11th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2021-125**
8-10 September 2021 - METU, Ankara TURKEY

PRELIMINARY DESIGN OF A MID-SCALE SUPERSONIC WIND TUNNEL

Kadir Giray Kuyumcu[1] and Yaşar Ostovan[2]
Sivas University of Science and Technology
Sivas, Turkey

ABSTRACT

This paper presents the preliminary design of a mid-scale supersonic wind tunnel. Test section cross-sectional dimensions are chosen as 0.4 m by 0.4 m and maximum speed is aimed as Mach 4. The type of the tunnel is chosen to be blow-down considering its relatively low cost and capability of operating at different Reynolds numbers. Preliminary calculations are performed to obtain a run time of 60 seconds at the speed of Mach 4. Assuming the outlet pressure as atmospheric pressure at sea level, the Required inlet pressure to be able to start the tunnel at Mach 4 is found to be about 15 bars. A compressor with a maximum compression capacity of 40 bars with a tank capacity of 40 m^3 *is selected to allow a 60 seconds experiment at Mach 4. The nozzle geometry of the tunnel is designed by utilizing Method of Characteristics. Nozzle contour is created by a code presented by (J. C. Sivells) which combines the method of characteristics with analytical solutions. Coordinates obtained from MoC are imported into a computer-aided drawing software and nozzle geometry is created. Finally, the nozzle geometry is validated by CFD method using ANSYS Fluent software. Results from CFD analysis show that the Mach number uniformity throughout the outlet boundary is satisfactory.*

INTRODUCTION

Wind tunnels are used to study the interaction of an object in contact with a moving fluid. They are utilized to test scaled models in a free stream flow provided under controlled conditions as if they are flying in real conditions [John D. Anderson, 2012]. Aerial vehicle aerodynamics and propulsion systems have to be examined and optimized in wind tunnels prior to finalizing the design and prototyping. Computational Fluid Dynamics simulations (i.e. turbulence models) also need approval by wind tunnel experiments. For high-speed aerodynamics, it is important to have sufficient infrastructures to study shock waves and their interactions with solid bodies as well as boundary layers around them. Transonic, supersonic, and hypersonic wind tunnels are crucial facilities in understanding the aerodynamic behavior of high-speed aerial vehicles. Supersonic wind tunnels are the most utilized tunnels for high-speed aerodynamic studies. There are several types of supersonic wind tunnels. Two main categories are intermittent and continuous closed-circuit tunnels [Bharath, 2015]. Continuous closed-circuit tunnels are advantageous over intermittent types in terms of achieving desired flow conditions and durability of experiments with relatively accurate flow conditions. On the other hand,

---

[1] Research Asst. in Astronautical Engineering Department, Email: kgkuyumcu@sivas.edu.tr
[2] Asst. Prof. in Aeronautical Engineering Department, Email: yostovan@sivas.edu.tr

intermittent tunnels are simpler to design and much cheaper to build. Moreover, a failure of the model support system will result in less catastrophic damage to the facility [Pope & Goin, 1965]. For this study, a blow-down intermittent supersonic tunnel is considered. In blow-down type, the air flows down from a high-pressure reservoir tank to the test section of the tunnel through a nozzle which transforms the high-pressure subsonic flow to a supersonic flow.

METHODOLOGY

As mentioned earlier, the tunnel is selected as intermittent blow-down due to its several advantages over the continuous type such as ease of installation, lower cost, and shorter setup time. In this type of tunnel, the air does not circulate in the system and is released into the atmosphere from the outlet of the nozzle. Figure 1 presents a schematic of a blow-down supersonic wind tunnel and its main components. Pressurized air is stored in a reservoir tank using a compressor. The pressure regulator is a high-frequency response valve to allow necessary pressure to start the tunnel and then regulate the flow rate to the desired operating conditions. In the settling chamber, the airflow becomes uniform utilizing flow straighteners upstream the nozzle. Then in the nozzle throat, the flow accelerates from subsonic speed to the desired supersonic speed and enters the test section. In addition to flow straighteners, nozzle geometry (contour) plays a crucial role to obtain a uniform flow inside the test section. Downstream of the test section, a diffuser is used to decelerates the flow back to subsonic speed. Finally, a silencer is utilized to minimize the sound level of the system [Xia, 2018].

