Converging - Diverging Nozzle Flow Analysis (Project)

Author: Rushi Faldu, Liam Sajnacki, Calvin Saia, Brock Sandford
Aerospace Engineering Undergraduate, Raleigh, NC, 27607

This report indicates the study of CD nozzle flow conducted at a lab at NCSU university. The primary objectives of the lab were to desing a CD nozzle, perform CFD simulations using the design, 3D print the CD nozzle, and understand the procedure to measure flow conditions within the nozzle. The lab utilized a pressure tank connected to a CD nozzle and a pitot tube to record data both analytically and experimentally for different flow conditions within the nozzle. The lab was successful in its primary objectives. However, the recorded data shows significant errors making the temperature graph unusable in the analysis of the conditions due to a malfunction in the temperature sensors while performing the experiment. There were some errors in the CFD simulation as well as due to the assumption of isentropic flow within the nozzle. Despite the errors, the designed CD nozzle was able to achieve supersonic flow conditions during the experiment.

Nomenclature

a	=	speed of sound
A	=	cross-sectional area
A^*	=	cross-sectional area at choke point
P_O	=	stagnation pressure
P	=	static pressure
P^*	=	pressure at choke point
P_B	=	back pressure
T_O	=	stagnation temperature
T	=	static temperature
T^*	=	temperature at choke point
T_B	=	back temperature
V	=	velocity
$\dot{m}$	=	mass flow rate
M	=	Mach number
R	=	ideal gas constant
ρ	=	density
ρ_O	=	stagnation density
γ	=	specific heat ratio
CD	=	converging - diverging

I. Introduction

CD nozzles are used everywhere in the world of aerospace. ramjets, scramjets, and rockets all use CD nozzles to accelerate the hot gases outside the engines to produce thrust for aircrafts and rockets. CD nozzles are known as converging diverging nozzles. The structure of a CD nozzle is shown in figure (1) below. A CD nozzle consists of 3 sections which are a converging section, a diverging section, and throat. In jet engines, when the hot exhaust leaves the combustion chamber, it goes through the converging section of the CD nozzle that converges the exhaust down to a minimum area known as the throat of the nozzle. The throat of the nozzle is set up in such a way that it chokes the flow that passes through it. The exhaust then comes to the diverging section that has a larger area which expands the flow to a supersonic speed generating enormous thrust for the engine.

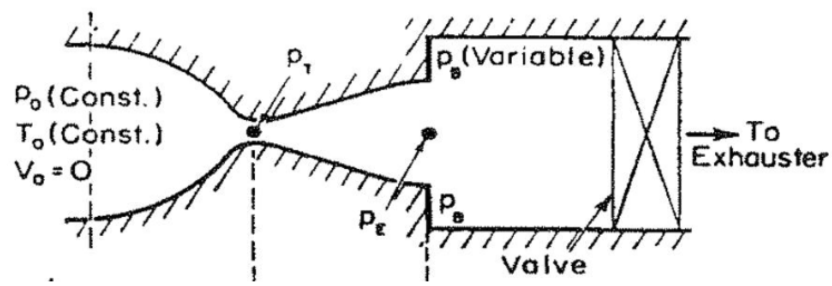


Figure (1): A generalized diagram of a converging nozzle

The objective of this experiment was to design, simulate, and analyse the performance of a converging-diverging (CD) nozzle using experimental, analytical, and computational approaches. The project provided hands-on experience in compressible flow analysis, numerical simulation, and experimental aerodynamics by integrating multiple components of engineering practice, design, manufacturing, testing, and post-processing. A CD nozzle was designed within specified geometric and operational constraints.

The experiment aimed to deepen understanding of compressible flow dynamics in nozzles by observing flow acceleration from subsonic to supersonic conditions. Analytical predictions of flow properties were obtained using isentropic relations under the assumption of choked flow. These predictions were then validated through CFD simulations in ANSYS Fluent and experimental measurements taken using a Pitot-static tube traversed along the centerline of a 3D-printed nozzle in NCSU's supersonic wind tunnel.

