Design and Fabrication of Small-Scale Supersonic Wind Tunnel

Deron Hurley, Johnny Arteaga, Amir Elzawawy
Vaughn College of Aeronautics and Technology, NY, USA

Introduction

Supersonic commercial air travel is on the rise. As a result, the study of aerodynamics is becoming more important in the understanding of the forces an object experiences as it moves through air. Research in the field of supersonic technology is scarce due to the need of large facilities such as supersonic wind tunnel. Thus, creating something that would benefit the research community in the aerospace and aviation field is integral to an academic progress of the subject. Supersonic flow testing facilities are required to safely and efficiently observe structures as they undergo a flow stream faster than the speed of sound. These large-scale testing wind tunnel facilities can be interpreted as subcomponents consisting of a compression chamber, a diaphragm, convergent-divergent nozzle, testing chamber, and a flow diffuser section. A smaller scale supersonic wind tunnel facility will be advantageous for future research to understand the design and analysis of aerodynamic flow above Mach 1.

A. *Background History*

Building a new generation of aircrafts is extremely expensive and requires extensive testing. In early 20th century there were forms of supersonic wind tunnels, however they were not very effective. Environment being produced low accuracy simulating aircraft flight experience. Despite the early wind tunnels inaccuracies, the experiments they produced helped to advance the geometry of airfoils and improved our understanding of the aerodynamics loads. One of the most successfully use wind tunnels was a 30-foot by 60-foot machine manufactured by NACA in 1931. During WWII, supersonic flight was a main focus throughout the world and testing facilities were required to produce a safe aircraft. The Supersonic wind tunnel was created and required an enormous amount of power, which resulted in high pressure storage tanks as well as vacuums to create supersonic speeds. One of the smallest supersonic wind tunnels created contains a 2.543 in by 2.5 in testing chamber. Three categories of supersonic wind tunnels are, indraft, blowdown, and pressure vacuum [1]. Indraft tunnels rely on a negative pressure at the exhaust with atmosphere conditions at the inlet. Blowndown tunnels consists of a high pressure at the inlet via chambers of compressed air and atmospheric conditions at the exhaust. The pressure vacuum tunnel is a combination of both indraft and blowdown, altering both inlet and outlet. The desired use for the report focuses on a blowdown tunnel.

Experimental Setup

A. *Compression Configuration*

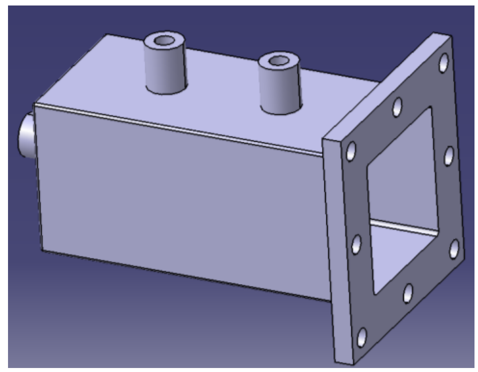

Fig. 1 Isometric view of Compression tank

By storing a significant amount of compressed air, the difference in pressure required to reach sonic flow and above is achieved. The greater the difference in pressure, the higher the Mach flow. Preliminary designs anticipated will be an aluminum cube with a minimum wall thickness of .25 of an inch that can withstand a maximum of 100 psi. The cube will have a .5-inch-thick flange to connect to the convergent-divergent nozzle and testing chamber. A unique aspect of this compression system will be a faulty diaphragm that ruptures at a specified pressure. The diaphragm is deemed desirable versus a ball valve since the flow exiting the compression system will be almost instantaneous rather than a gradual release. This is vital since flow must be distributed uniformly across the entire face of the nozzle to maintain low turbulence in the flow upstream of the testing area. A safety valve is used to release flow if pressure surpasses the desired value. A pressure gage will be fitted to monitor the rising pressure within the chamber. To take into consideration, condensation may accumulate in the chamber as the air is being compressed. Air flowing to the tank needs to be dehumidified to prevent icing. Research and calculation conducted concludes the desired pressure of the chamber to be 70 psi in order to achieve a Mach flow of 1.6. The compression configuration is an original design and necessary to prevent turbulent flow. Eq. 1 represents the pressure ratio of the ambient atmospheric temperature to the pressure the compression configuration will hold in order to achieve Mach flow [2].