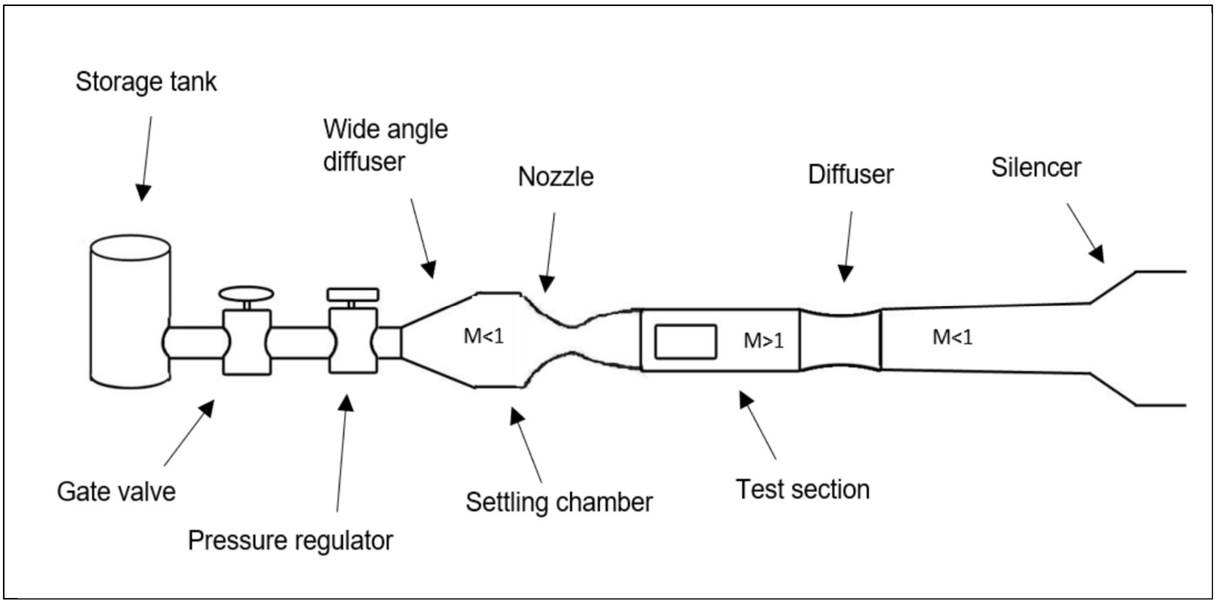

Figure 1: Schematic of the main components of a supersonic blow-down wind tunnel

The first challenge is to design an appropriate nozzle to have the maximum flow uniformity inside the test section at the desired operating conditions. The second objective is to design a diffuser with minimum losses. For the preliminary design, we assume to have sonic flow at both throats (nozzle and diffuser) as well as isentropic flow through the nozzle. Moreover, the flow is assumed to be adiabatic inside the tunnel. Using these assumptions and the following equations the nozzle and the diffuser throat areas are calculated.

$$A_{t1}.\rho_1^*.a_1^* = A_{t2}.\rho_2^*.a_2^* \quad (1)$$

$$A_{t2}/A_{t1} = \rho_1^*.a_1^* \,/\, \rho_2^*.a_2^* \quad (2)$$

In Equations (1) and (2), $\boldsymbol{A_{t1}}$ and $\boldsymbol{A_{t2}}$ represent the cross-sectional areas of nozzle and diffuser parts, respectively. $\boldsymbol{a_1^*}$ and $\boldsymbol{a_2^*}$ indicate the speed of sound in these parts and are considered to be equal.

$$\boldsymbol{A_{t2}/A_{t1} = \rho_1^* \,/\, \rho_2^* = (P_1^*/\, RT_1^*) \,/\, (P_2^*/\, RT_2^*)} \tag{3}$$

