Computational fluid dynamics analysis on the Convergent Nozzle Design

Ankit Kumar Mishra 1, Amandeep Singh 2, Vinit Goswami 3, Priyanka Sarma 4, Vishlesh Bhimrao Meshram 5, JV Muruga Lal Jeyan 6*

1, 2, 3, 4, 5 Undergraduate Student, 6*Research supervisor, 1, 2, 3, 4, 5, 6 Department of Aerospace Engineering, 1, 2, 3, 4, 5, 6 Lovely Professional University, Phagwara, India.

Abstract: Now the world is moving closer to the other planet. As we are searching for the new planets for living we had found that rockets are the only way capable to take the people from earth to the different planets, so scientist and engineers are focusing more on new rocket design which will be more efficient and more powerful in future, for example SpaceX company have rocket model SN8 which have ability to take the humans to the mars. In rockets, the nozzle used for exhaust the gases at high speed which produced by the combustion of propellants. Today we have different type of rocket nozzle like conical, bell or convergent divergent nozzles. In this paper we will discuss about the convergent and divergent nozzle design with varying inlet and throat area or throat area ratio and will see the variation in aerodynamic parameters and analyze all three designs on the ANSYS software which is computational fluid dynamic software. In rockets, convergent divergent shape is mostly used as nozzle which is also known as de-laval nozzle.

Index Terms - Pressure, Velocity, Dynamic Pressure, Mass flow rate, Drag.

I. INTRODUCTION

A nozzle is a device designed to control the direction and speed of the fluid flow. The structure of nozzle is often a pipe or tube of varying cross sectional area which is used to direct or control the flow of a fluid, speed, mass, and the pressure of the stream that emerges from them. In a nozzle, the velocity of fluid increases at the expense of its pressure energy. Nozzles are generally categorized into three types- Convergent Nozzle, Divergent Nozzle, and Convergent – Divergent Nozzle [1]. Nozzle was invented by German engineer named Ernst Korting in the year 1878 and Swedish inventor named Gustave de Laval and thus named after him. Based on shape nozzle can be classified as-

1. Bell Nozzle
2. Conical Nozzle

Convergent-Divergent Nozzle: - A convergent-divergent nozzle [13], also called De Laval nozzle, CD nozzle or con di nozzle, is a tube that is somewhat pinched in the middle so as to accelerate the flow of gas, which is passing through it. The CD nozzle consists of three major sections: -

1. The convergent section: - where the subsonic gas flow will be accelerated thus reducing the pressure and increasing its kinetic energy.
2. Throat section: - where it supports the transonic flow.
3. The divergent section: - that accelerate the supersonic flow and eventually matching the exit pressure to the outside pressure.

Figure1: Rocket Nozzle Main Section
This nozzle is designed for attaining speeds that are greater than speed of sound. The CD nozzle is commonly located at the end of the combustion chamber and it controls the gas expansion at the exhaust to efficiently convert the energy at the combustion into thrust. Its operation relies on the different properties of gases flowing at subsonic and supersonic speeds. The speed of a subsonic flow of gas will increase if the pipe carrying it narrows because the mass flow rate is constant [6]. The gas flow through a de Laval nozzle is isentropic and adiabatic. At subsonic flow the gas is compressible; a small pressure wave will propagate through it. Near the nozzle "throat", where the cross sectional area is a minimum, the gas velocity locally becomes transonic (Mach number = 1.0), a condition called choked flow. As the nozzle cross sectional area increases the gas continues to expand and the gas flow increases to supersonic velocities where a sound wave will not propagate backwards through the gas as viewed in the rest frame of the nozzle (Mach no > 1.0).

