California State University, Long Beach

Department of Mechanical and Aerospace Engineering

Fall 2025

Preliminary Design of a Subsonic Windtunnel

Author: Dararath Run

Abstract

This report presents the preliminary aerodynamic design of a subsonic wind tunnel intended to provide a steady, reasonably uniform, moderate-speed flow for experiments on lift, drag, and basic external aerodynamics. The facility is constrained to a 1 m x 1 m square test section operating at a design Mach number of 0.6 and must fit within a 20 m x 7 m x 7 m laboratory envelope. The main flow path, consisting of a 3 m test section, a contraction and diffuser, is sized using quasi-one-dimensional isentropic relations combined with geometric constraints and established wind-tunnel design guidelines. A contraction ratio of 8 and an optimized diffuser exit area of approximately 3.0 m^2 are selected through a trade study that balances fan power, pressure recovery, and overall tunnel size. A smooth cubic half-height function is implemented in Python to generate the tunnel wall geometry and to compute centerline distributions of Mach number, velocity, pressure, temperature, and density. The resulting test-section Reynolds number, $Re\ =\ 1.36\ x10^7$, is appropriate for subsonic airfoil and wing testing. Two-dimensional CFD simulations (Euler and Navier-Stokes) performed in Ansys show very good agreement with the quasi-1D predictions, confirming that the chosen geometry delivers the desired flow quality while satisfying the principal design constraints.

1. Introduction

Wind tunnels are one of the most important experimental tools in aerodynamics and aircraft designs. Wind tunnels are able to provide repeatable, well-controlled flow conditions and direct measurements of aerodynamic forces, moments, and pressure distributions. Computational fluid dynamics has been able to provide similar experimental data but one thing that sets these two apart is that windtunnel exist in the real world and could provide real world data which a program may not. So wind tunnels are very valuable for validating computational models, testing scaled prototype of components and exploring flow phenomena such as separation, stall, and unsteady wake dynamics which may be challenging to simulate accurately.

The main design objective is to provide a steady reasonably uniform, moderate-speed stream for experiments on lift, drag, and basic external aerodynamics (airfoils, simple wings, bluff bodies, etc.). The scope of the work is limited to the aerodynamic and geometric design of the main flow path. Specifically, this report will:

- Design the contraction, and diffuser profiles to achieve acceptable flow uniformity and reasonably low total pressure losses.
- Estimate the centerline flow properties along the tunnel using quasi-one-dimensional compressible flow relations and an assumed cross-sectional geometry.
- Provide a 2-D preliminary layout suitable for subsequent structural, mechanical, and detailed CFD design.
- Verify results inside a Computational Fluid Dynamics software to compare theoretical data calculated vs simulated data.

Out of scope of this work consist of :

- 3-D modeling the wind tunnel and running full simulations on a 3-D scale.
- Fully constructing the wind tunnels with necessary components consisted inside.

The main constraint of this designs are:

- Have a 1m height x 1m depth cross section
- A minimum of 3m long with a design mach number of 0.6 for the test section
- Tunnel driven by a fan to do work on the fluid isentropically

The key constraints for this designs involves:

- Limited laboratory space: the complete wind tunnel, including contraction, test section, diffuser, fan unit, and supports, must fit inside a 20 m × 7 m × 7 m volume.

The approach to this design begins by using isentropic flow relations and tables from a standard textbook appendix to relate area, Mach number, and area properties along the tunnel. Existing design guidelines, including a recommended contraction ratio, are adopted as a starting point to size the contraction and test section. Reasonable engineering assumptions are then made about losses, flow uniformity, and operating conditions in order to generate preliminary estimates of velocity, pressure, temperature, and density. These preliminary results will guide the choice of tunnel geometry and will later be refined as the design is checked against the stated objectives and constraints.

2. Design Method

The main design objective is to deliver a steady, reasonably uniform, moderate-speed airflow suitable for experiments on lift, drag, and fundamental external aerodynamics, including tests on airfoils, simple wings, and bluff bodies. This design begins by specifying the basic flow constants such as the ratio of specific heats $\gamma = 1.4$, the gas constant $R = 287$ J/(kg·K), a room (stagnation) temperature of 21°C (294.15 K), and an ambient pressure of 1 atm (101.325 kPa). For this application, an ideal choice is a subsonic flow with Mach number less than 1.

2.1 Preliminary dimension selection

A schematic of the wind tunnel was provided as a starting point for the design.

This sketch illustrates the fan location, test section, and varying cross-sectional areas used to guide the initial geometry.

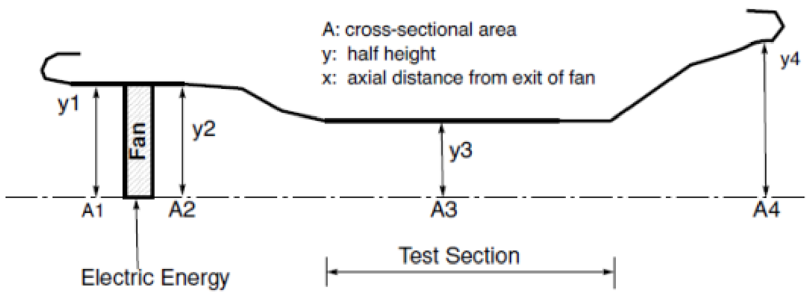

Figure 1. Initial design sketch of the wind tunnel

Since stations A_1 and A_2 lie in the same upstream section, they are assumed to have the same cross-sectional area. The test section is specified to operate at a Mach number of M_3 = 0.6; therefore, the Mach number in the upstream must be less than 0.6, requiring a larger cross-sectional area there. The contraction ratio is the ratio of the inlet cross-sectional area to the test-section area, and it controls how strongly the flow is accelerated and how effectively large-scale non-uniformities are smoothed out before entering the test section. A design study by Bell and Mehta on small low-speed wind tunnels indicates that contraction ratios in the range of about 6 to 10 provide a good compromise between flow quality, space, and cost [1]. Based on this guidance, a contraction ratio of 8 is adopted for the present tunnel as a representative mid-range value within their recommended interval. Starting from the specified test-section Mach number M_3 = 0.6, the isentropic flow tables give an area ratio of A_3/A* = 1.1880. With the test-section area fixed at A_3 = 1.0 m^2, the corresponding critical area is obtained from A* = A_3/ (A_3/A*) = 1/ 1.1880. The contraction ratio is subsequently calculated as CR = A_1/A_3 for this design and using A_3 = 1 m^2 gives a target inlet area of 8 m^2 which filters the result to Mach M_1

= 0.060 giving A_1/A* = 9.6660 with A_1 = 8.136364. Similarly, an exit Mach number of M_4 = 0.32 is selected, which through the same isentropic area relation gives $A_4 \approx 1.6\ m^2$; this provides a moderate diffuser area ratio relative to the test section, allowing useful static-pressure recovery while keeping wall expansion angles small enough to avoid separation and to satisfy the overall height constraint.

Next is to calculate the length of the converging segments and diverging segments. This was done using geometric relation shown in the figure below:

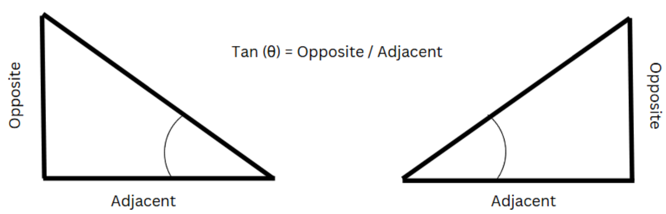

Figure 2. Tangent relationship

With the geometric relation shown this gives an equation of:

$L_c = \frac{\Delta y_c}{tan(\theta_c)}$ and $L_d = \frac{\Delta y_d}{tan(\theta_d)}$ where,

L_c is the contraction length with θ_c being contraction angle and Δy_c is the change in half height from ($y_1 - y_3$). Similarly L_D is the diffuser length with θ_d being defuser angle and Δy_d is the change in half height from ($y_4 - y_3$). Using the half angle assumptions for diffuser and contraction of wind tunnels for similar applications, a contraction angle of 15° and diffuser angle of 3° was selected. This gives the preliminary lengths of L_c = 3.41m and L_d = 2.53m.

2.2 Analysis and Performance Equations

The following equations will be used consistently throughout the iterative process of the design and trade studies performed in the results and discussion. The flow in the wind tunnel is modelled as a steady, one-dimensional, adiabatic, and with negligible friction. Air is assumed to be a perfect gas with constant properties of $\gamma = 1.4$, R = 287 J/(kg·K) and c_p = 1005 J/(kg·K).

Under these assumptions the flow between stations is isentropic everywhere except across the fan, which allows the use of standard isentropic relations to relate Mach number, static properties and stagnation properties. For any station with Mach number M, the stagnation static pressure is

$$\frac{p_0}{p} = (1 + \frac{\gamma - 1}{2} M^2)^{\frac{\gamma}{\gamma - 1}}, \qquad (1)$$

Which is used to compute the stagnation pressure p_0 at a given static pressure p and Mach number. The corresponding stagnation temperature ratio is can also be obtained from the isentropic relation

$$\frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2 = (\frac{p_0}{p})^{\frac{\gamma - 1}{1}}, \qquad (2)$$

and is applied to determine the stagnation temperature downstream of the diffuser exit when the exit Mach number is known. These expressions rely directly on the assumptions of adiabatic, reversible (lossless) flow and constant γ .