Nozzle throat area, $\boldsymbol{A_{t1}}$, is calculated as 0.015 $\boldsymbol{m^2}$ by using isentropic flow properties. Also, process is assumed as adiabatic. So, the diffuser cross-sectional area is determined by Equation (4):

$$\boldsymbol{A_{t2}/A_{t1} = P_1^* \,/\, P_2^*} \tag{4}$$

where $\boldsymbol{T_1^* = T_2^*}$ and $\boldsymbol{A_{t2}}$ = 0.108 $\boldsymbol{m^2}$

$$P^* = P_0\ (2/(\gamma + 1))^{\,\gamma/(\gamma-1)} \tag{5}$$

$$A_{t2} = A_{t1} \times P_{01} \,/\, P_{02} \tag{6}$$

The Pressure ratios P_{01}/P_{02} are estimated as 7,2 from the isentropic flow assumption to achieve Mach 4 at the test section. Required pressure ratio to start the tunnel can be considered as the double of pressure ratio for operating conditions [John D. Anderson, 2012; Pope & Goin,1965].

The mass flow rate at Mach 4 is calculated as 27 kg/s using the following equation:

$$\dot{m} = [A^* \,.\, P_0 \,/\, \sqrt{T_0}\,] \,.\, \sqrt{\frac{\gamma}{R} \cdot \left[\frac{2}{\gamma+1}\right]^{\left(\frac{\gamma+1}{\gamma-1}\right)}} \tag{7}$$

where $A^* = A_{t1}$

The total mass of the air needed for a 60 seconds tunnel operation is calculated as 1650 kg. Utilizing a compressor with a compression capacity of 40 bars, a reservoir tank of *40* m^3 will be sufficient to store enough air to run the tunnel for 60 seconds.

With the same pressurized air reservoir capacity, If the desired speed in the test section is changed, the operation time changes as well. Table 1 shows the run times of the tunnel at different Mach Numbers, beginning from Mach 1,5 up to Mach 4 with 0,5 increment. Our calculations show that the tunnel will have a run time of 111 (sec) at Mach 1,5.

Table 1: Run times vs. Mach Number

Mach Number	Operational Pressure (bar)	Run Time (sec)
1,5	1,08	111
2	1,4	92
2,5	2	81
3	3,04	73
3,5	4,7	67
4	7,2	60

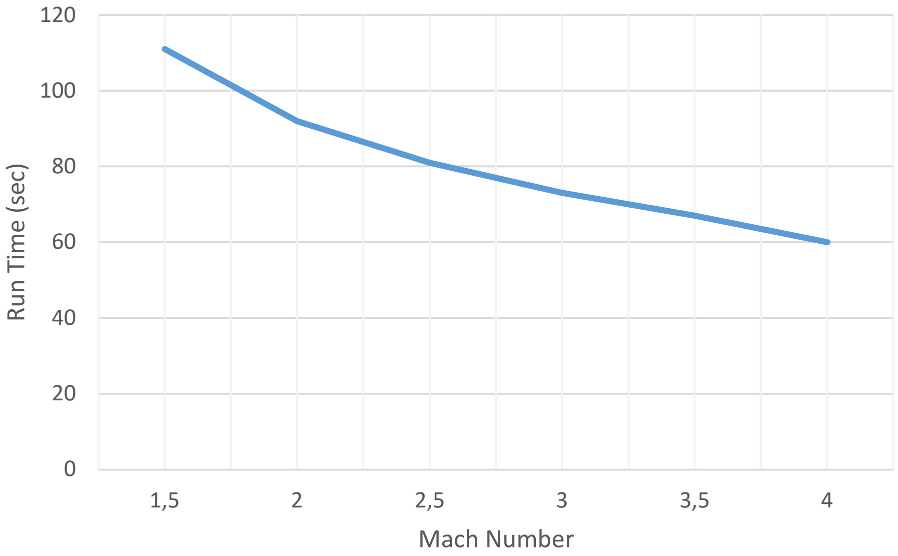

Figure 2: Run time vs Mach number

Nozzle Geometry Design with Method of Characteristics

Nozzles are such components that expand airflow to desired supersonic conditions. There are two main categories of supersonic nozzles: minimum length nozzles and gradually expanded nozzles (Figure 3). Minimum length nozzles are commonly used to decrease the weight and length of the nozzle, especially in rockets and jet engines. Gradual-expansion type of nozzles are generally used where providing a high-quality flow is a significant factor, such as wind tunnels [Khan & Sardiwal, 2013].