A key component of the analysis was to compare the Mach number and temperature distributions from the analytical, numerical, and experimental methods. The experiment also required careful calibration and application of transducers to convert voltage readings into physical quantities, with all results presented in SI units. Ultimately, the project emphasized the convergence of theory, simulation, and experimentation in compressible aerodynamics.

II. Methodology

The data for this lab will be collected through the below mentioned procedure.

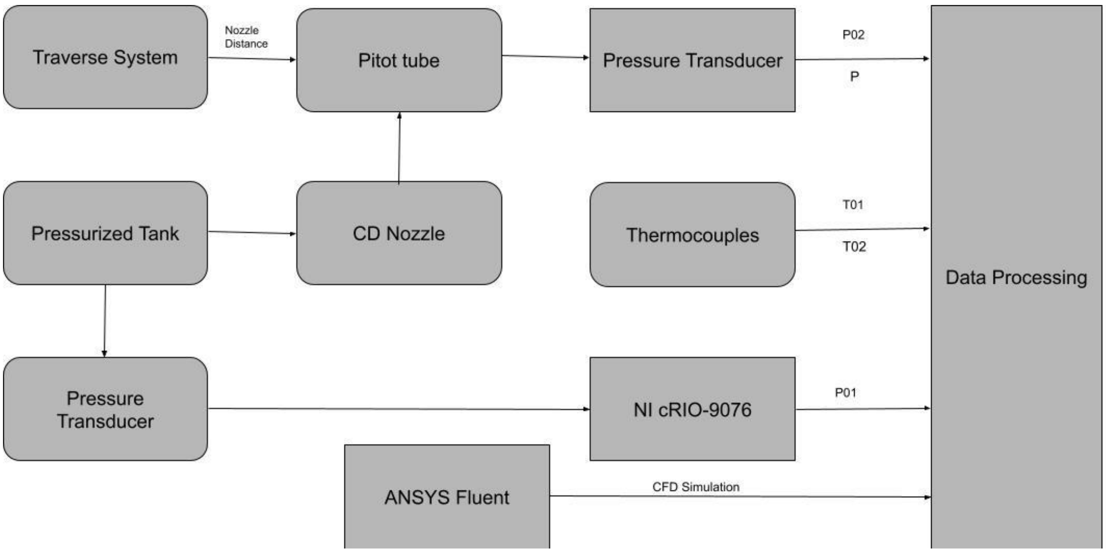


Figure (2): Experimental Flow Diagram for this lab

Before starting with the experiment, a CD nozzle was designed. The figure shown below highlights the design constraints taken into account when designing the nozzle. In addition to that, to reduce the blockage effect of the pitot tube, the ratio of the pitot tube cross section to the nozzle throat area was kept less than 10%. The flange shown in the design was used for nozzle installation.

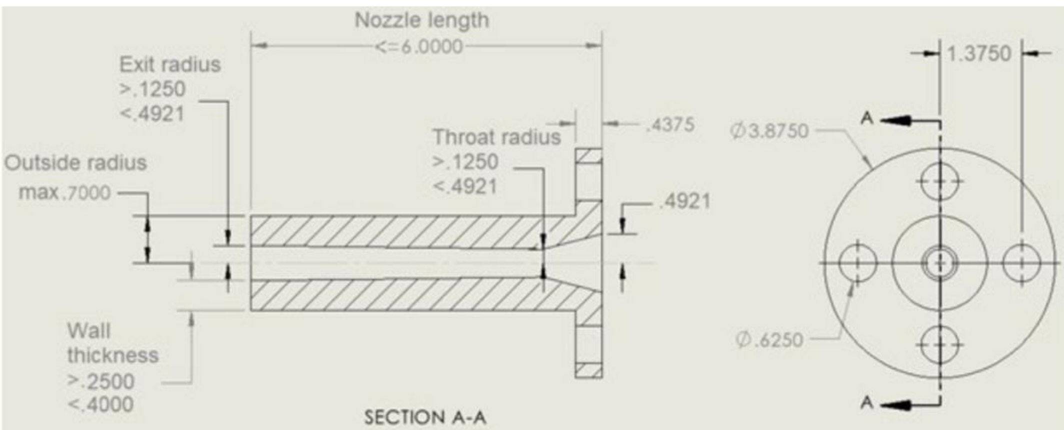


Figure (3): Design Constraints of the CD nozzle

Lastly, a few dimensions were selected by the team. Given the inlet radius of 12.5 mm, a throat radius of 6 mm, inlet to throat distance of 25 mm, and an exit radius of 10 mm were selected.