The mass flow rate through any station is obtained from the quasi-one-dimensional continuity equation, combined with the ideal-gas law and the isentropic relations. For a perfect gas with constant γ and R, the result can be written as

$$\dot{m} = \Gamma \frac{p_0}{\sqrt{R T_0}} q(M) A, \qquad (3)$$

where ṁ denotes the mass flow rate in kilograms per second, Γ is $\sqrt{\gamma}$, p_0 is the local stagnation pressure in pascals, T_0 is the local stagnation temperature in kelvins, R is the specific gas constant for air A is the local cross-sectional area in square metres, M is the local Mach number, and q(M) is the non-dimensional mass-flow function that depends only on Mach number. The equation below defines q(M):

$$q(M) = M(1 + \frac{\gamma - 1}{2} M^2)^{-\frac{\gamma + 1}{2(\gamma - 1)}}. \quad (4)$$

Across the fan, the aerodynamic power input is computed from the change in stagnation temperature using

$$P_{aero} = \dot{m}\, c_p\, \Delta T_0 , \quad (5)$$

where ṁ is the mass flow rate, c_p is the constant-pressure specific heat, and $\Delta T_0 = T_{0,2} - T_{0,1}$ is the fan-induced rise in stagnation temperature. This expression follows from the steady-flow energy equation for an adiabatic device with negligible changes in potential energy and with all shaft work appearing as an increase in stagnation enthalpy of the flow.

In addition, the static-to-ambient pressure ratio at the diffuser exit is obtained from the inverse isentropic pressure relation

$$\frac{p_4}{p_a} = \frac{p_4}{p_{01}} = [\frac{1}{1 + \frac{\gamma - 1}{2} M_4^{\ 2}}]^{\frac{\gamma}{\gamma - 1}}, \quad (6)$$

where M_4 is the exit Mach number, p_4 is the static pressure at station 4, and $p_a \approx p_{01}$ is the ambient (upstream stagnation) pressure. This expression is derived under the same assumptions of steady, adiabatic, one-dimensional, lossless flow of a perfect gas, and it is used to evaluate how closely the exit static pressure matches the surrounding atmospheric pressure. A ratio p_4 / p_a close to unity indicates that the tunnel exhausts smoothly to the room with little pressure mismatch, while lower values indicate a larger static-pressure drop, helping to quantify the amount of kinetic energy remaining in the jet and the effectiveness of pressure recovery in the diffuser.

Finally the tunnel wall geometry is defined by a smooth half-height function y(x) chosen to produce the required cross-sectional changes while avoiding excessive wall gradients in the converging and diverging sections and maintaining an approximately one-dimensional, uniform flow in the test section. The contraction and diffuser are each described by a cubic polynomial in the normalized axial coordinate, joined to a constant half-height in the test section so that both the height and its slope are continuous at the junctions. Neglecting the round-to-rectangular transition duct at the fan exit and focusing only on the contraction, test section, and diffuser, the final piecewise definition of the half-height is

$$y(x) = \begin{cases} \frac{H_1}{2} + \left(\frac{H_4}{2} - \frac{H_1}{2}\right)\left[3\left(\frac{x}{L_c}\right)^2 - 2\left(\frac{x}{L_c}\right)^3\right], & 0 \le x \le L_c, \\ \frac{H_4}{2}, & L_c \le x \le L_c + L_t, \\ \frac{H_4}{2} + \left(\frac{H_5}{2} - \frac{H_4}{2}\right)\left[3\left(\frac{x-(L_c+L_t)}{L_d}\right)^2 - 2\left(\frac{x-(L_c+L_t)}{L_d}\right)^3\right], & L_c + L_t \le x \le L_c + L_t + L_d. \end{cases} \tag{7}$$

where H_1, H_4, and H_5 are the full tunnel heights at the inlet, test section, and diffuser exit respectively, and L_c, L_t, and L_d are the lengths of the contraction, test section, and diffuser.

3. Results and Discussion

The preliminary geometric parameters obtained from the design methods are $A_1 = A_2 \approx 8\ m^2$, $A_3 = 1\ m^2$, and $A_4 = 1.6\ m^2$. The corresponding section lengths are L_c= 3.41 m for the contraction, L_t = 3.0 m for the test section (set by the constraint), and L_d= 2.53 m for the diffuser. With a test-section height of 1 m, a nominal contraction wall angle of 15°, and a diffuser wall angle of

3°, these values define the preliminary tunnel geometry. The length of the duct was an arbitrary estimation and is not the true value for this specific wind tunnel; it was made specifically to assume where the fan would be located. The resulting configuration is shown in the following schematic.

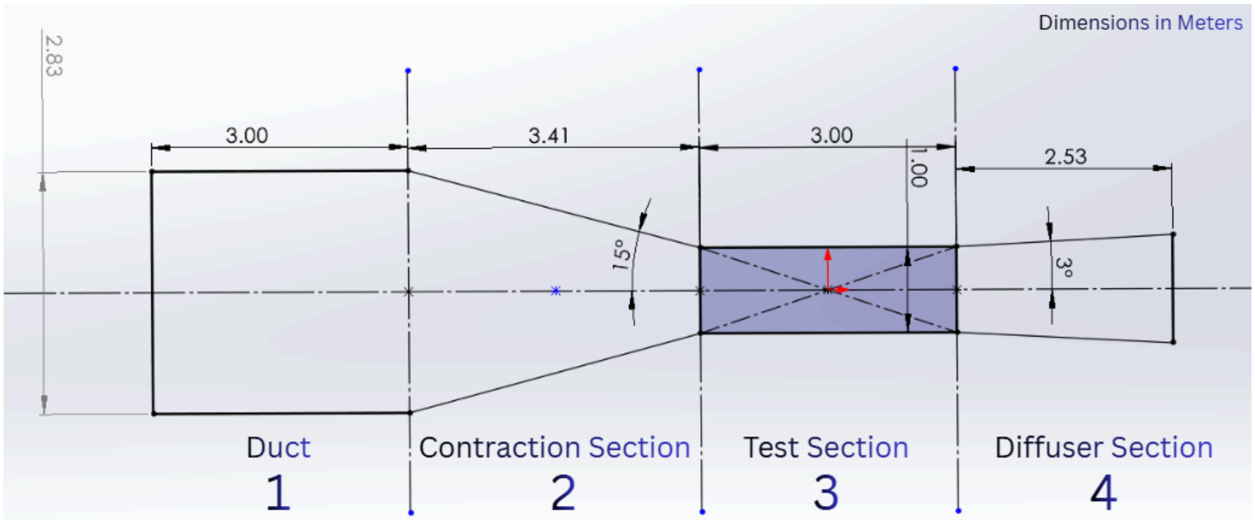

Figure 3. Preliminary schematic of the wind tunnel

Using the schematic and dimensions above, the wind-tunnel pressure ratio and aerodynamic power requirement were calculated from the governing formulas based on the preliminary design parameters. These calculated values were then used to guide the initial selection of a suitable fan from external supplier data. Using equation 1 on stagnation relation, the pressure ratio p_{02} / p_{01} was calculated to be 1.0735 with a stagnation temperature ratio $T_{02} / T_{01} = 1.02047$ from equation 2. The stagnation temperature T_{02} = 300.17K with the assumption of $T_{01} = T_a$. The mass flow rate was calculated to be 199 kg/s from equation 3 and the power required was calculated to be 1.2MW from equation 5. Using a technical catalog from available fans outside a motor, AXR500MN2 was selected for this current design. It is capable of providing a shaft power of 1250kW or 1.25MW. It has an efficiency of 96.9% at full load giving its electrical power usage to 2760V [3]. The initial design features a diffuser that is shorter than the contraction, which can promote adverse pressure gradients and increase the risk of boundary-layer separation. In addition, the required fan power for this configuration is relatively high. These two issues indicate that the original choice of $A_4$and diffuser length is not optimal in terms of both flow quality and energy usage. In light of this, a trade study was performed to select a revised A_4 and diffuser geometry that reduces power demand while providing a longer, more gradual diffusion to improve pressure recovery and minimize separation risk. The trade study in the table below was carried out using isentropic flow data taken from the textbook Fundamentals of Aerodynamics (6th ed., Anderson). This ensured that the Mach-area relations used in the analysis were accurate and consistent for the range. All calculations were carried out in Python using a program developed specifically for this wind-tunnel design. The governing equations implemented in this code are the same isentropic and performance relations presented earlier in the analysis and performance equations section.