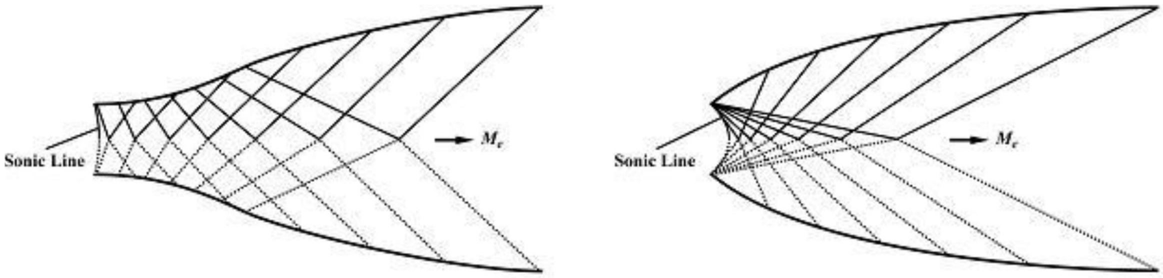

Figure 3: Gradual-expansion nozzle (left) vs minimum length nozzle (right)

Method of characteristics (moc) is a suitable numerical method for solving two-dimensional compressible flow problems. Using this technique, flow characteristics such as direction and velocity can be calculated at different points throughout the flow field [Moore, 2009]. Figure 4 shows characteristic lines of a gradual-expansion nozzle.

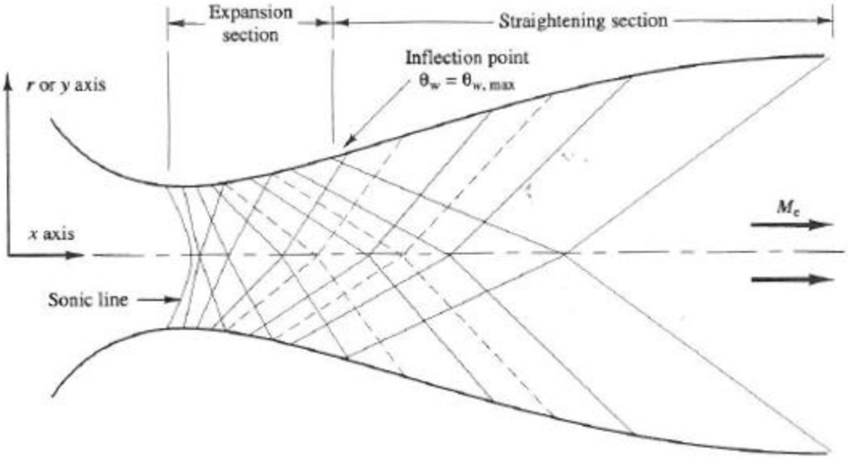

Figure 4: Characteristic lines of a gradual-expansion nozzle

To create the contour of the nozzle, the method of characteristics is a helpful process for minimum length nozzles. However, gradual expansion nozzles, the type presented in this paper, needs an analytical solution in addition to the method of characteristics [J. C. Sivells, 1978]. For this purpose, J. C. Sivells, developed a FORTRAN code, called “CONTUR”.