$$\frac{P_0}{P^*} = \left(\frac{T_0}{T^*}\right)^{\frac{\gamma}{\gamma-1}} \quad (1)$$

Where:
P_0 = Total Pressure
P^* = Desired Pressure
T_0 = Total Temperature
T^* = Desired Temperature
γ = Specific Heat ratio

B. Mylar Diaphragm

Rupturing a diaphragm between the compression section and convergent nozzle is integral as it allows for a more uniform flow. The diaphragm will consist of sheets of mylar. The quantity will be determined through trials until a specified pressure is reached. The mylar will be effectively sealed using a high-density gasket. The compression configuration is fastened to the nozzle with material capable of withstanding and securing low to high pressure loads. Once diaphragm ruptures it will generate the flow instantaneously. If the diaphragm is not utilized, the air from the compressor will gradually flow into the nozzle and will not increase the flow speed to the desired supersonic level. The use of a diaphragm will allow to obtain the desired pressure ratio that is suddenly felt by the flow once it ruptures. This change in flow pressure required for supersonic speed can be achieved.

C. Convergent-Divergent Nozzle Geometry

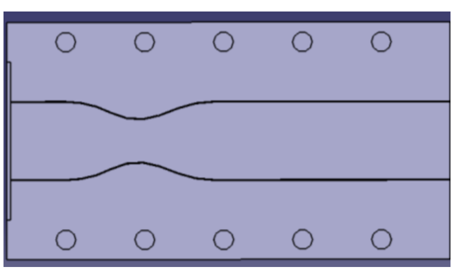

Fig. 2 C-D Nozzle Geometry

A convergent-divergent nozzle is a specially shaped tube designed to speed up the flow of the air and thrust it through the testing chamber. Both sections of the nozzle will be fixed, making the design of the nozzle vital with no room for error. The convergent section will increase the velocity of the compressed air to a sonic flow.

$$\left(\frac{A}{A^*}\right)^2 = \frac{1}{M^2}\left[\frac{2}{\gamma+1}\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{\gamma-1}} \quad (2)$$

Where:
A = Exit Area
A^* = Throat Area
M = Mach Number
γ = Specific Heat ratio

The calculation required to gain the ratio desired to speed up the flow to a desired speed can be seen by Eq.2 [2]. The ratio seemed minimal in an effort to achieve a high Mach flow, and after simulation and consideration of friction due to boundary layer flow, the area ratio was increased by approximately 25% to achieve better flow [6]. Considering the flow to be sonic at the throat, the divergent duct will increase the speed to a supersonic flow. Initial design based on isentropic relations produced flow that is subsonic at the throat where the flow velocity decreased in the divergent duct section. These affects can be seen using Eq.3 which relates the area *(dA)* and velocity *(du)*. A large difference in pressure from the compression system and testing chamber is used to achieve the sonic flow at the throat. Designing the nozzle can be tedious as any sharp corners or angles can cause disturbances in the flow and will diffuse the velocity too soon. From research conducted, with the desired Mach flow of 1.8, the area ratio between the throat and testing chamber must be 1.439.

$$\frac{dA}{A} = (M2-1)\frac{du}{u} \quad (3)$$

Where:
dA = Change in area
A = Area
du = Change in velocity
u = Velocity
M = Mach number

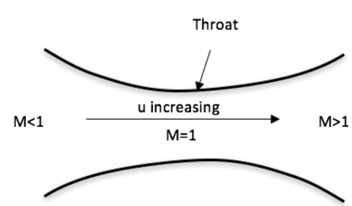

Fig. 3 Convergent-Divergent Nozzle Geometry

D. Testing Chamber

This section is where the flow has to have the highest Mach value. Design intent is to have an oblique shockwave occurs on the object being investigated so that the wave reflections produced will be at an angle that doesn't reflect on the object to simulate real flight scenario where there are no wind tunnel walls. If a slight disturbance takes place at some point in the air, information is transmitted to other points by sound waves which propagate in all directions away from the source of the disturbance. The gas molecules which impinge into the body surface experiences a change in momentum. In turn, this change is transmitted to neighboring molecules by random molecular collisions [2]. Oblique shockwaves, and compression waves are prevalent in two- and three- dimensional supersonic flows. The angle that the Mach wave makes with respect to the direction of motion of the material is defined as the Mach angle μ.