Table 1. Trade study among different diffuser sizing to performance

A4 [m²]	M4	V4 [m/s]	p02/p01	P_aero [MW]	L_diff [m]	p4/pa
24.360	0.020	6.876	1.000	0.005	37.548	1.000
12.189	0.040	13.751	1.001	0.019	23.768	0.999
8.136	0.060	20.627	1.003	0.043	17.673	0.997
6.113	0.080	27.503	1.004	0.076	14.048	0.996
4.901	0.100	34.379	1.007	0.119	11.580	0.993
4.094	0.120	41.254	1.010	0.172	9.764	0.990
3.520	0.140	48.130	1.014	0.236	8.360	0.986
3.092	0.160	55.006	1.018	0.309	7.235	0.982
2.759	0.180	61.882	1.023	0.392	6.307	0.978
2.495	0.200	68.757	1.028	0.487	5.529	0.972
2.279	0.220	75.633	1.034	0.592	4.864	0.967
2.101	0.240	82.509	1.041	0.708	4.288	0.961
1.950	0.260	89.385	1.048	0.836	3.783	0.954
1.823	0.280	96.260	1.056	0.976	3.342	0.947
1.713	0.300	103.136	1.064	1.128	2.946	0.939
1.618	0.320	110.012	1.074	1.292	2.595	0.932
1.535	0.340	116.888	1.083	1.470	2.278	0.923
1.461	0.360	123.763	1.094	1.662	1.992	0.914
1.396	0.380	130.639	1.105	1.868	1.734	0.905
1.338	0.400	137.515	1.117	2.088	1.497	0.896
1.287	0.420	144.391	1.129	2.325	1.283	0.886
1.241	0.440	151.266	1.142	2.577	1.087	0.875
1.199	0.460	158.142	1.156	2.846	0.908	0.865
1.162	0.480	165.018	1.171	3.132	0.742	0.854
1.128	0.500	171.893	1.186	3.437	0.592	0.843
1.097	0.520	178.769	1.202	3.761	0.451	0.832
1.069	0.540	185.645	1.219	4.105	0.324	0.820
1.044	0.560	192.521	1.237	4.470	0.207	0.808
1.021	0.580	199.396	1.256	4.857	0.100	0.796
1.000	0.600	206.272	1.276	5.267	0.000	0.784

As A_4 increases, the diffuser exit Mach number and velocity decrease, so the required aerodynamic power drops and the pressure ratio p_4/p_a moves closer to 1, indicating good pressure recovery. This is advantageous because more of the flow's kinetic energy is converted back into static pressure and the fan does not have to overcome a large pressure mismatch at the tunnel exit. A larger A_4 also means weaker, less energetic exhaust jets and a lower risk of flow separation if the diffuser angle is kept modest. However, achieving a large A_4 with small wall angles requires a much longer and physically larger diffuser, which increases overall tunnel size, cost, and structural complexity. A smaller A_4 gives a compact diffuser but leaves the flow exiting at higher Mach number and lower p_4/p_a, wasting more energy in the jet and demanding higher fan power. For this design, it is therefore desirable to choose A_4 so that p_4/p_a is as close to 1 as practical while still satisfying the geometric and cost constraints. An A_4 of approximately 3.0 m^2 was selected from this trade study, giving a diffuser length of about 7.24 m with an exit Mach number $M_4 \approx 0.16$ and an exit velocity of about 55 m/s. This case was chosen because it offers a good compromise between performance and size: p_4/p_a remains close to unity, indicating good pressure recovery and a reasonable fan power requirement, while the diffuser length is still acceptable within the overall space constraint and keeps wall angles small enough to limit separation risk. The schematic below shows the updated tunnel layout, incorporating all revised dimensions and design parameters.

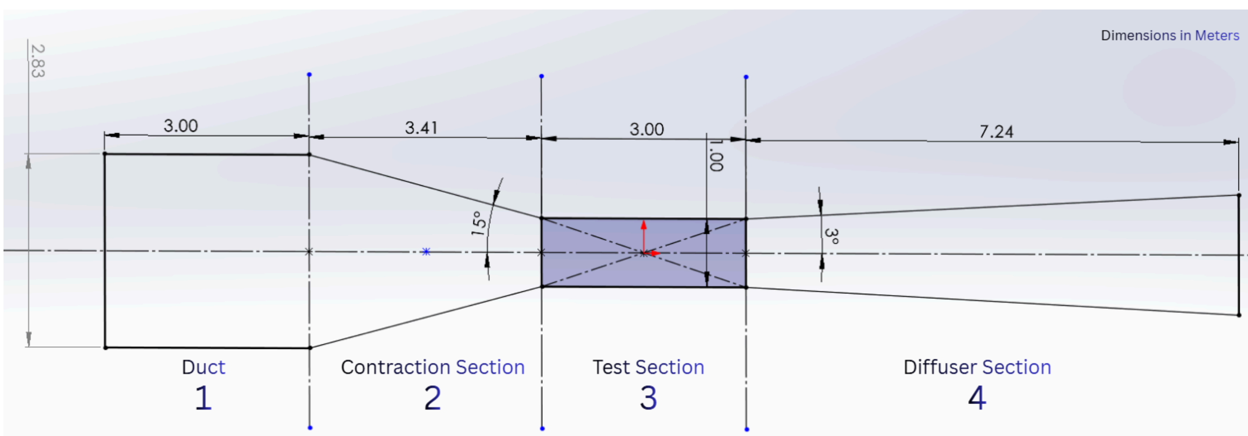

Figure 4. Updated wind tunnel schematic

Excluding the duct, the wind tunnel has a total length of 13.65 m with a maximum height of 2.83 m; depending on the (arbitrary) duct length in this schematic, the total room required for the installation is therefore equivalent to a laboratory at least 20 m in length and about 5 m in both width and height to provide sufficient space and clearance. In addition to geometric space, an important consideration for the test section is the Reynolds number, Re = ρVL / μ, which measures the ratio of inertial to viscous forces based on a characteristic length L. Using the design test-section conditions and a representative model length, the resulting Reynolds number is approximately $Re\ =\ 1.36\ x10^{7}$, a value typical of tests on airfoils and moderate-size aircraft wings in subsonic facilities. This Reynolds-number level is appropriate for the intended subsonic wind-tunnel experiments because it produces predominantly turbulent boundary layers and realistic lift, drag, and separation behaviour for external-aerodynamics studies. With the new dimensions given, the fan selected for this is an AXR 400MM10 with power of 355kW at an efficiency of 94.5% at full load and electrical power consumption of 503V.

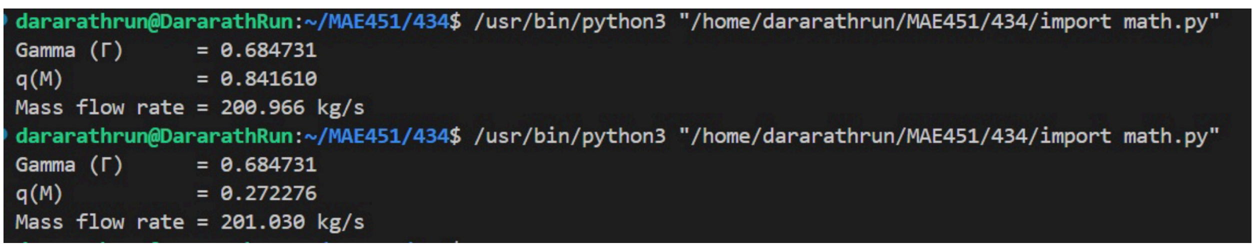

Figure 5. Automatic calculation for mass flow rate

To check the internal consistency of the quasi-one-dimensional model, the mass flow rate ṁ was evaluated both in the test section and at the tunnel exit with the appropriate stagnation conditions, area, and Mach number at each station. For the test section, this gave ṁ = 200.97 kg/s, while for the exit the result was ṁ = 201.03 kg/s. These values differ by only about 0.064 kg/s, or roughly 0.03%, indicating that the assumed flow field satisfies mass conservation to a very good approximation and that the chosen areas and Mach numbers at the two stations are mutually consistent.

Finally a Python program was developed that implements Eq. (7) to generate a smooth half-height function y(x), producing the desired cross-sectional variation while avoiding excessive wall gradients in both the converging contraction and the diverging diffuser. Using this function, the final wind-tunnel geometry is shown in figure 6. The top and bottom walls smoothly contract from the large inlet to the 1 m-high test section, remain parallel through the test section to maintain nearly one-dimensional flow, and then expand gently in the diffuser to recover static pressure without inducing separation.

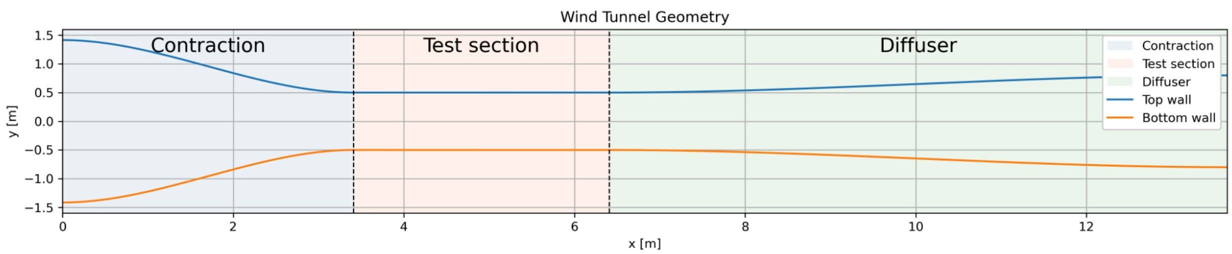

Figure 6. Final schematic of wind tunnel

4. Assessment

Using the selected wall shape y(x), the chosen areas A_1, A_3, and A_4, and the fan-induced stagnation conditions, the quasi-one-dimensional isentropic model was applied to compute the centerline distributions of Mach number, velocity, static pressure, temperature, and density along the tunnel length. These profiles provide a direct assessment of how well the final geometry and operating point satisfy the design objectives and constraints. Figure 7 presents the Mach number distribution as a function of axial position and serves as a reference for interpreting the behaviour of the other flow variables along the centerline.

