARO is partially funded by "Ministry of Science and Culture of Lower Saxony" and "Federal Ministry for Economic Affairs and Climate Action".

References

- [1] Moonen, P., Blocken, B., Roels, S., and Carmeliet, J., "Numerical modeling of the flow conditions in a closed-circuit low-speed wind tunnel," *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 94, No. 10, 2006, pp. 699–723. <https://doi.org/10.1016/j.jweia.2006.02.001>, URL <https://www.sciencedirect.com/science/article/pii/S0167610506000341>.
- [2] Nayani, S., Sellers, W. L., Tinetti, A. F., Brynildsen, S. E., and Walker, E. L., "Numerical Simulation of a Complete Low-Speed Wind Tunnel Circuit," *54th AIAA Aerospace Sciences Meeting*, 2016. <https://doi.org/10.2514/6.2016-2117>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2016-2117>.
- [3] Brion, V., Dandois, J., Couliou, M., and Florentin, D., "Towards the improvement of the ONERA transonic S3Ch wind tunnel," *STO-AVT-338 - Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation II"*, 2021. <https://doi.org/hal-03255343>.
- [4] Mouton, S., "Numerical Simulation of the Flow in the ONERA F1 Wind Tunnel," *Journal of Aircraft*, Vol. 60, No. 2, 2023, pp. 355–367. <https://doi.org/10.2514/1.C036880>.
- [5] Ciobaca, V., Melber-Wilkending, S., Wichmann, G., Bergmann, A., and Küpper, A., *Experimental and numerical studies of the low speed wind tunnel DNW-NWB's open test section towards an aeroacoustic facility*, 2012. <https://doi.org/10.2514/6.2012-2174>.
- [6] Tanguy, G., and Thomas, R., "Flow field assessment of the ONERA L1 Wind Tunnel numerical simulations," *STO-AVT-338 - Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation II"*, 2021. <https://doi.org/hal-03218282>.
- [7] Szőke, M., Borgoltz, A., and Devenport, W. J., "Fluid–Structure Interaction Modeling of Flowfields Within Kevlar-Walled Wind Tunnels," *Journal of Aircraft*, Vol. 59, No. 5, 2022, pp. 1207–1220. <https://doi.org/10.2514/1.C036724>.
- [8] Elmiligui, A., Rivers, S. M., and Walker, E., "Numerical Simulation of the High-Speed Leg of the National Transonic Facility," *Journal of Aircraft*, Vol. 60, No. 3, 2023, pp. 733–746. <https://doi.org/10.2514/1.C036961>.
- [9] Frenk, W., Melber-Wilkending, S., and Gebbink, R., "Numerical Simulation of Transonic Wind Tunnel DNW-HST including Slotted Walls at Supersonic Mach Numbers," *STO-AVT-338 - Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation II"*, 2021. <https://doi.org/10.14339/STO-MP-AVT-338-01-PDF>.
- [10] Ito, Y., Murayama, M., Yokokawa, Y., Yamamoto, K., Tanaka, K., and Hirai, T., "Wind Tunnel Installation Effects on Japan Aerospace Exploration Agency's Standard Model," *Journal of Aircraft*, Vol. 59, No. 5, 2022, pp. 1281–1302. <https://doi.org/10.2514/1.C036741>.
- [11] Hantrais-Gervois, J.-L., and Piat, J.-F., "A Methodology to derive wind tunnel wall corrections with RANS simulations," *STO-AVT-284 - Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation I"*, 2018. <https://doi.org/10.14339/STO-MP-AVT-284-14-PDF>.
- [12] Mouton, S., Maunoury, M., and Alauzet, F., "Aerodynamic Simulations of the Common Research Model -High Lift Wind Tunnel Experiment to Evaluate Wall and Support Interference Effects," *STO-AVT-391 Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation III"*, Washington DC, USA, 2025.
- [13] Morrison, J. H., and Broughton, C. A., "Advanced Wind Tunnel Boundary Simulation," *Proceedings of the STO Applied Vehicle Technology (AVT) Panel Research Workshop*, Turino, Italy, 2018. <https://doi.org/10.14339/STO-MP-AVT-284>.
- [14] Morrison, J. H., Broughton, C. A., and Walker, E., "Advanced Wind Tunnel Boundary Simulation II," *Proceedings of the STO Applied Vehicle Technology (AVT) Panel Research Workshop*, online, 2021. <https://doi.org/10.14339/STO-MP-AVT-338>.
- [15] Hantrais-Gervois, J.-L., Tanguy, G., Rembaut, N., Werner, M., Katırcı, A., Nasuhbeyoğlu, B., and Di Pasquale, D., "Detailed Numerical Analysis of the S3Ch Transonic Empty Test Section," *STO-AVT-391 Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation III"*, Washington DC, USA, 2025.
- [16] Le Sant, Y., and Bouvier, F., "A new adaptive test section at ONERA Chalais-Meudon," *European Forum on Wind Tunnels and Wind Tunnel Test Techniques*, Southampton University (UK), 1992.
- [17] Rembaut, N., and Brion, V., "Experimental characterization of ONERA S3Ch transonic wind tunnel," *STO-AVT-391 Specialists' Meeting on "Advanced Wind Tunnel Boundary Simulation III"*, Washington DC, USA, 2025.