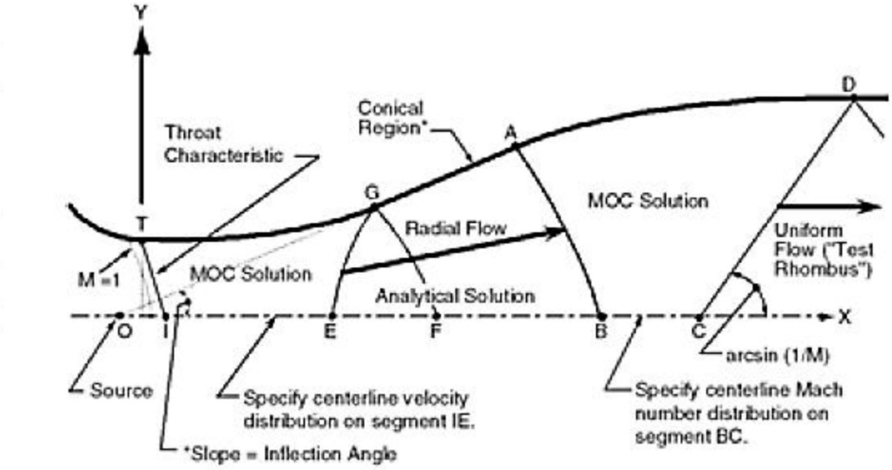

Figure 5: Design Procedure of CONTUR [Shope, 2005]

Sivell’s CONTUR, combines the method of characteristics, analytical solutions, and centerline distribution [Adams, 2016] as shown in Figure 5. Also, it provides both axisymmetric and planar solutions. The program begins with a transonic solution in order to solve throat of the nozzle. Then, it works with analytical solution for the radial flow and continues with the method of characteristics solution. For ease of manufacturing, the planar or 2D nozzle design is utilized in this work. The code is to be provided by input cards and these cards are created with different parameters, some of them are given in Table-2. The input values are selected according to our calculations given above.

Table 2: Input parameters for CONTUR

Contour type	Planar
Specific heat ratio	1.4
Gas constant	1716,5 ft^2/sec^2R
Design Mach number	4
Half distance between parallel walls	7,87 inches
Number of characteristic lines	50

Outputs of the CONTUR are obtained as x and y-axis coordinates. To create the geometry of the nozzle, these coordinates are imported to a computer-aided drawing program. Figure 6 shows the created nozzle geometry. Area ratio between outlet of the nozzle and the throat is found as 10,72 which is consistent with our calculations assuming isentropic flow condition through the nozzle.

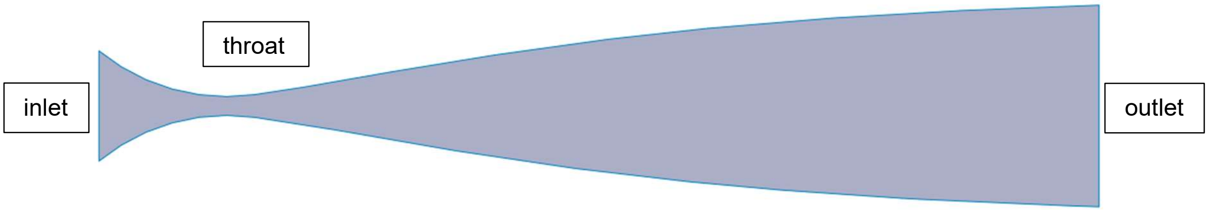

Figure 6*:* Nozzle geometry

CFD Validation of Nozzle Geometry Design

After the nozzle geometry is created, the flow evolution and its quality is validated with CFD (Computational Fluid Dynamics) analysis. ANSYS Fluent R18.2 provides comprehensive modeling capabilities for various laminar and turbulent compressible and incompressible fluid flow problems. For all flows, ANSYS Fluent solves mass and moment conservation equations. Additionally, for flows involving heat transfer or compressibility, it also considers energy conservation equation [Fluent Theory Guide, 2013].

There are two main solver types as pressure-based and density-based in Fluent. For high-speed flows, a density-based solver is a better option due to high compressibility of the air in supersonic flows. The density-based solver simultaneously solves the equations that govern continuity, momentum, and (when appropriate) the transfer of energy and species [Fluent Theory Guide, 2013].