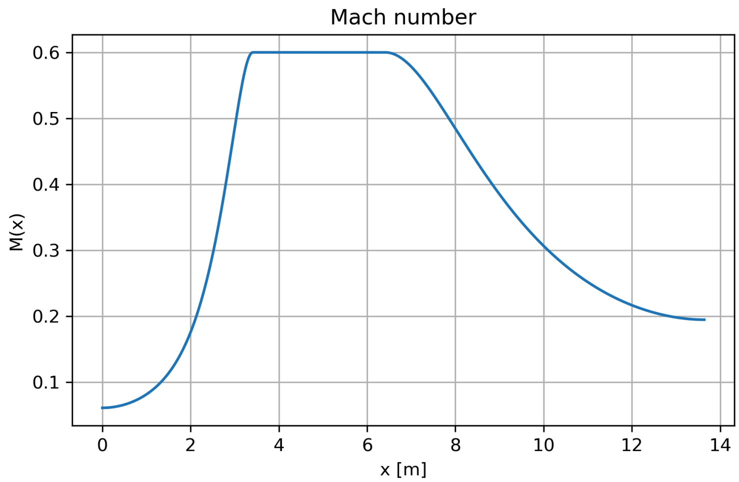

Figure 7. Mach number graph vs centerline

The Mach number profile in Figure 7 confirms that the flow accelerates smoothly from a very low value at the inlet to the design value of M = 0.6 through the contraction, remains essentially constant at this target level over the entire 3 m test section, and then decelerates gradually in the diffuser to a moderate subsonic exit Mach number of 0.160. This behaviour indicates that the contraction and diffuser provide the required area variation without introducing abrupt changes in Mach number and that the test section operates at a nearly uniform, well defined flow speed. The next graph, shown in Figure 8, presents the corresponding velocity distribution along the centerline of the wind tunnel and further illustrates the confinement of the high-speed region to the test section.

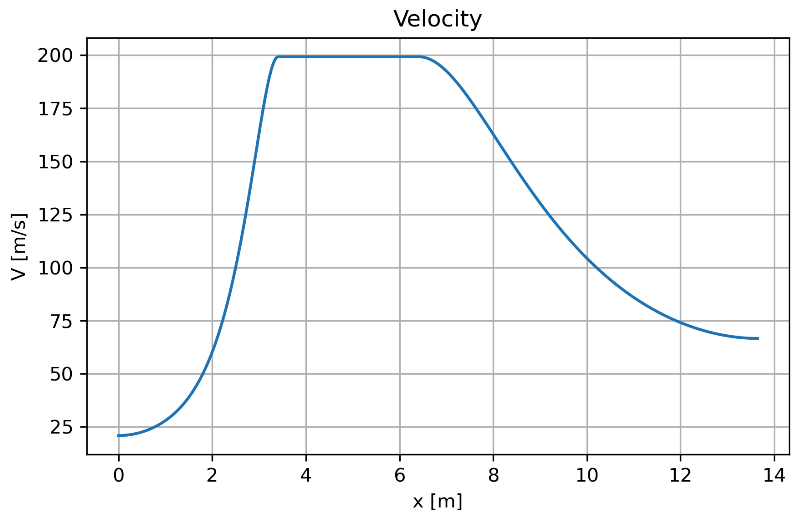

Figure 8. Velocity graph vs centerline graph

The corresponding velocity curve peaks at approximately 200 m/s in the test section, while the inlet and exit velocities remain much lower, so the region of highest dynamic pressure and aerodynamic loading is deliberately confined to the measurement section where models are installed. This distribution also shows a smooth deceleration in the diffuser, which is desirable for pressure recovery and for minimizing the risk of flow separation. The associated variation in static pressure along the tunnel, shown in Figure 9, reflects this behaviour and provides a complementary view of how the contraction and diffuser influence the overall pressure field.

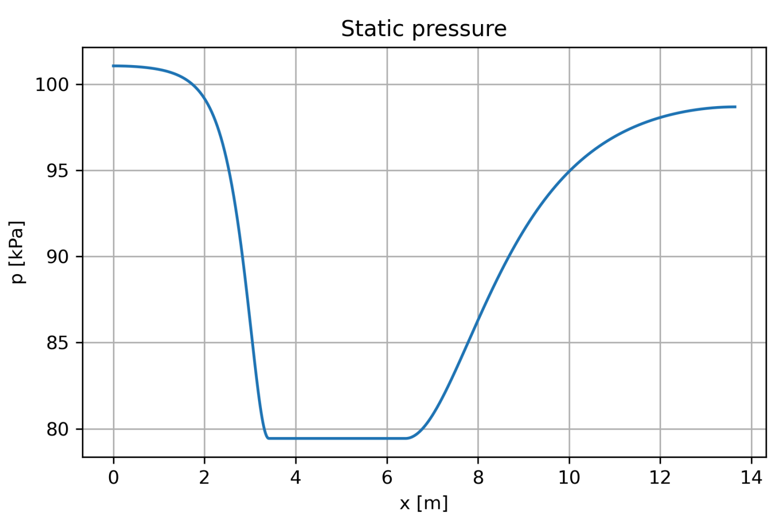

Figure 9. Static pressure vs centerline graph

The static-pressure distribution shows that the flow starts close to ambient pressure at the inlet, then drops as the flow accelerates through the contraction and into the test section, reaching a minimum of roughly 80 kPa. In the diffuser, the pressure recovers smoothly back toward the ambient level by the exit, indicating effective pressure recovery and suggesting that the diffuser expansion is gentle enough to avoid large adverse pressure gradients or incipient separation. These pressure changes are directly tied to the temperature variations along the tunnel, which are shown in Figure 10.

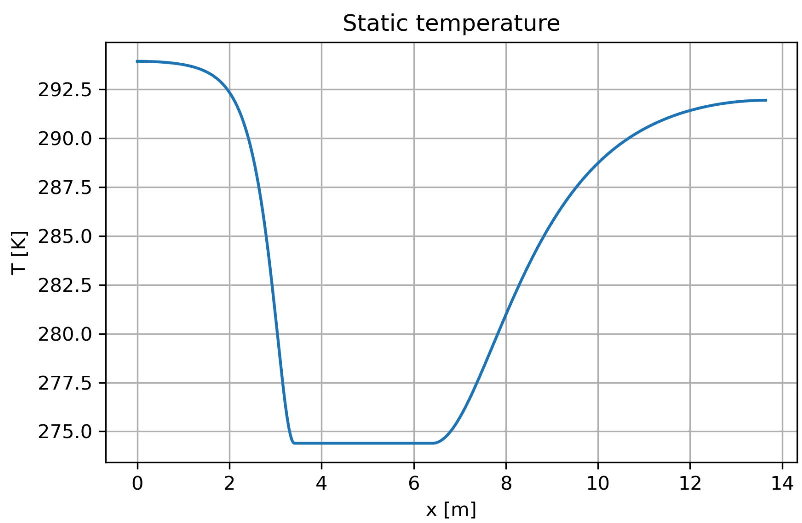

Figure 10. Static temperature vs centerline graph

The static-temperature curve follows the expected isentropic trend: the temperature falls from the inlet value of about 294 K down to a minimum of roughly 274 K in the high speed test section and then rises again through the diffuser as the flow decelerates. The changes are moderate, consistent with the subsonic design Mach number of 0.6, and confirm that the flow remains within the low-speed compressible regime. Since density depends on both pressure and temperature, the combined effect of Figures 9 and 10 is captured in the density distribution presented in Figure 11.

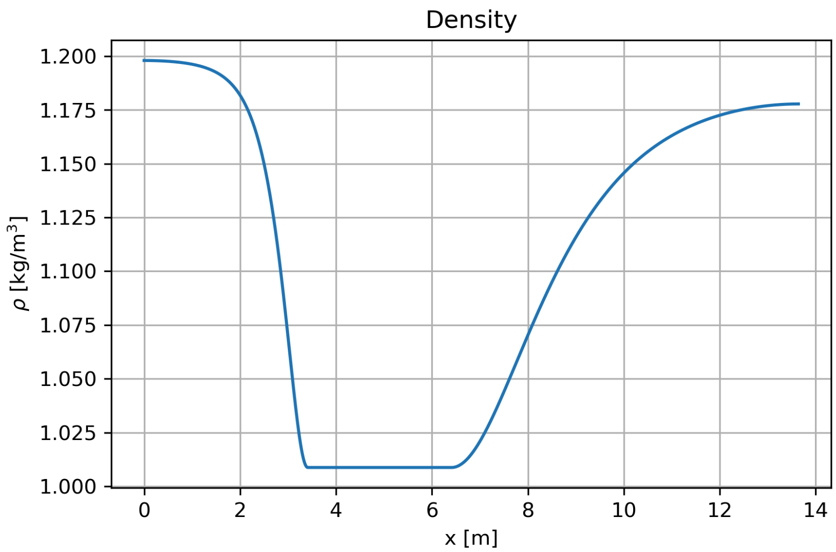

Figure 11. Density vs centerline graph

The density distribution combines the effects of the pressure drop and temperature change: density decreases from about 1.20 kg/m^3 at the inlet to roughly 1.01 kg/m^3 in the test section and then increases again toward the exit as the pressure recovers and the temperature rises. This pattern shows that the largest density changes are confined to the contraction and test section, where the flow is fastest, while the exit conditions return close to ambient. Using the test-section density together with the local velocity and a representative model length gives a Reynolds number of approximately $Re\ =\ 1.36\ x10^7$, which is typical for subsonic testing of airfoils and moderate-span wings and is therefore appropriate for the intended experiments.