Bell Nozzle: - In a bell nozzle combustion gases flow through a throat and then the expansion away from the centerline is contained by the diverging walls of the nozzle up to the exit plane. Bell nozzle has a high angle expansion section (20 to 50 degrees) right behind the nozzle throat; this is followed by a gradual reversal of nozzle contour slope so that at the nozzle exit the divergence angle is small, usually less than a 10 degree half angle. Bells nozzles are a point design with optimum performance at one specific ambient pressure (i.e., altitude). Careful design is needed to achieve desired high altitude performance while avoiding flow separation at the walls of the nozzle near the exit when operating at low altitudes (launch), which can lead to loss of performance and possible structural failure of the nozzle due to dynamic loads. Therefore a compromise altitude must be used for the design point of a bell nozzle [4].

Conical Nozzle: - The conical nozzle was used often in early rocket applications because of its simplicity and ease of construction. The cone gets its name from the fact that the walls diverge at a constant angle. A small angle produces greater thrust, because it maximizes the axial component of exit velocity and produces a high specific impulse. The penalty, however, is a longer and heavier nozzle that is more complex to build. At the other extreme, size and weight are minimized by a large nozzle wall angle. Unfortunately, large angles reduce performance at low altitude because the high ambient pressure causes overexpansion and flow separation [1].

![Rocket Nozzle](image.png)

**Figure 2: Real View Rocket Nozzle**

### III. PROBLEM STATEMENT

The objective of this paper is to show the variation of aerodynamic parameter with varying the throat area ratio and show the variation on aerodynamic characteristic with different Mach number.

### IV. RESEARCH METHODOLOGY

In this paper the rocket nozzle designed by using Creo parametric software. The first step is to create a 2D shape and revolve with respect to the central axis. Last step, convert into 3D model for CFD testing [3]. The commonly used tools in designing to create a model in Creo 4.0 parametric are- Extrude, extrude cut, Revolve, revolve cut Sweep, Swept cut, Fillet, Chamfer, Mirror. CFD Analysis is carried out in three steps i.e.

(i) Pre-processing, geometry, – Designing, meshing, boundary conditions and numerical method.

(ii) Processing – Solving fluid flow governing equations by numerical method till the convergence is reached.

(iii) Post processing – extracting results in terms of graphs, contours which explains the physics of flow and required results. The above three steps are carried out in ANSYS using fluid fluent CFD for designing and meshing with Hybrid grid that is prismatic layer around missile design and unstructured grid. Simulations are carried out using ANSYS fluent a finite volume solver at with inlet conditions. In this analysis we use the automatic mesh generation method because of the complexity of structure [4].
V. 3-D Nozzle Design

Figure 3: Methodology

Figure 4: Meshing

Figure 5: inlet: 0.5m, throat: 0.29m

Figure 6: inlet: 0.25m, throat: 0.135m

Figure 7: inlet: 0.125m, throat: 0.0625m
### IV. INLET AND BOUNDARY CONDITION

#### Table 4.1: Inlet and boundary condition

<table>
<thead>
<tr>
<th>SL.NO</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Flow Medium</td>
<td>Air</td>
</tr>
<tr>
<td>2</td>
<td>Mach Number</td>
<td>0.3, 1.0, 1.2, 2.0</td>
</tr>
<tr>
<td>3</td>
<td>Density</td>
<td>1.225Kg/m(^3)</td>
</tr>
<tr>
<td>4</td>
<td>Turbulent Model</td>
<td>k-omega</td>
</tr>
<tr>
<td>5</td>
<td>Kinematic Viscosity</td>
<td>1.7894e-05kgs/m(^2)</td>
</tr>
<tr>
<td>6</td>
<td>Altitude condition</td>
<td>Standard sea level</td>
</tr>
</tbody>
</table>

#### VI. RESULTS AND DISCUSSION

Table 4.2: Variation of Inlet, throat area with drag

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Inlet Area (meter)</th>
<th>Throat Area (meter)</th>
<th>Nozzle Area (meter)</th>
<th>Throat area ratio</th>
<th>Mass flow rate</th>
<th>Drag</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.5</td>
<td>0.29</td>
<td>0.5</td>
<td>0.58</td>
<td>24.39</td>
<td>103.5</td>
</tr>
<tr>
<td>0.3</td>
<td>0.25</td>
<td>0.135</td>
<td>0.25</td>
<td>0.54</td>
<td>6.02</td>
<td>5.61</td>
</tr>
<tr>
<td>0.3</td>
<td>0.125</td>
<td>0.0625</td>
<td>0.125</td>
<td>0.50</td>
<td>1.47</td>
<td>0.0601</td>
</tr>
</tbody>
</table>