- [18] Cambier, L., Heib, S., and Plot, S., “The ONERA elsA CFD software: input from research and feedback from industry,” *Int. J. Engineering Systems Modelling and Simulation*, Vol. 14, No. 3, 2013. <https://doi.org/10.1051/meca/2013056>.
- [19] Spalart, P. R., “Strategies for Turbulence Modelling and Simulations,” *International Journal of Heat and Fluid Flow*, Vol. 21, 2000, pp. 252–263.
- [20] Stefanin Volpiani, P., Chapelier, J.-B., Schwöppe, A., Jägersküpper, J., and Champagneux, S., “Aircraft Simulations Using the New CFD Software from ONERA, DLR, and Airbus,” *Journal of Aircraft*, Vol. 61, No. 3, 2024, pp. 857–869. <https://doi.org/10.2514/1.C037506>.
- [21] Allmaras, S. R., Johnson, F. T., and Spalart, P. R., “Modifications and Clarifications for the Implementation of the Spalart-Allmaras Turbulence Model,” *Computational Fluid Dynamics*, 2012, p. 11.
- [22] Anderson, W. K., and Bonhaus, D. L., “An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids,” *Computers and Fluids*, Vol. 23, No. 1, 1994, pp. 1–22.
- [23] Barth, T. J., and Jespersen, D. C., “The Design and Application of Upwind Schemes on Unstructured Meshes,” *27th Aerospace Sciences Meeting*, 1989. <https://doi.org/10.2514/6.1989-366>.
- [24] Frink, N. T., “Tetrahedral Unstructured Navier-Stokes Method for Turbulent Flows,” *AIAA Journal*, Vol. 36, No. 11, 1998, p. 1975–1982. <https://doi.org/10.2514/2.324>.
- [25] Frink, N. T., Pirzadeh, S. Z., Parikh, P. C., Pandya, M. J., and Bhat, M., “The NASA Tetrahedral Unstructured Software System (TetrUSS),” *The Aeronautical Journal*, Vol. 104, No. 1040, 2000, p. 491–499. <https://doi.org/10.2514/1.C036429>.
- [26] Wick, A. T., Hooker, J. R., and Zeune, C. H., “Integrated Aerodynamic Benefits of Distributed Propulsion,” *53rd AIAA Aerospace Sciences Meeting*, 2015. <https://doi.org/10.2514/6.2015-1500>.
- [27] Hooker, J. R., Wick, A., Zeune, C., and Agelastos, A., “Over Wing Nacelle Installations for Improved Energy Efficiency,” *53rd AIAA Aerospace Sciences Meeting*, 2013. <https://doi.org/10.2514/6.2013-2920>.
- [28] Pandya, M. J., Frink, N. T., Ding, E., and Parlette, E., “Toward Verification of USM3D Extensions for Mixed Element Grids,” *31st AIAA Applied Aerodynamics Conference*, 2013. <https://doi.org/10.2514/6.2013-2541>.
- [29] Elmiligui, A., Pandya, M. J., Carter, D. B., M. B., and Nayani, S. N., “USM3D Simulations for the Third AIAA Sonic Boom Prediction Workshop,” *Journal of Aircraft*, Vol. 59, No. 3, 2022, pp. 647–659. <https://doi.org/10.2514/1.C036429>.
- [30] Elmiligui, A., Pandya, M. J., Bozeman, M. D., Hunter, C., Diskin, B., Hooker, R., and Wick, A., “USM3D-ME Contributions to the 5th AIAA High-Lift Prediction Workshop,” *AIAA SCITECH 2025 Forum*, 2025. <https://doi.org/10.2514/6.2025-0274>.
- [31] Pandya, M. J., Diskin, B., Thomas, J. L., and Frink, N. T., “Assessment of USM3D Hierarchical Adaptive Nonlinear Method Preconditioners for Three-Dimensional Cases,” *AIAA Journal*, Vol. 55, No. 10, 2017, p. 3409–3424. <https://doi.org/10.2514/1.J055823>.
- [32] Pandya, M. J., Jespersen, D. C., Diskin, B., Thomas, J. L., and Frink, N. T., “Efficiency of Mixed-Element USM3D for Benchmark Three-Dimensional Flows,” *AIAA Journal*, Vol. 59, No. 8, 2017, pp. 2820–3011. <https://doi.org/10.2514/1.J059720>.
- [33] Delrieux, Y., Bailly, J., and Bailly, D., “Improvement of Experiment/Computation BVI Noise Comparisons by Use of a Geometrical Method Applied to Data Assimilation,” *74th Annual Vertical Flight Society Forum and Technology Display 2018 (FORUM 74)*, Phoenix, Arizona, USA, 2018.



DSPACE

<https://dspace.org/>

Numerical simulation of flows in the ONERA S3Ch transonic wind tunnel

Hantrais-Gervois, Jean-Luc; Tanguy, Geoffrey; Rembaut, Nicolas; Werner, Michael; Di Pasquale, Davide ...

2026-06-08

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2026-4458>

Hantrais-Gervois J-L, Tanguy G, Rembaut N, et al., (2026) Numerical simulation of flows in the ONERA S3Ch transonic wind tunnel. In: AIAA AVIATION 2026 Forum, 8-12 June 2026, San Diego, CA, Article number 2026-4458