The assessment highlights a few weaknesses and possible improvements. The low static pressure in the test section implies relatively high structural loads on the tunnel walls and windows, which would need to be considered in a detailed mechanical design. The analysis assumes perfectly isentropic, one-dimensional flow and neglects boundary-layer growth, secondary flows, and fan losses; in practice these effects will reduce the actual Mach number and increase the required fan power compared with the ideal predictions. The design is also somewhat sensitive to the chosen test-section Mach number and contraction ratio: small changes in M_3 or CR would alter the required areas and significantly affect velocity, pressure, and fan power. A natural next step would be a parametric study that varies M_3, A_4, and the diffuser length/angle to quantify this sensitivity and explore trade-offs between flow quality, pressure recovery, and tunnel size, followed by CFD or experimental validation of the 1D predictions.

4.1 Comparison to Computational Simulation

To further assess the design, a two-dimensional CFD model of the contraction to test section to diffuser channel was constructed in Ansys. The geometry from the quasi-one-dimensional analysis was generated through a special python program developed and imported into Ansys space claim to develop its geometry. The upper and lower walls follow the same half-height function y(x) used in the design method. The inlet plane corresponds to station A_2 and the outlet plane to the tunnel exit. The inlet boundary condition was set using the same stagnation pressure and stagnation temperature as in the Q1-D model, and the outlet was prescribed at ambient static pressure. A structured mesh was generated along the walls and through the test section to resolve the acceleration and deceleration regions.

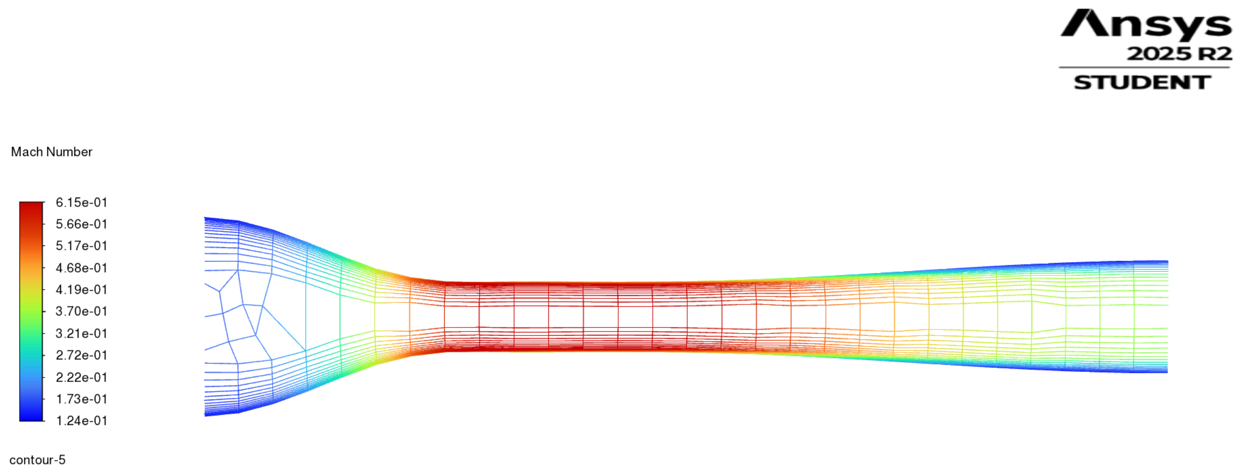

Figure 12. Contour plot of mach number throughout wind tunnel

The Mach number contours from the CFD solution show a smooth acceleration from the inlet to the throat of the contraction, followed by a nearly uniform high-Mach core through the test section and a gradual deceleration in the diffuser.

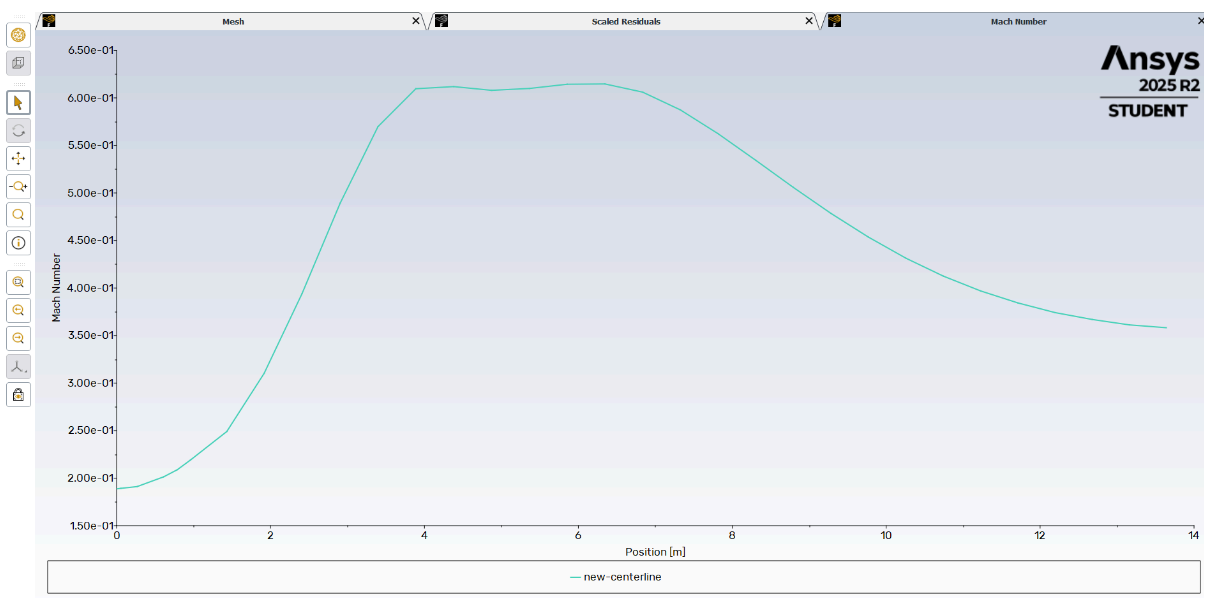

Figure 13. Mach number plot inside Ansys

The corresponding centerline plot indicates that the Mach number peaks at just above M = 0.6, in excellent agreement with the quasi-1D prediction. Small deviations between the CFD and Q1-D curves occur mainly near the contraction and diffuser entrances, where two-dimensional effects and streamline curvature become more important, but the overall shape and peak value match closely, confirming that the selected area distribution achieves the intended test-section Mach number.

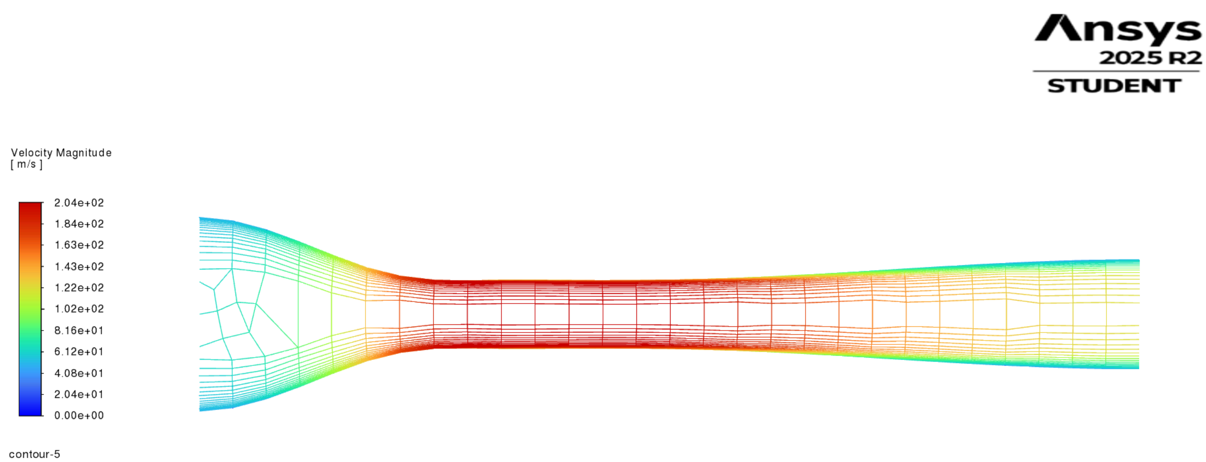

Figure 14. Contour plot of velocity magnitude throughout wind tunnel

The velocity-magnitude contour plot shows the expected behaviour: relatively low speeds at the inlet, strong acceleration through the contraction, a nearly uniform high-velocity core in the test section, and smooth deceleration toward the exit. The high-speed region is well confined to the test section, and the absence of large recirculation zones or strong gradients near the walls suggests that the diffuser operates without significant separation.