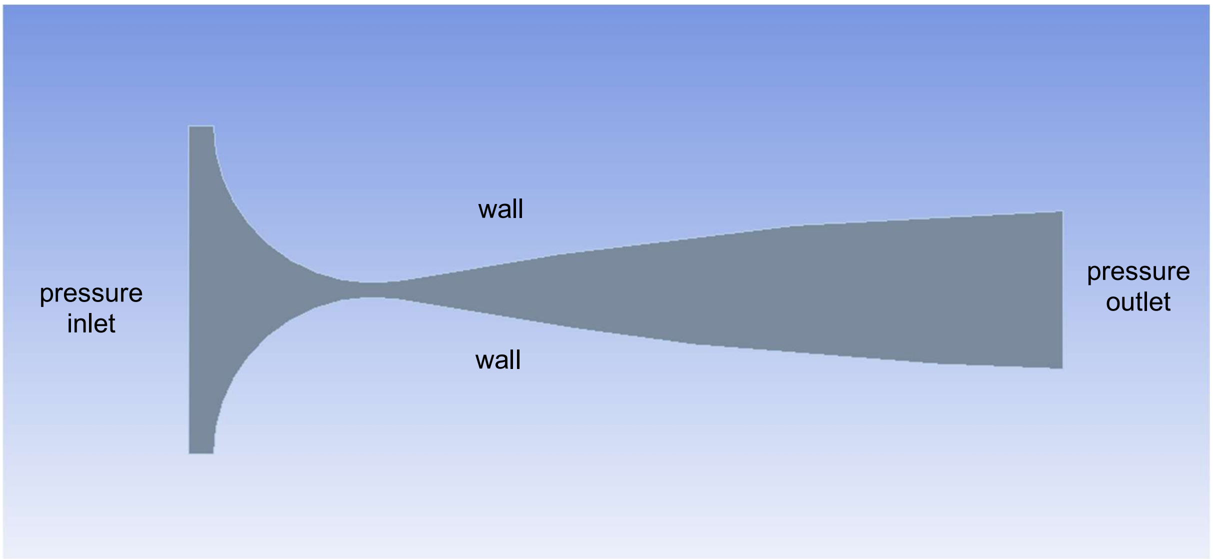

Figure 7: Boundaries of the geometry

For our CFD analysis, boundary conditions are selected as pressure-inlet, pressure-outlet and non-slip wall as shown in Figure 7. For the meshing process, the same ANSYS module is utilized. To get a better solution, triangular mesh is used and inflation is imported on the walls to capture the boundary layer effect clearly. Meshing process created 752,178 nodes and 1,455,962 elements on the geometry. Figure 8 shows the gradual increase in the mesh size moving from the wall toward the centerline.