Lastly before designing the CD nozzle in Solidworks, a CFD simulation of the CD nozzle was performed in ANSYS. Firstly, in ANSYS a 2D cross-section of the nozzle with a rectangular meshing was created, as shown in figure (4), following the geometric design listed in figure (3) and then uploaded to ANSYS Fluent for CFD simulations. The simulation was ran as linear flow with a supplied pressure of 275790 Pa and the Mach number at each element. The data from CFD analysis will be used in plotting graphs and analysing.

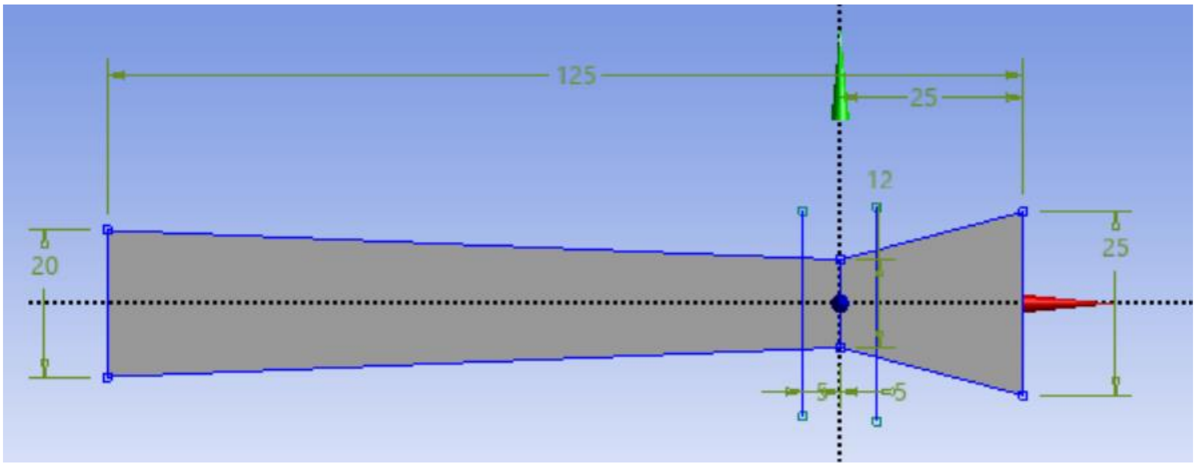


Figure (4): ANSYS drawing of nozzle cross-sectional area

Afterwards, a SolidWorks design of the nozzle was created to 3D print the nozzle. The figure below shows the 3D design created in SolidWorks. The dimensions in the drawing are listed in inches. After designing the nozzle in SolidWorks, it was 3D printed with 60% infill following which the acutal experiment was conducted.