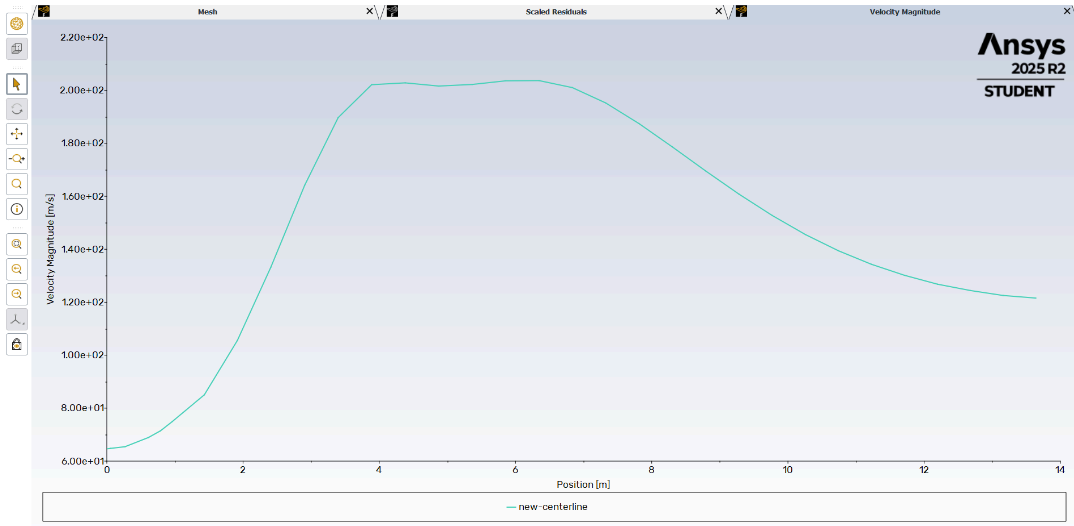

Figure 15. Velocity magnitude plot inside Ansys

The centerline velocity graph from the viscous CFD solution reaches a maximum slightly above 200 m/s, which is only a few percent higher than the value predicted by the quasi-1D model. The position of this peak aligns closely with the designed test-section region, and the downstream deceleration through the diffuser follows the same trend as the 1D result, with only minor differences in slope due to viscous losses. Overall, the comparison indicates that the 1D area sizing provides a realistic estimate of the actual core velocities in the full CFD solution.

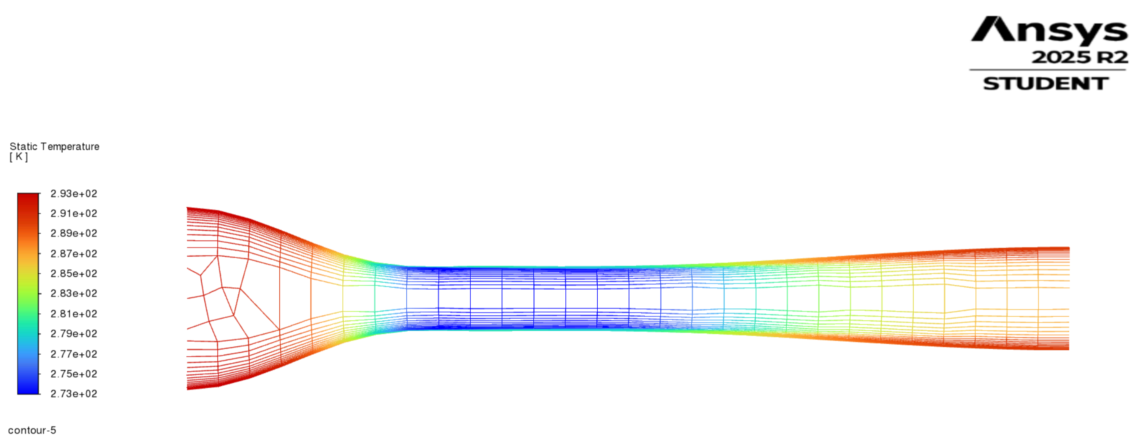

Figure 16. Contour plot of static temperature throughout wind tunnel

The static-temperature contour plot reflects the expected inverse relationship between temperature and speed in subsonic compressible flow: temperature decreases as the flow accelerates through the contraction and test section, and increases again as it decelerates in the diffuser. The coldest region is clearly confined to the test section, while the inlet and exit remain close to the ambient temperature, indicating that the high-speed core is localized where intended and that no strong thermal anomalies or hot spots develop in the diffuser.

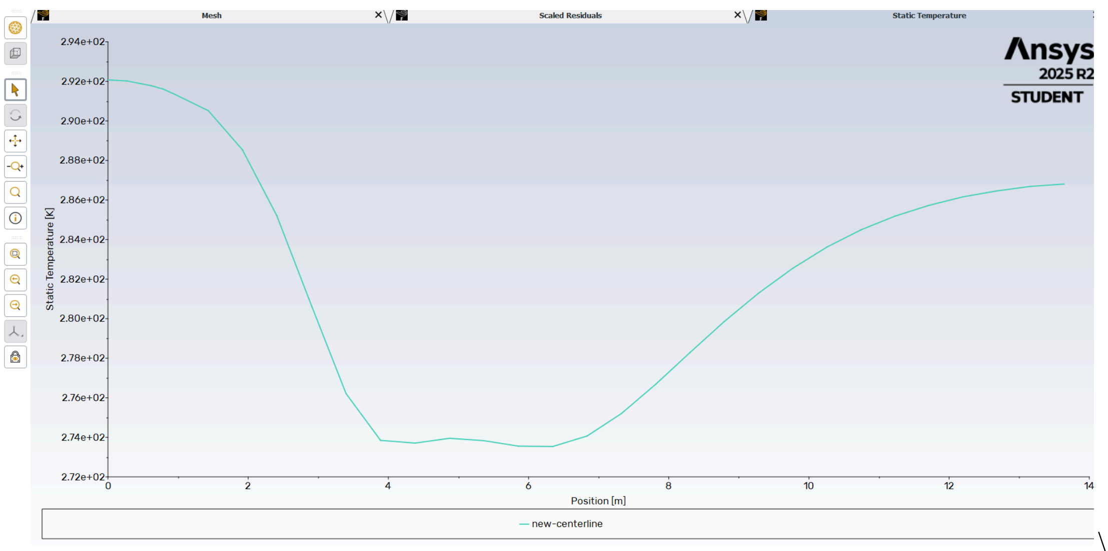

Figure 17. Static temperature plot inside Ansys

The centerline temperature graph from the viscous CFD solution shows a drop from an inlet value near 293K to a minimum of approximately 274–275 K in the test section, followed by a smooth recovery toward the exit. This trend closely matches the quasi-1-D isentropic prediction, with only small differences that can be attributed to viscous dissipation and finite total-pressure loss in the Navier–Stokes solution. Overall, the comparison confirms that the Q1-D analysis provides a good quantitative estimate of the thermal behaviour along the tunnel and that the flow remains comfortably within the intended low-speed compressible regime.

Overall, the CFD solutions agree well with the Q1D design: the peak Mach number, velocity level (around 200 m/s), and the locations of acceleration and deceleration all match within a few percent, and the temperature and density trends are very similar. The main differences come from genuinely two-dimensional and viscous effects, with the CFD showing slightly lower pressure recovery and mild non-uniformity near the walls due to boundary-layer growth in the diffuser. These discrepancies are modest and do not indicate any fundamental problem with the geometry. At most, the results suggest small refinements such as a slightly longer or more gently expanding diffuser, rather than a major redesign of the tunnel.

5. Conclusion

The work has produced a coherent preliminary design for a subsonic closed-circuit wind tunnel intended for basic external-aerodynamics testing. The final configuration features a square test section of 1 m^2 with a design Mach number of M_3 = 0.6, a contraction of length L_c = 3.4 m, a 3m test section, and a diffuser of length L_d = 7.24 m. The cross-sectional areas are approximately A_1 = 8 m^2 and A_2 = 8 m^2 at the inlet, A_3 = 1 m^2 in the test section, and A_4 = 3.09 m^2 at the exit, corresponding to a contraction ratio CR = 8. A smooth cubic half-height function y(x) was formulated to connect these areas while controlling wall gradients and avoiding abrupt geometric changes. Under the chosen stagnation conditions, the design delivers a peak test-section velocity of roughly 200 m/s and a Reynolds number of $1.36\,x10^7$, appropriate for airfoil and small-wing experiments in a low-speed facility.

The quasi-one-dimensional isentropic analysis provided centerline distributions of Mach number, velocity, static pressure, temperature, and density that satisfy the main design objectives: the test section exhibits nearly uniform Mach number and velocity, the diffuser recovers most of the static pressure without excessive adverse gradients, and the overall tunnel fits within an approximate 20 m laboratory length constraint. A simple trade study on contraction ratio and exit Mach number, guided by published recommendations, showed that a contraction ratio of 8 offers a good compromise between flow quality, fan power, and tunnel size, while an exit Mach number near 0.32 yields acceptable pressure recovery with manageable diffuser angles.

Two-dimensional CFD simulations, first inviscid (Euler) and then viscous (Navier–Stokes), were used. Both solutions reproduced the key features of the Q1-D design: a Mach number near 0.6 in the test section, peak velocity close to 200 m/s, and similar temperature and density trends. Differences are mainly due to two-dimensional and viscous effects, which lead to slightly reduced pressure recovery and mild non-uniformities near the walls from boundary-layer growth. These discrepancies are small and do not suggest a fundamental flaw in the geometry, but they do highlight limitations of the Q1D model and indicate where refinements could be beneficial, such as marginally lengthening or softening the diffuser and further tuning the contraction shape. Overall, the design appears reasonably robust to small parameter changes and provides a solid basis for a teaching-scale subsonic wind tunnel, while leaving clear avenues for future optimization and higher-fidelity validation.