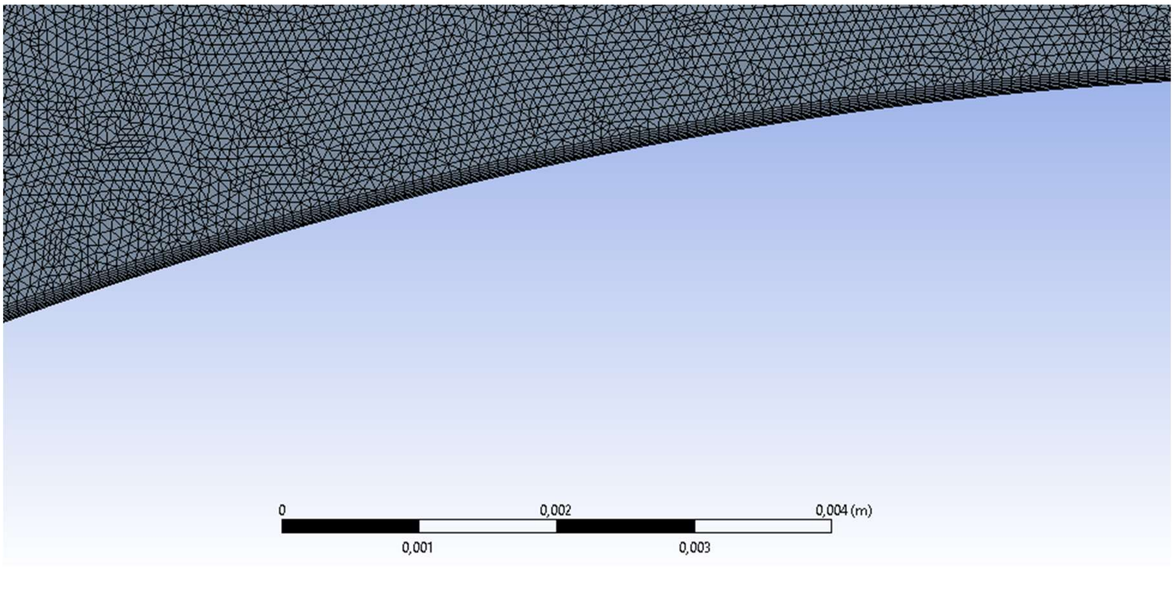

Figure 8: Gradual increase in the mesh size moving from the wall toward the centerline

CFD Setup and Solution: To perform the simulation, $k-\omega\,SST$ (shear stress transport) turbulence model is utilized. This model is widely used in many aerodynamic applications involving boundary layer evolution. Basically $k-\omega\,SST$ combines two equations $k-\omega$ on near the wall and $k-\mathcal{E}$ model in the free-stream. [Fluent Theory Guide, 2013]. It gives a better solution compared to the inviscid model and other viscous models on boundary layer. Considering isentropic relations through the nozzle, inlet and outlet pressures are calculated as 7 bars and 0,046 bar respectively and implemented in boundary conditions of the simulation. The solution is converged after 1500 iterations achieving desired residual limits. Figure 9 shows the Mach number contour from the Fluent simulation in the designed nozzle geometry using MoC.

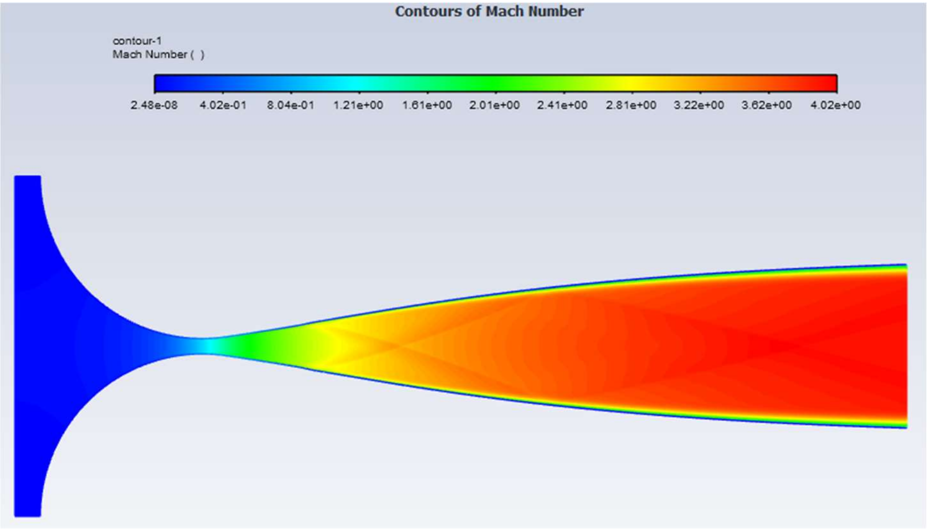

Figure 9: Mach number contour from the Fluent simulation in the designed nozzle geometry