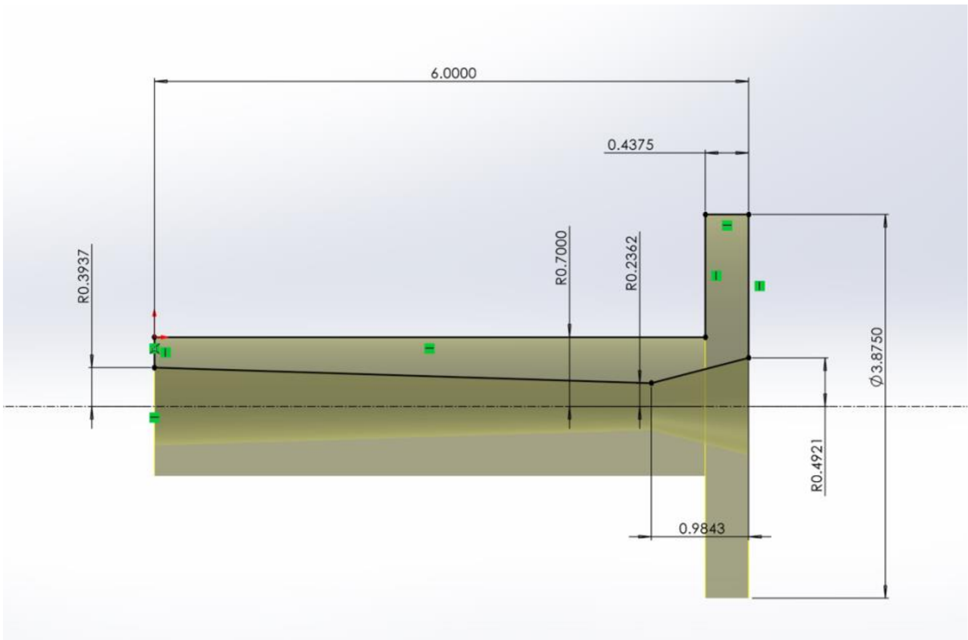


Figure (5): SolidWorks 3D design of the CD nozzle

The experiment setup was like the experimental flow diagram shown in figure (2). Using the NI cRIO-9076, back pressure was measured. The collected data on the back pressure went through a data acquisition system to the data processor. For this lab, the P_B = 14.7 psi.

After measuring the back pressure, the equipment for the experiment was setup properly. First, the CD nozzle was attached to the pressure tank. In front of the nozzle, a traverse system was placed with an accurate scale from which the position of the pitot tube was fed to the data processor. The pitot tube along with static and temperature tube was mounted on the traverse system allowing it to move back and forth freely. Both the pitot tube and pressure tank were then connected to pressure transducers and temperature sensors to measure the stagnation pressure and temperature along with pitot static pressure. All of this data was then fed to the data processor during the experiment. The tank was then pressurized to 40 psi and the experiment started as the airflow was released from the pressure tank through the CD nozzle.

The back temperature and pressure are the primary factors that drive the flow and determine the flow conditions inside the CD nozzle. When the $P_O = P_B$, there is no airflow through the nozzle. For a CD nozzle, as the P_B is reduced, the Mach number at the throat of the nozzle will keep on rising until the flow is choked. Choked flow conditions for pressure and temperature can be calculated using the following formulae:

$$\frac{P^*}{P_O} = 0.5283 \qquad \textbf{(1)}$$

$$\frac{T^*}{T_O} = 0.8333 \qquad \textbf{(2)}$$

For this lab, the stagnation temperature was 300 K. These will then be used in the isentropic equations to find the pressure and temperature inside the CD nozzle analytically. However, as all isentropic relations are functions of γ and M, they need to be found first. γ for air is 1.4. Mach number on the other hand can be found both analytically and experimentally using isentropic equations. To find the Mach number analytically, the following equation is used:

$$\frac{A}{A^*} = \left(\frac{\gamma+1}{2}\right)^{-\frac{\gamma+1}{2(\gamma-1)}} * \frac{\left(1+\frac{\gamma-1}{2}*M^2\right)^{\frac{\gamma+1}{2(\gamma-1)}}}{M} \qquad \textbf{(3)}$$

Using Matlab, this equation is solved numerically to find the analytical values of Mach number. However, to solve for Mach number, A and A^* need to be found. Those can be calculated through the CD nozzle geometry provided in figure (3) and figure (5).

