3-D printing and computational fluid analysis of single expansion ramp nozzle

S Benjamin Franklin¹, R Arul² and T K Sakthivel³

¹Assistant Professor (Sl.Gr), Department of Mechanical Engineering, Sri Ramakrishna Institute of Technology, Coimbatore, India.
²,³Assistant Professor, Department of Mechanical Engineering, Sri Ramakrishna Institute of Technology, Coimbatore, India.

Email: benjaminfranklin.me@srit.org

Abstract. To develop a scram jet nozzle by altering the existing supersonic nozzle design. A SCRAM jet nozzle alias, Supersonic combustion RAM nozzle is typically a derived structure of supersonic nozzle, in which the exit of the nozzle is truncated for minimizing the losses. The speed at the inlet should be greater than five Mach (>5), to give the full characteristics of a SCRAM jet. That means the exit will be greater than five Mach (>5), it attains a hypersonic condition. The SCRAM jet is not used in any of the voyages because it is said to less efficient than other nozzles. This SCRAM jet is a developing stage conceptual design in the flight era. The future will rely on these technologies. Our modification in this technology is to attain a hypersonic condition our modified supersonic nozzle. To attain this there are certain controlling parameters to be considered before designing a nozzle, they are thrust required, lift, drag, friction on sidewalls. These are minimized by swirling the air in the divergent section of the nozzle. The swirling action is attained by having internal helical grooves on the divergent section of the nozzle.

1. Introduction

Supersonic Combustion Ramp Nozzle is typically an advanced type nozzle used for Scram jets or foe supersonic flights. The SERN is convergent at the inlet and divergent at the exit. The exit of the nozzle is truncated to an angle so that it is more of Ramp than the crawl. Sutton investigated to find an optimal shape of a three-dimensional supersonic nozzle for a rectangular ramjet combustion chamber with rounded edges by taking account the skin friction effects. Here the nozzle inlet conditions are mean values of the combustion chamber exit conditions with the assumption of a constant specific heat ratio[1].

One of the challenges in the development of a turbine-based combined cycle (TBCC) propulsion systems is the transition of the propulsion system from one engine to the other. This will require a large ground-based test facility that can accommodate a dual propulsion flow path vehicle [2]. The experimental and computational results of the single expansion ramp nozzles to identify the various thrust performance concerning the nozzle pressure ratio conditions the internal performance of single expansion ramp nozzles with various combinations of the geometric parameters[3]. By considering the nozzle pressure ratio conditions concerning the different cases the internal and static performance
he finds out the thrust variations occurred in the nozzles [4]. Advances in CFD for turbulent separated flows and transonic potential flows. The development of code is given that solves the complete potential equation in an exceedingly finite volume, cell-centred formulation. The distinctive feature concerning this code is that solutions are earned on unstructured grids. The experimental results on shock structure, flow separation, and mixing enhancement of ambient dry air in rectangular over-expanded supersonic nozzles. This experiment it is observed that the asymmetric lambda foot is formed when the nozzle pressure ratio is further increased. The unsteady phenomena in supersonic nozzle flow separation. This work considers the instability of the jet plume from an over-expanded, shock containing convergent-divergent nozzle and attempts to correlate this instability to internal shock-induced separation phenomena [5].

The coinciding wall pressure measurements in an exceedingly blast wave / turbulent physical phenomenon interaction work consist of coinciding speed fields and unsteady wall pressure measurements. These results provide access for the primary time to the spatial-time correlation between wall pressure and speed in an exceedingly blast wave turbulent physical phenomenon interaction [6].

**Figure 1. Ideal nozzle**

The aesthetics of the nozzle is designed based on the unofficial dimensions of the above ideal nozzle dimensions. But the real challenge is designing the nozzle based on the flow parameters. Usually, nozzles are designed based on the shape. We create a nozzle based on the requirements of the Flight, the methods are by designing with the required speed of the flight. By designing the nozzle with the required thrust. Figure 1 shows the ideal nozzle. We choose the first method for designing the nozzle. By, knowing the essential parameters such as:

- Maximum Altitude of the flight
- Mach number at inlet and exit
- Exit pressure
- Specific heat capacity of the fuel.