Table 4.3: Variation of Inlet, throat area with drag

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Inlet Area (meter)</th>
<th>Throat Area (meter)</th>
<th>Nozzle Area (meter)</th>
<th>Throat area ratio</th>
<th>Mass flow rate</th>
<th>Drag</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.5</td>
<td>0.29</td>
<td>0.5</td>
<td>0.58</td>
<td>24.39</td>
<td>103.5</td>
</tr>
<tr>
<td>0.3</td>
<td>0.25</td>
<td>0.135</td>
<td>0.25</td>
<td>0.54</td>
<td>6.02</td>
<td>5.61</td>
</tr>
<tr>
<td>0.3</td>
<td>0.125</td>
<td>0.0625</td>
<td>0.125</td>
<td>0.50</td>
<td>1.47</td>
<td>0.0601</td>
</tr>
</tbody>
</table>

Table 4.4: Variation of Inlet, throat area with drag

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Inlet Area (meter)</th>
<th>Throat Area (meter)</th>
<th>Nozzle Area (meter)</th>
<th>Throat area ratio</th>
<th>Mass flow rate</th>
<th>Drag</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.5</td>
<td>0.29</td>
<td>0.5</td>
<td>0.58</td>
<td>73.19</td>
<td>900.5</td>
</tr>
<tr>
<td>0.9</td>
<td>0.25</td>
<td>0.135</td>
<td>0.25</td>
<td>0.54</td>
<td>18.08</td>
<td>43.4</td>
</tr>
<tr>
<td>0.9</td>
<td>0.125</td>
<td>0.0625</td>
<td>0.125</td>
<td>0.50</td>
<td>4.41</td>
<td>1.01</td>
</tr>
</tbody>
</table>
PRESSURE DISTRIBUTION

a. 1st Design Result

Figure 8: Mach 0.3

Figure 9: Mach 0.6

Figure 10: Mach 0.9

b. 2nd Design Result

Figure 11: Mach 0.3

Figure 12: Mach 0.6

Figure 13: Mach 0.9
c. 3rd Design Result

Figure14: Mach 0.3  
Figure15: Mach 0.6  
Figure16: Mach 0.9  

VELOCITY DISTRIBUTION

a. 1st Design Result

Figure17: Mach 0.3  
Figure18: Mach 0.6  
Figure19: Mach 0.9
b. 2nd Design Result

Figure 20: Mach 0.3

Figure 21: Mach 0.6

Figure 22: Mach 0.9


c. 3rd Design Result

Figure 23: Mach 0.3

Figure 24: Mach 0.6

Figure 25: Mach 0.9
a. 1st Design Result

Figure 26: Mach 0.3

Figure 27: Mach 0.6

Figure 28: Mach 0.3

a. 2nd Design Result

Figure 29: Mach 0.3

Figure 30: Mach 0.6

Figure 31: Mach 0.9
b. 3rd Design Result

![Figure 32: Mach 0.3](image1)

![Figure 33: Mach 0.6](image2)

![Figure 34: Mach 0.9](image3)

VI.) CONCLUSION
From results, we conclude that with decreasing the throat area ratio mass flow rate decrease but velocity at throat increase and we analyses the all three design and we see that with decrease in throat area ratio, effect of pressure area increase, dynamic pressure increase and we are easily approaching the supersonic speed. But for heavy rocket we want more mass flow rate because by the thrust equation we see that thrust is depend on the mass flow rate so our Last conclusion of this paper is that, first rocket nozzle design is good for heavy weight rocket’s because in first design the mass flow rate is greater than our two design as compare.

VII.) REFERENCES