The CD nozzle is divided into 2 sections as seen in the drawings, converging and diverging. Then each section is divided into a number of equal parts to find the Mach number at each location along the nozzle. For this lab, the nozzle is divided into increments of 6.25 mm from 0 to 25mm and increments of 25 mm from 25 mm to 125 mm. The analytical Mach number is then used to find other flow conditions within the CD nozzle. Static temperature, pressure, and density are found using gamma and Mach number through isentropic equations.

$$\frac{P}{P_O} = \left(1+\frac{\gamma-1}{2}*M^2\right)^{-\frac{\gamma}{\gamma-1}} \qquad \textbf{(4)}$$

$$\frac{T}{T_O} = \left(1+\frac{\gamma-1}{2}*M^2\right)^{-1} \qquad \textbf{(5)}$$

$$\frac{\rho}{\rho_O} = \left(1+\frac{\gamma-1}{2}*M^2\right)^{-\frac{1}{\gamma-1}} \qquad \textbf{(6)}$$

The stagnation density is found through the use of ideal gas law. Together all of these values help determine the conditions in the CD nozzle at every interval. Finally, velocity and mass flow rate are calculated using the following equations for each interval within the nozzle:

$$a = \sqrt{\gamma R T} \qquad \textbf{(7)}$$

$$V = Ma \qquad \textbf{(8)}$$

(9)

$$\dot{m} = \rho A V$$

In these equations, 'a' stands for the speed of sound and A stands for the corresponding cross-sectional area of the nozzle at a particular interval. The experiment was conducted one time for each interval to measure the different conditions at that point in the CD nozzle. Then, after finding the analytical values, experimental values for Mach number can also be calculated using the same isentropic equations through the data derived in the experiments. However, the isentropic pressure equation (Eq. 4) can only be used to calculate Mach number below 1. As the Mach number goes beyond 1 after the throat of the nozzle, it is calculated using the Rayleigh Pitot tube formula. The equation as shown below is solved numerically in MATLAB for accurate results.

$$\frac{P}{P_O} = \frac{\gamma + 1}{\gamma - 1 + 2\gamma M^2}\left[\frac{(\gamma+1)^2 M^2}{4\gamma M^2 - 2(\gamma - 1)}\right]^{-\frac{\gamma}{\gamma-1}} \quad (M > 1) \tag{10}$$

III. Results & Discussion

To better compare and contrast the results, the analytical data is first displayed in Table (1), as shown below.

R (J/K kg)	gamma	P0 (Pa)	T0 (K)	Stagnation Density (kg/m^3					
287	1.4	900550	300	10.4593					
Point	**Location (mm)**	**Area (mm^2)**	**Pressure (Pa)**	**Temperature (K)**	**Density (kg/m^3**	**Velocity (m/s)**	**Mach Number**	**Mass Flow (kg/s)**	**Note**
1	0	490.874	889188	298.914	10.3651	46.7162	0.1348	0.23769	Inlet
2	6.25	371.542	880508	298.077	10.2927	62.1549	0.1796	0.23769	
3	12.5	268.803	861335	296.208	10.1321	87.2818	0.253	0.23772	
4	18.75	182.654	809806	291.033	9.69534	134.22	0.3925	0.23769	
5	25	113.097	475744	250	6.63068	346.074	1	0.25953	Throat
6	50	153.938	207953	188.114	3.25656	594.931	1.7245	0.29824	
7	75	201.062	151054	162.144	2.24627	705.054	2.0618	0.31843	
8	100	254.469	120698	143.947	1.66806	737.896	2.3282	0.31322	
9	125	314.159	101324	130.077	1.29483	702.627	2.5557	0.28582	Exit

Table (1): Analytically calculated flow conditions

The above mentioned table (1) shows the isentropic calculations for the geometry of the nozzle. It can be seen that the air reaches a Mach 1 status at the throat, afterwards as it goes through the divergent section, the Mach number increases to supersonic from 1 to almost 2.55 at the exit of the nozzle. A decrease in pressure, density, and temperature is noticed along the nozzle sections. An increase in velocity is seen as the Mach number of the airflow increases throughout the nozzle.