$$\frac{T_0}{T^*} = 1 + \frac{\gamma - 1}{2}(M)^2 \quad (4)$$

Where:
T_0 = Total Temperature
T^* = Desired Temperature
M = Mach number
γ = Specific Heat ratio

The extreme temperature being experienced in the chamber will be -125.15 degrees Celsius according to Eq. 4, this is important due to the critical points of the air [2]. This calculation was based off of a room temperature of 70 degrees Fahrenheit (21.1 degrees Celsius). Nitrogen makes up 78.09 % of the air, Oxygen holds 20.95 %, keeping this is mind we must take into consideration the critical points of these gasses. Nitrogen becomes a liquid at -196 degrees Celsius and Oxygen turns to liquid at -183 degrees Celsius [4]. The air being compressed will not reach a critical point to become liquefied. This is integral due to the fact that, the air will not be detrimental to the system at our required compression temperature and Mach flow temperature.

E. Flow Diffuser Section

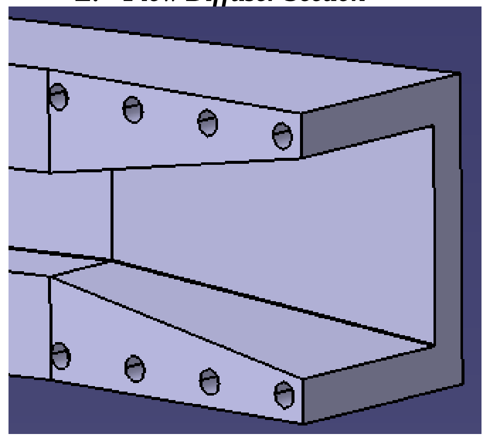

Fig. 4 Diffuser Geometry

The diffuser section role is to slow the airflow to a desired low speed value. Successfully done, this will achieve as small a loss in total pressure as possible. Flow enters the diffuser at supersonic speeds and due to its isentropic nature, the pressure is constant throughout the entire diffuser. It is extremely difficult to slow a supersonic flow without generating shock waves in the process. In real life, the flow is viscous; there will be an entropy increase within the boundary layers on the walls of the diffuser. As realized by the flow simulation using COMSOL CFD module, this boundary layer flow creates separation, in some cases, that narrows the area before entering the diffuser, acting like a supersonic diffuser. The behavior of the flow determines whether the geometry can be depicted as a nozzle or a diffuser.

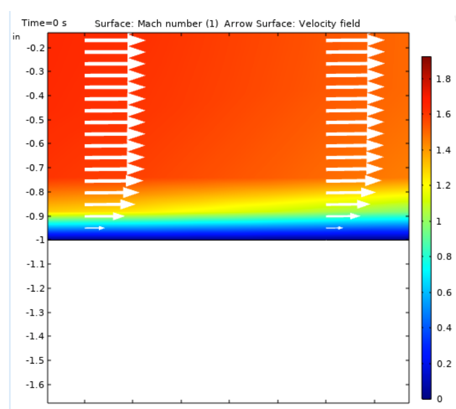

Fig. 5 Boundary Layer Flow

For these reasons, the flow must be slowed and redirected generating a series of expansion waves. Once exiting the testing chamber, the flow encounters a rapid convex wall angle of 9.5 degrees. This angle is integral in order to allow the expansion waves to occur for the incoming flow. As the walls are deflected, the streamlines at the wall is also deflected downward through the angle θ. The bulk of the gas is above the wall, and the streamlines are turned downward, away from the main bulk of the flow. As the flow is turned away from itself, an expansion wave will occur as seen from the simulation. The original horizontal streamlines ahead of the expansion wave are deflected smoothly and continuously through the expansion fan such that the streamlines behind the wave are parallel to each other and inclined downward at the deflection angle θ [2].

$$\theta = \upsilon(M2) - \upsilon(M1) \quad (5)$$

Where:
θ = deflection angle
$\upsilon(M2)$ = Prandtl Meyer Function after flow
$\upsilon(M1)$ = Prandtl Meyer Function before flow

Eq.5 above represents the deflection angle of the geometry, where υ is the Prandtl-Meyer angle based on the flow Mach value before or after the wave [2]. Across the expansion wave, the Mach number increases, and the pressure, temperature, and density decrease. Hence, expansion waves are the direct antithesis of a shock wave. Due to the wide angle being utilized, the flow separated and began to slow down.