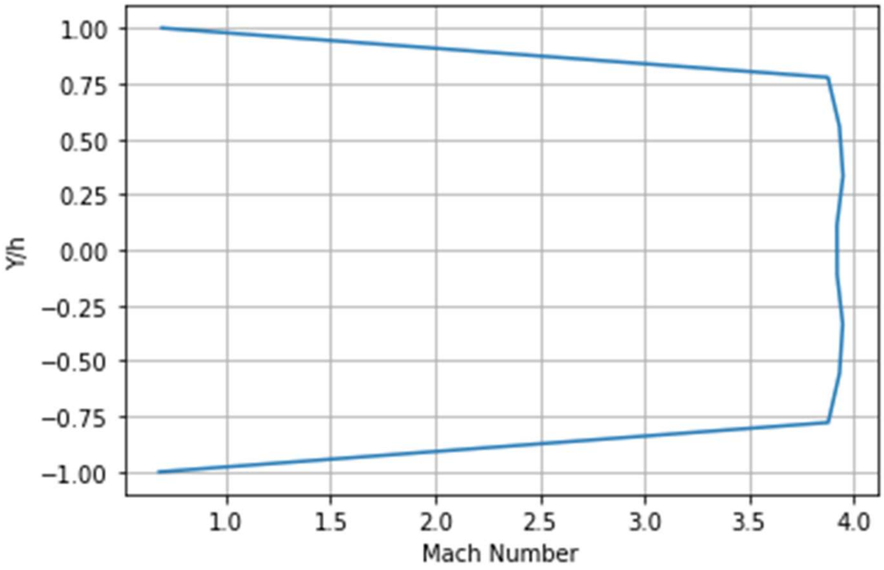

Figure 10: Mach number vertical distribution in the outlet of the nozzle

By post-processing the simulation results, axial velocity (Mach number) distribution in the outlet of the nozzle is obtained. It is shown in Figure 10 that Mach number uniformity throughout the outlet boundary is satisfactory.

CONCLUSION

The preliminary design of an intermittent blow-down supersonic wind tunnel with maximum speed of Mach 4 is presented. Necessary calculations and assumptions are performed to obtain working conditions and nozzle throat dimensions. Nozzle geometry contour is created by Method of Characteristics utilizing a code that combines the MoC and analytical solutions. Coordinates of the nozzle boundary are imported to a CAD software and CFD analysis with ANSYS Fluent program are performed to validate the flow evolution through the designed nozzle geometry. Results from CFD analysis show that Mach number magnitude and its uniformity throughout the outlet boundary of the nozzle are satisfactory.

References

Adams, S. E. (2016). *The Design and Computational Validation of a Mach 3 Wind Tunnel Nozzle Contour*.

Bharath, B. K. (2015). Design and Fabrication of a Supersonic Wind Tunnel. *International Journal of Engineering and Applied Sciences*, *2*(5).

Fluent Theory Guide. (2013). Ansys Fluent Theory Guide. In *ANSYS Inc., USA* (Vol. 15317, Issue November, pp. 724–746).

John D. Anderson, J. (2012). Fundamentals of Aerodynamics (in SI units). In *The Aeronautical Journal* (Vol. 116, Issue 1176).

Khan, M. A., & Sardiwal, S. K. (2013). *Design of a 2D Minimum- - - Length Supersonic Nozzle Using the Method of Characteristics*. *2*(August 2015), 19–24.

Moore, P. J. (2009). *Design of a supersonic wind tunnel. October*.

Pope, A., & Goin, K. L. (1965). *High-Speed Wind Tunnel Testing*.

Shope, F. L. (2005). *Design Optimization of Hypersonic Test Facility Nozzle Contours Using Splined Corrections ARNOLD ENGINEERING DEVELOPMENT CENTER ARNOLD AIR FORCE BASE , TENNESSEE AIR FORCE MATERIEL COMMAND*. *September 2004*.

Sivells, J.C., “A Computer Program for the Aerodynamic Design of Axisymmetric and Planar Nozzles for Supersonic and Hypersonic Wind Tunnels”, AEDC-TR-78-63, Dec. 1978.

Xia, Y. (2018). *Imperial College London Department of Aeronautics Individual Research Project Tackling inflow conditions in experiment and computation Yu Xia Supervisor : Dr Paul Bruce Submitted on*. *November*. https://doi.org/10.13140/RG.2.2.12037.40162