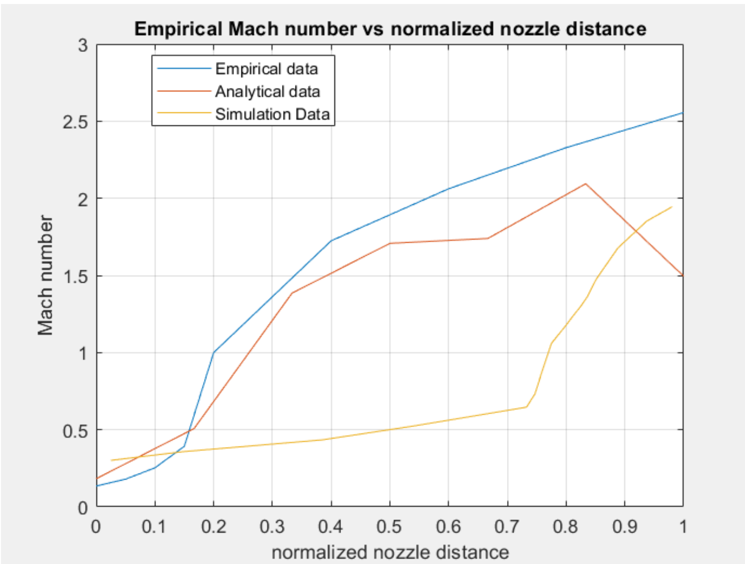


Figure (6): Comparison of analytical, experimental, and CDF simulation Mach number values along nozzle centerline

In the above graph, the mach numbers are plotted against normalized nozzle distance. The Empirical data and analytical data are close, only varying significantly at 80% down the nozzles length. This is likely due to a shock occurring inside the nozzle diverging section, which causes a jump in Mach numbers and properties, overall slowing down the flow. This is not seen in the analytical data as that data was calculated assuming isentropic conditions throughout the nozzle, thus shock wave was not considered. The Simulation data shows significant differences throughout the test until the end. Possible reasons for this are discussed in the error section.

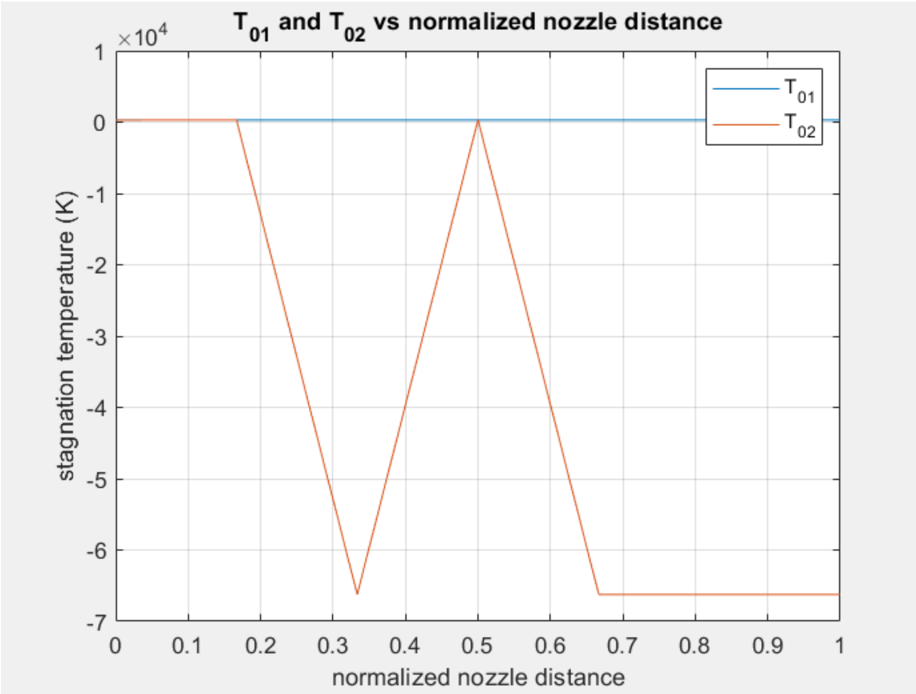


Figure (7): Stagnation temperatures for tank and pitot tube along the nozzle

In the above figure, the stagnation temperature in the pitot tube and tank are plotted. Significant errors are present in this data. Instead of showing a positive temperature, the tank and pitot tube temperatures are 0 and negative. Furthermore, the T_{o2} appears to fluctuate back and forth going a few Kelvin below zero. The possible reasons for the same are discussed below.