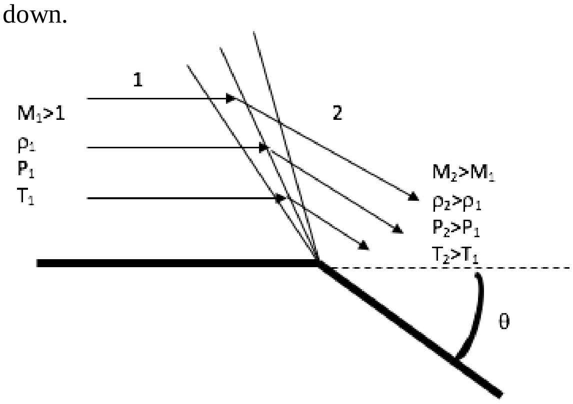

Fig. 6 Visual of Expansion Wave

The test section viewing window will be made of plexi glass and will be flushed with the inside of the tunnel to prevent any disturbances in the flow. The testing section will be a 1” by 1” cross sectional area. It will be secured with a flange and gasket as well.

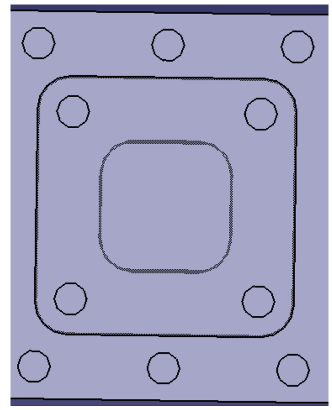
Fig. 7 Viewing Window

Flow Simulation and analysis

In order to achieve the correct geometry necessary to achieve Mach flow and above, a simulation was required to verify our analytical calculations. The utilization of COMSOL-high Mach number flow module was vital for determining the Mach flow received within the C-D Nozzle, Testing Chamber, and understanding how the flow will diffuse [3]. The software package includes a form of CAD environment interface that allows the user to directly import a form of 3D geometry created with CAD. With this in mind, the negative space of the wind tunnel was required to conduct testing and geometry manipulation, in order to physically visualize the motion of the flow, and its characteristics. Boolean and partition waere conducted to subtract the remaining geometry of the wind tunnel, so the user is left with the negative space alone. After the import of the CAD model to the design modeler, a mesh of elements was applied in order to gain more accurate results for specific areas of the geometry. The inlet, outlet, and wall boundary conditions were then created in order to specify certain characteristics of the flow. Using the negative space of the wind tunnel a COMSOL simulation can be created based on a change in Pressure. The inlet was set to 70 psi while the exhaust is set to 1 atm. Supersonic flow was achieved past the throat of the C-D Nozzle, due to the fact that the conditions were not sonic at the throat, supersonic flow was not continuous into the testing chamber. An analysis on the size of the inlet was executed, producing the same result in the testing chamber however, the narrower area had a greater incoming speed than that of the larger inlet area. Using oblique shockwaves to reduce the velocity of the air results in a narrow exhaust causing the air speed to increase without a decrease at the exhaust. The undesirable results of initial diffuser section design, which is due to boundary layer separation as discussed previously, prompted the use of a diversion duct that produces expansion waves in order to better diffuse the flow. Noticing the negative results in the diffusion, the inlet pressure was decreased due to a direct relationship between the pressure and velocity. This resulted in a more effective diffusion.