APPENDIX

```python
# ========================================== #
# Author: Dararath Run (US 3223 Affiliate)
# ========================================== #

import numpy as np
import matplotlib.pyplot as plt
import math

# Constants
gamma = 1.4
R = 287.0                              # J/(kgB·K)
cp = gamma * R / (gamma - 1.0)         # в‰€ 1004.5 J/(kgB·K)

# Ambient / upstream stagnation
T01 = 294.15                           # K
p01 = 101325.0                         # Pa (1 atm)  -> this is pa

# Test section data
A3 = 1.0                               # m^2 (test-section area)
Ratio_A3 = 1.188                       # A3/A*
M3 = 0.6                               # test-section Mach

A_star = A3 / Ratio_A3                 # throat (star) area

# Diffuser design assumption
theta_d_deg = 3                        # diffuser half-angle in degrees
theta_d_rad = math.radians(theta_d_deg)

print(f"A3        = {A3:.6f} m^2")
print(f"A3/A*     = {Ratio_A3:.6f}")
print(f"A*        = {A_star:.6f} m^2\n")

def mass_flow(M, A, p0, T0):
    term = 1.0 + 0.5 * (gamma - 1.0) * M**2
    exponent = -(gamma + 1.0) / (2.0 * (gamma - 1.0))
    Gamma = math.sqrt(gamma) * M * term**exponent   # mass-flow function
    return Gamma * p0 / math.sqrt(R * T0) * A       # kg/s


def process_file(filename):
    data = np.loadtxt(
        filename,
        delimiter=',',
        comments=('#', '%'),
        skiprows=1
    )

    M4 = data[:, 0]           # exit Mach numbers
    Ratio_A4 = data[:, 1]     # A4/A*

    # rows: [A4, M4, V4, fan_PR, P_aero_MW, L_diff, m_dot, p4_over_pa]
    results = []

    for M, ratio in zip(M4, Ratio_A4):
        # Geometry: exit area
        A4 = ratio * A_star   # m^2

        # Fan total-pressure ratio from exit Mach (assuming p4 в‰€ pa)
        p02_over_p01 = (1.0 + 0.5 * (gamma - 1.0) * M**2)**(gamma / (gamma - 1.0))
        p02 = p02_over_p01 * p01
        T02 = T01 * p02_over_p01**((gamma - 1.0) / gamma)

        # Mass flow, using test section (same M3 and A3 for all cases)
        m_dot = mass_flow(M3, A3, p02, T02)   # kg/s

        # Exit static temperature and velocity
        T4 = T02 / (1.0 + 0.5 * (gamma - 1.0) * M**2)
        a4 = math.sqrt(gamma * R * T4)
        V4 = M * a4                           # m/s

        # Exit static pressure using the formula you gave:
        # p4 = p02 * ( 1 / (1 + (gamma-1)/2 * M^2) )^(gamma/(gamma-1))
        term = 1.0 + 0.5 * (gamma - 1.0) * M**2
        p4 = p02 * (1.0 / term)**(gamma / (gamma - 1.0))
        p4_over_pa = p4 / p01                # since pa = p01

        # Aerodynamic fan power
        P_aero = m_dot * cp * (T02 - T01)    # W
        P_aero_MW = P_aero / 1e6             # convert to MW

        # Diffuser length from A3 -> A4
        y3 = math.sqrt(A3) / 2.0
        y4 = math.sqrt(A4) / 2.0
        L_diff = (y4 - y3) / math.tan(theta_d_rad)  # m

        results.append([A4, M, V4, p02_over_p01, P_aero_MW, L_diff, m_dot, p4_over_pa])

    results = np.array(results)

    # PNG table
    col_labels = ["A4 [mBI]", "M4", "V4 [m/s]", "p02/p01",
                  "P_aero [MW]", "L_diff [m]", "p4/pa"]

    fig, ax = plt.subplots(figsize=(10, 0.6 + 0.3 * len(results)))
    ax.axis('off')

    cell_text = [[f"{val:.3f}" for val in row[[0,1,2,3,4,5,7]]] for row in results]

    table = ax.table(cellText=cell_text,
                     colLabels=col_labels,
                     loc='center')
    table.auto_set_font_size(False)
    table.set_fontsize(9)
    table.scale(1.0, 1.2)

    plt.tight_layout()
    plt.savefig("trade_study_table.png", dpi=300)
    plt.close()
    print("\nSaved PNG table as 'trade_study_table.png'.")


if __name__ == "__main__":
    process_file("M and Ratio.csv")
```

```python
# ============================================ #
# Author: Dararath Run (US 3223 Affiliate)
# ============================================ #

import math

# Parameters

gamma = 1.4          # ratio of specific heats
R     = 287.0        # J/(kg K)
T0    = 294.15       # K
p0    = 101325.0     # Pa
A     = 3.092         # m^2
M     = 0.160         # Mach number

# Equations

def q_of_M(M, gamma):
    # q(M) = M * [ 2/(Oi+1) * (1 + (Oi-1)/2 * M^2 ) ]^{-(Oi+1)/(2(Oi-1))}

    term = (2.0 / (gamma + 1.0)) * (1.0 + 0.5 * (gamma - 1.0) * M**2)
    exponent = -(gamma + 1.0) / (2.0 * (gamma - 1.0))
    return M * term**exponent

def Gamma(gamma):
    # O" = sqrt(Oi) * ((Oi+1)/2)^{-(Oi+1)/(2(Oi-1))}

    exponent = -(gamma + 1.0) / (2.0 * (gamma - 1.0))
    return math.sqrt(gamma) * ((gamma + 1.0) / 2.0)**exponent

# Compute values
Gamma_val = Gamma(gamma)
qM        = q_of_M(M, gamma)
m_dot     = Gamma_val * (p0 / math.sqrt(R * T0)) * qM * A   # kg/s

# Print

print(f"Gamma (O")      = {Gamma_val:.6f}")
print(f"q(M)           = {qM:.6f}")
print(f"Mass flow rate = {m_dot:.3f} kg/s")
```

```python
# ========================================== #
# Author: Dararath Run (US 3223 Affiliate)
# ========================================== #

import numpy as np
import matplotlib.pyplot as plt
import math
import os
from datetime import datetime

# Parameters
gamma = 1.4
R     = 287.0            # J/(kgB·K)
T0    = 294.15           # K  (stagnation temperature)
p0    = 101325.0         # Pa (stagnation pressure)

# Full heights [m]
H1 = 2.83   # Contraction inlet height
H3 = 1.00   # Test section height  <-- station 3
H4 = 1.60   # Diffuser exit height (adjust if needed)

# Lengths [m] (no duct now)
Lc = 3.41   # Contraction
Lt = 3.00   # Test section
Ld = 7.24   # Diffuser

L_total = Lc + Lt + Ld

# Design Mach number in Test Section
M3_design = 0.60

# x grid
dx = 0.02
x  = np.arange(0.0, L_total + dx/2, dx)

#  Half-height Function
def y_half(x):
    y = np.zeros_like(x)

    # Section 1: contraction (0 -> Lc)
    mask = (x >= 0.0) & (x <= Lc)
    xi = x[mask] / Lc                  # 0..1
    S  = 3.0 * xi**2 - 2.0 * xi**3
    y[mask] = H1/2.0 + (H3/2.0 - H1/2.0) * S

    # Section 2: test section (Lc -> Lc+Lt)
    mask = (x > Lc) & (x <= Lc + Lt)
    y[mask] = H3 / 2.0

    # Section 3: diffuser (Lc+Lt -> end)
    mask = (x > Lc + Lt) & (x <= L_total)
    eta = (x[mask] - (Lc + Lt)) / Ld  # 0..1
    S  = 3.0 * eta**2 - 2.0 * eta**3
    y[mask] = H3/2.0 + (H4/2.0 - H3/2.0) * S

    return y

y   = y_half(x)
H   = 2.0 * y          # full height
A   = H**2             # assume square cross-section: A = H^2

# Area-Mach relation
def area_ratio(M):
    term = (2.0 / (gamma + 1.0)) * (1.0 + 0.5 * (gamma - 1.0) * M**2)
    exponent = (gamma + 1.0) / (2.0 * (gamma - 1.0))
    return (1.0 / M) * term**exponent

# Use station 3 (test section) to set A*
A3 = H3**2
A3_over_At = area_ratio(M3_design)
A_star = A3 / A3_over_At

def solve_M_subsonic(A_over_At_target, tol=1e-6):
    if A_over_At_target <= 1.0:
        return 1.0  # close to choked, just return ~1

    M_lo, M_hi = 1e-4, 0.999
    f_lo = area_ratio(M_lo) - A_over_At_target

    for _ in range(60):
        M_mid = 0.5 * (M_lo + M_hi)
        f_mid = area_ratio(M_mid) - A_over_At_target
        if abs(f_mid) < tol:
            return M_mid
        if f_lo * f_mid < 0.0:
            M_hi = M_mid
        else:
            M_lo = M_mid
            f_lo = f_mid
    return M_mid

# Centerline properties
M   = np.zeros_like(x)
T   = np.zeros_like(x)
p   = np.zeros_like(x)
rho = np.zeros_like(x)
V   = np.zeros_like(x)

for i, Ax in enumerate(A):
    A_ratio_i = Ax / A_star
    M[i] = solve_M_subsonic(A_ratio_i)

    theta = 1.0 + 0.5 * (gamma - 1.0) * M[i]**2
    T[i]  = T0 / theta
    p[i]  = p0 * theta**(-gamma / (gamma - 1.0))
    rho[i]= p[i] / (R * T[i])
    V[i]  = M[i] * math.sqrt(gamma * R * T[i])

# Create output folder
timestamp = datetime.now().strftime("%Y-%m-%d_%H-%M-%S")