Regarding the error within the tabulated results, the temperature sensing device was malfunctioning on the day of the test and presented highly unrealistic values. This played a massive role in skewing the stagnation temperature vs normalized nozzle distance presented in figure (7). Furthermore, likely sources of error present could've occurred as a result of the nozzle construction material and method. The nozzle was 3-D printed utilizing a plastic material, which within nozzle left an unsmooth surface presenting minor ridges and small flared pieces of plastic, which could've played a role in the presentation of data or the formation of a weak shock in the divergent section of the nozzle mentioned previously. The weak shock was not taken into account due to the isentropic assumption made during the experiment and the data calculations. Additionally, the intrusion of the pitot tube into the nozzle could've affected the flow through the nozzle. Despite trying to reduce the pitot tube blocking the flow, a significant amount of flow could have been blocked resulting in data with errors. The CFD simulation data is also significantly different from analytical and experiemtental data. The likely cause of this could be an improper setting of conditions within the software cuasing a difference in data. It could be due to not assuming enough initial conditions. Although these sources of error were present, all of the results tabulated except for figure (7) were to be expected, and displayed accurate information regarding the flow conditions within the nozzle.

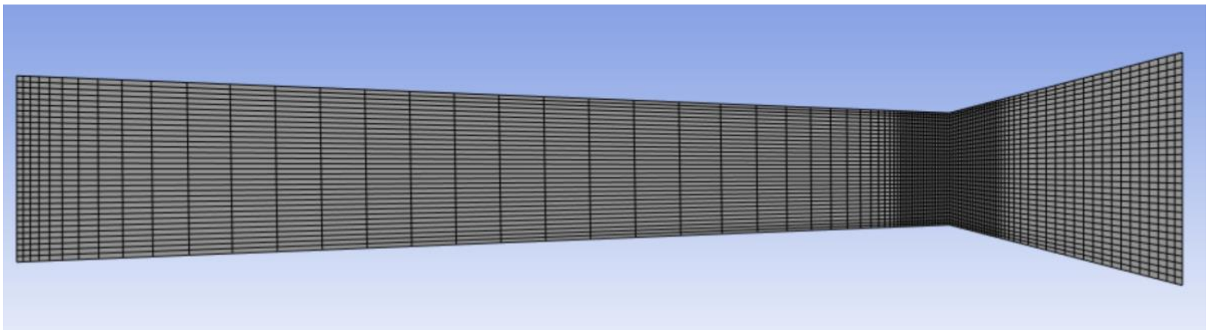
IV. Conclusion

In conclusion, this lab highlighted the importance of CD nozzles and why understanding the flow through CD nozzles is important in the field of aerospace. Apart from that, the lab helped understand the desining process, constraints, CFD simulation, SolidWorks software, testing procedure and the equipment used to measure flow conditions through a CD nozzle. The results presented in the analysis of our converging-diverging nozzle delivered reasonable results that were expected from such a nozzle geometry. With the exception of malfunctioning or ill-prepared lab equipment, the results produced are accurate and display the flow conditions of our constructed nozzle at various points within it. Lastly, additional sources of error may have resulted from the nozzle construction method and material, contributing to the formation of a weak shock in the divergent section of the nozzle and thereby reducing the Mach number at the exit. However, although these conditions were present, the nozzle was able to produce supersonic flow with the given initial conditions.