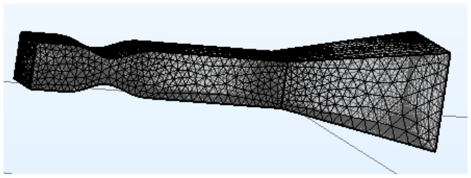

Fig. 8 Visual of Meshed CAD Model

COMSOL analyses were further studied in a mesh conversion analysis. The meshes ranged from 78,713 elements to a finer mesh with 1,613,751 elements. Each analysis provided more accurate results maintaining the same successful outcome.

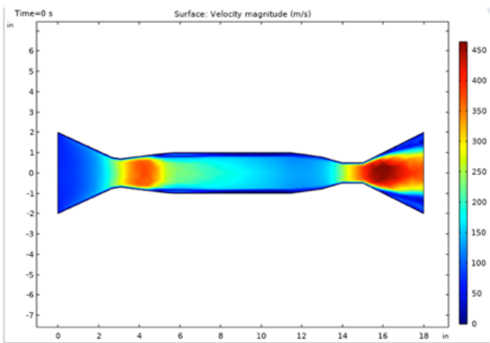

Fig. 9 Initial Flow Simulation

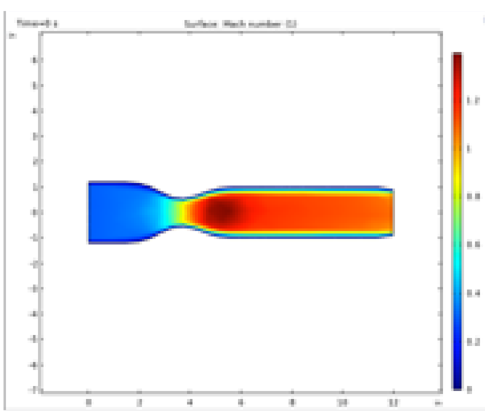

Fig. 10 Convergent Diffuser

The convergent diffuser was based on analytical theory. The application of utilizing a converging diffuser should allow the flow to diffuse with an oblique shockwave. However, this geometry configuration did not produce the results desired.

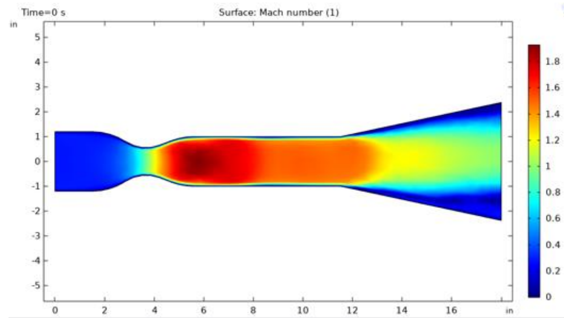

Fig. 11 Redesigned Flow Simulation

A. *Aerodynamic loads*

In order to manufacture and test a safe tunnel structure, the forces due to pressure and shear were integrated in the streamwise direction. The three sources of force are, the net momentum produced by the inlet and outlet flow, the net pressure, and the total shear inside the surface. The net momentum was calculated using a surface integration applied to the inlet and the outlet.

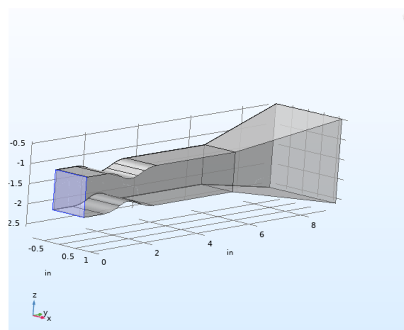

Fig. 12 Inlet Force Due to Pressure

Using the surface integration over the inlet area, a maximum force of 198.27 N or 44.57 lbf was produced due to pressure.

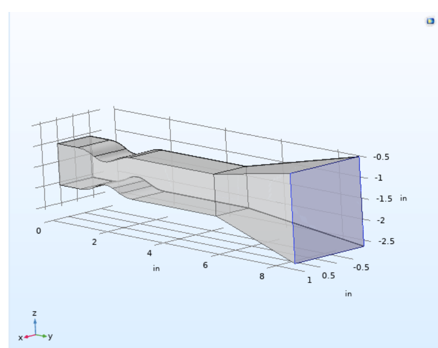

Fig. 13 Outlet Force Due to Pressure

The maximum outlet force produced was 296.76 N or 66.71 lbf.