output_dir = f"centerline_plots_{timestamp}"
os.makedirs(output_dir, exist_ok=True)

# Individual plots (one PNG per graph) ===
individual_plots = [
    (M,           "Mach number",       "x [m]",    "M(x)",                 "M_vs_x.png"),
    (p/1000.0,    "Static pressure",   "x [m]",    "p [kPa]",              "p_vs_x.png"),
    (T,           "Static temperature","x [m]",    "T [K]",                "T_vs_x.png"),
    (V,           "Velocity",          "x [m]",    "V [m/s]",              "V_vs_x.png"),
    (rho,         "Density",           "x [m]",    r"$\rho$ [kg/m$^3$]",  "rho_vs_x.png"),
]

for y_data, title, xlabel, ylabel, fname in individual_plots:
```

```python
    plt.figure(figsize=(6, 4))
    plt.plot(x, y_data)
    plt.title(title)
    plt.xlabel(xlabel)
    plt.ylabel(ylabel)
    plt.grid(True)
    plt.tight_layout()
    plt.savefig(os.path.join(output_dir, fname), dpi=300)
    plt.close()

# Combined 3x2 plot
fig, axs = plt.subplots(3, 2, figsize=(10, 9), sharex=True)
axs = axs.flatten()  # easier indexing: axs[0]..axs[5]

# Mach number
axs[0].plot(x, M)
axs[0].set_ylabel("M(x)")
axs[0].set_title("Mach number")
axs[0].grid(True)

# Static pressure
axs[1].plot(x, p / 1000.0)
axs[1].set_ylabel("p [kPa]")
axs[1].set_title("Static pressure")
axs[1].grid(True)

# Static temperature
axs[2].plot(x, T)
axs[2].set_ylabel("T [K]")
axs[2].set_title("Static temperature")
axs[2].grid(True)

# Velocity
axs[3].plot(x, V)
axs[3].set_ylabel("V [m/s]")
axs[3].set_title("Velocity")
axs[3].grid(True)
axs[3].set_xlabel("x [m]")

# Density
axs[4].plot(x, rho)
axs[4].set_ylabel(r"$\rho$ [kg/m$^3$]")
axs[4].set_title("Density")
axs[4].grid(True)
axs[4].set_xlabel("x [m]")

# Last subplot empty
axs[5].axis("off")

plt.tight_layout()
combined_path = os.path.join(output_dir, "centerline_properties_3x2.png")
plt.savefig(combined_path, dpi=300)
plt.close()

# 1x3 plot of p, T, rho
fig, axs = plt.subplots(1, 3, figsize=(12, 4), sharex=True)

# Pressure
axs[0].plot(x, p / 1000.0)
axs[0].set_title("Static pressure")
axs[0].set_ylabel("p [kPa]")
axs[0].set_xlabel("x [m]")
axs[0].grid(True)

# Temperature
axs[1].plot(x, T)
axs[1].set_title("Static temperature")
axs[1].set_ylabel("T [K]")
axs[1].set_xlabel("x [m]")
axs[1].grid(True)

# Density
axs[2].plot(x, rho)
axs[2].set_title("Density")
axs[2].set_ylabel(r"$\rho$ [kg/m$^3$]")
axs[2].set_xlabel("x [m]")
axs[2].grid(True)

plt.tight_layout()
ptrho_path = os.path.join(output_dir, "p_T_rho_1x3.png")
plt.savefig(ptrho_path, dpi=300)
plt.close()

print(f"Saved individual plots, 3x2 combined plot, and 1x3 p-T-rho plot in folder: {output_dir}")
```

```python
# ========================================= #
# Author: Dararath Run (US 3223 Affiliate)
# ========================================= #

import numpy as np
import matplotlib.pyplot as plt
import math

# Flow + Geometry Parameters
gamma = 1.4
R     = 287.0             # J/(kgB·K)
T0    = 294.15            # K  (stagnation temperature)
p0    = 101325.0          # Pa (stagnation pressure)

# Full tunnel heights [m]
H1 = 2.83   # Contraction inlet height
H3 = 1.00   # Test section height  <-- station 3
H4 = 1.60   # Diffuser exit height (adjust if needed)

# Lengths [m] (no duct)
Lc = 3.41   # Contraction length
Lt = 3.00   # Test section length
Ld = 7.24   # Diffuser length

L_total = Lc + Lt + Ld

# Design Mach number in TEST SECTION
M3_design = 0.60

# x grid
dx = 0.02
x  = np.arange(0.0, L_total + dx/2, dx)

# Half-height function
def y_half(x):
    x = np.asarray(x)
    y = np.zeros_like(x)

    # Section 1: contraction (0 -> Lc)
    mask = (x >= 0.0) & (x <= Lc)
    xi = x[mask] / Lc
    S  = 3.0 * xi**2 - 2.0 * xi**3
    y[mask] = H1/2.0 + (H3/2.0 - H1/2.0) * S

    # Section 2: test section (Lc -> Lc+Lt)
    mask = (x > Lc) & (x <= Lc + Lt)
    y[mask] = H3 / 2.0

    # Section 3: diffuser (Lc+Lt -> L_total)
    mask = (x > Lc + Lt) & (x <= L_total)
    eta = (x[mask] - (Lc + Lt)) / Ld
    S  = 3.0 * eta**2 - 2.0 * eta**3
    y[mask] = H3/2.0 + (H4/2.0 - H3/2.0) * S

    return y

# compute half-height and full height/area
y   = y_half(x)
H   = 2.0 * y          # full height
A   = H**2             # (if you still want area later)

# Mesh creation

# Top wall:    (x, +y)
top_wall    = np.column_stack((x,  y))
# Bottom wall: (x, -y)
bottom_wall = np.column_stack((x, -y))

# Curve IDs: 1 for top, 2 for bottom (two separate polylines)
top_ids    = np.ones_like(x, dtype=int)          # 1 1 1 ...
bottom_ids = np.full_like(x, 2, dtype=int)       # 2 2 2 ...

# Build [curve_id, x, y]
top_export    = np.column_stack((top_ids,    top_wall))
bottom_export = np.column_stack((bottom_ids, bottom_wall))

# Stack into one table for one TXT file
export_data = np.vstack((top_export, bottom_export))

# Fixed-point formats: first col int, then decimals (no scientific notation)
fmt_list = ['%d', '%.6f', '%.6f']

# Header row: "Polyline = true" in A1, nothing in B1/C1
header_line = "Polyline = true,,"

np.savetxt(
    "tunnel_walls.txt",
    export_data,
    delimiter=",",
    header=header_line,
    comments="",
    fmt=fmt_list
)

print("Wrote 'tunnel_walls.txt' for SpaceClaim import (two polylines, IDs 1 and 2).")

# Geometry Shape Plot
fig, ax = plt.subplots(figsize=(14, 3))

# Shaded sections
contraction_patch = ax.axvspan(0.0, Lc, facecolor='C0', alpha=0.08, label="Contraction")
test_patch        = ax.axvspan(Lc, Lc + Lt, facecolor='C1', alpha=0.08, label="Test section")
diffuser_patch    = ax.axvspan(Lc + Lt, L_total, facecolor='C2', alpha=0.08, label="Diffuser")

# Walls
top_line,    = ax.plot(x,  y,  label="Top wall")
bottom_line, = ax.plot(x, -y, label="Bottom wall")

# Vertical dashed lines at section boundaries
ax.axvline(Lc,       color='k', linestyle='--', linewidth=1)
ax.axvline(Lc + Lt,  color='k', linestyle='--', linewidth=1)

# Text labels for each section (tweak y-position to taste)
label_y = 1.3
ax.text(Lc/2.0,             label_y, "Contraction", ha="center", va="center", fontsize=16)
ax.text(Lc + Lt/2.0,        label_y, "Test section", ha="center", va="center", fontsize=16)
ax.text(Lc + Lt + Ld/2.0,   label_y, "Diffuser",    ha="center", va="center", fontsize=16)

# Axes labels and title
ax.set_xlabel("x [m]")
ax.set_ylabel("y [m]")
ax.set_title("Wind Tunnel Geometry")
```

```
# Limits to resemble your example
ax.set_xlim(0.0, L_total)
ax.set_ylim(-1.6, 1.6)

# Grid and formatting
ax.grid(True)
ax.ticklabel_format(style='plain', axis='both')  # no sci-notation on axes

# Legend (shaded regions + walls)
handles = [contraction_patch, test_patch, diffuser_patch, top_line, bottom_line]
labels  = [h.get_label() for h in handles]
ax.legend(handles, labels, loc="upper right", framealpha=1.0)

plt.tight_layout()
plt.savefig("geometry_check.png", dpi=300)
plt.close()

print("Saved 'geometry_check.png'.")
```

References

[1] Bell, J. H., & Mehta, R. D. (1988). *Contraction Design for Small Low-Speed Wind Tunnels*. NASA Technical Memorandum 100547.
https://ntrs.nasa.gov/api/citations/19880012661/downloads/19880012661.pdf

[2] Anderson, J. D. (2017). *Fundamentals of Aerodynamics* (6th ed.). McGraw-Hill Education.

[3] ABB Group. (2017). *Technical Guide No. 5: Fundamentals of AC Drives*.
https://search.abb.com/library/Download.aspx?DocumentID=9AKK103508&LanguageCode=en&DocumentPartId=&Action=Launch

[4] Liu, P., & Hu, Z. (2020). *Three-Dimensional Simulation of Wind Tunnel Turbulence Characteristics*. *Physics of Fluids, 32*(5), 052006.
https://pubs.aip.org/aip/pof/article-abstract/32/5/052006/963424

[5] NASA. (1977). *Low-Speed Wind Tunnel Design and Performance Data*. NASA Technical Report 19770005050.
https://ntrs.nasa.gov/api/citations/19770005050/downloads/19770005050.pdf