V. Appendix

I. Picture of the CD Nozzle

II. Computational Mesh of the CD nozzle



III. MATLAB Code

```
clear
clc
close all
R = 287;
```

```
Pstag = 900550;
Tstag = 300;
rho = Pstag/(R*Tstag);
one = readmatrix("206_B1.xls.xlsx");
local = transpose(one(:,3)/39.37);
tankP01 = transpose(one(:,4)*6894.76);
P02 = transpose(one(:,5)*6894.76);
Ps = transpose(one(:,6)*6894.76);
tankT01 = transpose(one(:,7)+273.15);
PT2 = transpose(one(:,8)+273.15);
local = [0,6.25,12.5,18.75,25,50,75,100,125];
for i = 1:5
   area(i) =pi*((-0.26*(local(i)))+12.5)^2;
end
for i = 6:9
area(i) =pi*((.04*(local(i)))+5)^2;
end
arearatio = area./area(5);
for i = 1:9
   syms M
   eqn = (.5787037037)*(((1+.2*M^2)^3)/(M)) == arearatio(i);
   sol = solve(eqn,M);
   Mach(i) = sol(1);
end
for i = 6:9
   syms M
   eqn = (.5787037037)*(((1+.2*M^2)^3)/(M)) == arearatio(i);
   sol = solve(eqn,M);
   for j = 1:length(sol)
     if isreal(double(sol(j))) && double(sol(j)) > 1
           Mach(i) = sol(j);
    end
   end
end
%stag pressure calc
% p0 = 101325/(  (2.4/(-.4+2.8*Mach(9)^2))*(
(5.76*Mach(9)^2)/(5.6*Mach(9)^2-.8) )^(-3.5)                             )
% p02 = 101325/((1+(.2*Mach(9)^2))^(-3.5))
pressureratio = flip( (Ps./P02) );
for i = 1:2
   syms M
   eqn = (1+(.2*(M^2)))^(-3.5) == pressureratio(i);
   sol = solve(eqn,M);
   for j = 1:length(sol)
       if isreal(sol(j)) && sol(j) > 0
           filtered_value = sol(j);

       end
   end
   MachE(i) = filtered_value
end
for i = 3:7
   syms M
  eqn = (2.4/(-.4+2.8*M^2))*(((5.76*M^2)/((5.6*M^2)-.8)))^(-3.5) ==
pressureratio(i);
   sol = solve(eqn,M);
   for j = 1:length(sol)
       if isreal(double(sol(j))) && double(sol(j)) > 1
           filtered_value = sol(j);

       end
```

```
    end
    MachE(i) = filtered_value
end
filename = 'MachData.xlsx';
opts = detectImportOptions(filename, 'NumHeaderLines', 0);
rawData = readcell(filename, opts);
% Convert the first column (which contains comma-separated strings) into
strings
rawStrings = string(rawData(:,1));
% Split each row by commas
splitData = split(rawStrings, ',');
% Remove header row
splitData(1,:) = [];
% Convert strings to numeric values
dataNumeric = double(splitData);
% Assign variables
nodeNumber   = dataNumeric(:,1);
x_coordinate = dataNumeric(:,2);
y_coordinate = dataNumeric(:,3);
mach_number  = dataNumeric(:,4);
[~, sortIdx] = sort(dataNumeric(:,4));
dataSorted = dataNumeric(sortIdx, :);
nodeNumber   = dataSorted(:,1);
x_coord = dataSorted(:,2);
y_coord = dataSorted(:,3);
machS  = dataSorted(:,4);
dis = [1,2,3,4,5,6,7];
x2 = (dis - min(dis)) / (max(dis) - min(dis));
x = (local - min(local)) / (max(local) - min(local));
x3 = flip((x_coord - min(x_coord)) / (max(x_coord) - min(x_coord)));
simdatapre = [x3,machS];
% [C, new] = unique(simdatapre(:,1))
Simfinal = simdatapre;
SimAvP = reshape(Simfinal(:,1),196, []);
SimAvPy = reshape(Simfinal(:,2),196, []);
SimAv = [mean(SimAvP,1);mean(SimAvPy,1)];
%
figure
plot(x,Mach);
hold on
grid on
plot(x2,MachE);
plot(SimAv(1,:),SimAv(2,:))
title('Empirical Mach number vs normalized nozzle distance')
xlabel('normalized nozzle distance')
ylabel('Mach number')
legend('Empirical data','Analytical data','Simulation Data',Location
='best')
figure
plot(x2,tankT01);
hold on
grid on
plot(x2,PT2);
title('T_{01} and T_{02} vs normalized nozzle distance')
legend('T_{01}','T_{02}')
xlabel('normalized nozzle distance')
ylabel('stagnation temperature (K)')
```