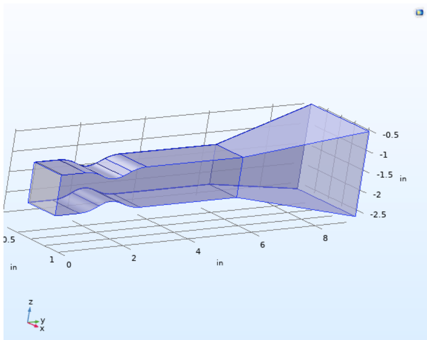

Fig. 14 Shear Force

The entire surface is experiencing a maximum shear force of 230 N or 51.71 lbf. In order to secure each section of the tunnel, steel zinc coated bolts are utilized to fasten the parts together. They are grade 5 rating, with a proof load of 85,000 psi [5]. The total force each section experiences is distributed based on the number of bolts. The compression chamber utilizes 8 bolts, each experiencing 5.57 lbf. The viewing window will experience 3.63 lbf based on pressure, producing 0.9075 lbf per bolt with a 4-bolt configuration.

Flow Simulation Results

Initially, a geometry selection was chosen based solely off analytical calculations, without the considerations of viscous effect due to boundary layer flow, the inlet size of the compression chamber attached to the C-D nozzle, length of the testing chamber and diffuser, and the angle and shape of the diffuser. All of these traits have a direct relation in the performance of the flow and its diffusion. The selection of inlet pressure was too large to effectively diffuse the flow within a short range, resulting in a pressure of less magnitude. The curvature of the C-D nozzle originally contained sharp corners filleted edges. This geometry failed to produce a smooth transition of flow to the testing chamber. With this in mind, the geometry contours were redesigned to obtain less turbulent characteristics and better streamlines. The direction of the diffuser in the Y and Z direction created better results, considering an angle of 19 degrees in all directions. After extensive analysis and simulation, a final geometry was chosen. The utilization of divergent duct that produces expansion waves was a key factor in producing diffusion results. Utilizing the geometry that produced the highest flow in the chamber, with the best diffusion was a building block to the final product. With an alteration in the incoming pressure, we began to obtain the desired results. The highest Mach value of 1.929 was obtained in the testing chamber and a diffusion result of subsonic at 0.25 in the exhaust. To ensure our results were accurate a mesh conversion study was performed, and a comparison of the results were obtained. With a finer mesh quality and smaller tetrahedral elements, results were altered, a higher Mach values were obtained in the chamber with even lower flow temperature than anticipated. While still receiving subsonic flow in the exhaust. The flow becomes steady with no alteration due to time at .05 of a second. The fine mesh produced a Mach value of 1.927 at the testing chamber and 0.25 at the exhaust. Meanwhile the finer mesh produced a Mach value of 1.929 at the testing section and 0.25 At the exhaust. The difference in values reflects the number of elements used in each analysis yielding more accurate results. In order to finalize the results to ensure complete accuracy, an aluminum material was applied to the model and another study was done utilizing a normal mesh size, this produced a Mach value of 1.82 in the testing chamber and diffusing at 0.25 in the exhaust.

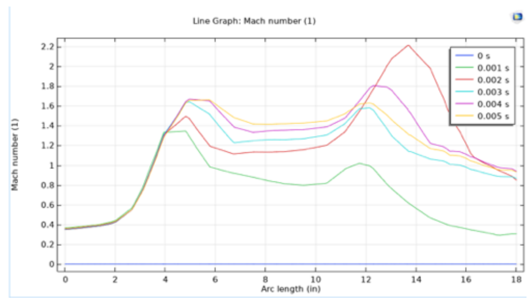

Fig. 15 Mach Number vs. Length

Fig. 17 represents the Mach value with respect to the location in the flow direction each length of the overall tunnel. The different lines represents different time instances.

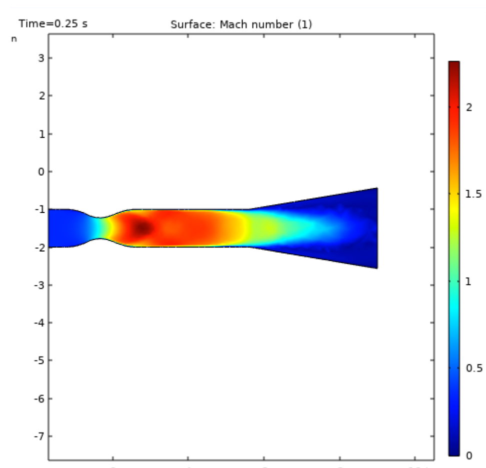

Fig. 16 Final Simulation

With the necessary dimensions of the materials and parts being considered, a final CAD design was developed and produced for further fabrication. The CAD design takes into consideration spacing and stress dispersion based on FEA results.

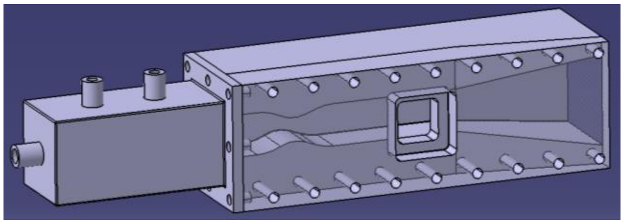

Fig. 17 Final CAD Design

Conclusion

A final geometry configuration is designed and selected for the fabrication of the Small-Scale Supersonic Wind Tunnel. The analysis of the Supersonic Wind tunnel is conducted using COMSOL Multiphysics, along with analytical calculation for the geometry configuration, and a mesh conversion study to produce more accurate results. An inlet pressure of 70 psi produces supersonic flow at speeds of Mach 1.82 in the testing chamber while diffusing the flow to subsonic at Mach 0.25 in the exhaust. Flow loads are identified to verify loading conditions. Manufacturing and fabrication will be done to produce a model fit for testing and analysis of aerospace structures and components.

References

[1] Chang, Wilbur. Design and development of a rectangular supersonic wind tunnel facility for the study of shock/boundary layer interactions.2011, pp. 1–140

[2] Anderson, John D. Fundamentals of Aerodynamics. McGraw Hill Education, 2017.

[3] “Application Gallery.” COMSOL, www.comsol.com/models.

[4] Diep, Francie. “Why Having A Liquid Nitrogen Pool Party Is A Bad Idea.” Popular Science, 20 June 2013, www.popsci.com/science/article/2013-06/why-having-liquid-nitrogen-pool-party-bad-idea.

[5] “About Fastener Materials.” Bolt Depot - Selecting Fastener Materials - Steel Grades, Brass, Bronze, Stainless Steel, www.boltdepot.com/fastener-information/materials-and-grades/materials.aspx.

[6] K. Butler, D. Cancel, B. Earley, S. Morin, E. Morrison, M. Sangenario (2010). Design and Construction of a Supersonic Wind Tunnel- WPI

[7] “Atmosphere of Earth.” Wikipedia, Wikimedia Foundation, 2 Mar. 2019, en.wikipedia.org/wiki/Atmosphere_of_Earth.

[8] NASA Online Resource (2019) – https://www.grc.nasa.gov/www/k-12/airplane/tunnozd.html

[9] Bhavin K Bharath, Design and Fabrication of a Supersonic Wind Tunnel (IJEAS), ISSN: 2394-3661, Volume-2, Issue-5, May 2015

[10] F. Ommi, V. Farajpour Khanaposhtani, M. Agha Seyed Mirzabozorg, K. Nekoufar, A New Approach for Supersonic Diffuser Design (Journal of Applied Sciences Research, 6(5): 401-414, 2010)

[11] *ASM Material Data Sheet*, asm.matweb.com/search/SpecificMaterial.asp?bassnum=ma6061t6.

[12] Frank M. White. (2009). Fluid Mechanics 7th edition. New York: McGraw-Hill.

[13] “Non-Asbestos Gasket Materials.” Ram Gasket Solutions Ltd, 16 Nov. 2017, www.ramgaskets.com/gasket-materials/non-asbestos-gasket-materials/.

[14] “Understanding Gasket Pressure.” Ram Gasket Solutions Ltd, www.ramgaskets.com/help/understanding-gasket-pressure/.