Table 1 shows the values of the assumed parameters.
Table 1. Assumed parameters

| Altitude (Metres) | Mach number at inlet | Mach number at exit | Exit atmospheric pressure (Bar) | Specific heat capacity (Cp) k J/kg | Specific heat capacity (Cv) k J/kg | Gas constant (R) |
|------------------|----------------------|---------------------|---------------------------------|-----------------------------------|-----------------------------------|-----------------|
| 25000            | 3.5                  | 12                  | 0.0245                          | 10.183                            | 14.381                            | 4.124           |

Assume the material of the nozzle is graphite rich steel. By assuming the above parameters, the design of the nozzle is calculated. Before calculating, the main parameter of the scramjet rocket nozzle is its dimensions. The exhaust gas coming out from the cryogenic engine is fed inside the rocket nozzle. So, When the cryogenic fuel is the exit from the engine, the gas used to get expanded from its highly compresses state. Thus, the velocity increases, when the velocity of the air gets increased the air entering the nozzle will decrease because there occurs the concept of choking, due to the choking condition, the pressure increases in the throat section of the nozzle and thereby creating shock waves. This shockwave produced will ablate the pressurized gas to make the gas to expand. The expansion of gas is caused due to the shock waves produced. More the expansion of the gas more will be the thrust produced. So, the inlet area, exit area and the throat area are defined. After that, the length of the nozzle is also defined. By using the above formulas, the dimensions are defined by considering the flow fully as isentropic. The Ratio of inlet and exit of the nozzle is obtained by the formula:

\[
\frac{A_x}{A_y} = \frac{M_y}{M_x} \sqrt{\left[\frac{1+\frac{k-1}{2}}{1+\frac{k-1}{2}}\right]M_x^2/(k-1)/(k+1)}
\]

(1)

From the area ratio that is obtained from the above formula, it’s time for the pressure, temperature and velocity at inlet and exit to be found out by using the equations stated below. For finding the ratio of temperature at inlet/exit to the temperature at stagnation, Equation 2 is taken:

\[
\frac{T}{T_o} = \frac{1}{1+\left(\frac{k-1}{2}\right)M_x^2}
\]

(2)

For finding the pressure at inlet/exit to the stagnation, Equation 3 is taken:

\[
\frac{P}{P_o} = \left(\frac{T}{T_o}\right)^{\gamma-1}
\]

(3)

For finding the Mach number at choking:

\[
M^* = \frac{C}{C^*} = \sqrt{\left[\left(\frac{y+1}{2}\right)M_x^2/(1 + \left(\frac{y-1}{2}\right)M_x^2)\right]}
\]

(4)

To find an area at inlet/exit and area at stagnation:

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

(5)

The formula for finding Mass flow rate:
By defining the parameters by using the above equations, the results are obtained they are.

**Table 2. Defining parameters**

| Parameter               | Value            |
|-------------------------|------------------|
| Temperature at inlet    | 275 K            |
| Temperature at outlet   | 4200 K           |
| Mass flow rate          | 2.1891 kg/sec    |
| Velocity at exit        | 12456.66 m/s     |
| Pressure at exit        | 0.006 bar        |
| Pressure at inlet       | 3.5 bar          |
| Radius at inlet         | 25.1175 mm       |
| Radius at exit          | 63.6125 mm       |

Definition of the area of the nozzle is an essential parameter and it is one of the important parameters used to drop the pressure at the exit of the nozzle. Table 2 shows the defining parameters. The designed nozzle will be perfect enough to convert to a SERN. The designed nozzle is displayed in Figure 2.

**Figure 2. Single expansion ramp nozzle**

The development of the SERN is a tedious process and it does not have a standard process for designing it. It has some unofficial process for designing it. The only design process remaining is the truncation of the nozzle. The truncation of the nozzle process does not have a standard process for it. The best truncation is selected by taking various truncation ratios and the CFD is done for obtaining its results and the best truncation is taken as the best design.

For various truncation lengths, the efficiency of the nozzle is taken. And the best result is obtained by truncating exit at the ratio of 1/0.354 (length of the nozzle to the length of the nozzle from exit).

2. **Swirl flow in the nozzle**

The ultimate aim is to swirl the air in the exit of the nozzle. This swirling action is to create a drastic and extreme pressure drop at the exit of the nozzle. The swirl flow is used to mix the Air-fuel mixture
sufficiently. The effect would be very good resulting in high expansion of air in the nozzle divergence section.

3. **Novel design in the nozzle**

The innovation will be the next generation rocket nozzle that produces an extreme pressure drop at the exit of the nozzle. To create a swirling action of the nozzle a small change is done in the nozzle’s inner wall design. Spiral helical grooves are given on the inner walls of the nozzle. The Dimension of the groove is displayed below in Figure 3. The inner grooves will swirl the air which is passing through it. The grooves are given in such a way that the edges of the nozzle will not have any sharp edges which result in a higher level of entropy.

From the above diagram, it is said that there are no sharp grooves on the nozzle. And the semicircle given is of 3.5mm diameter and the fillet is given is of 1mm.

![Figure 3 Inner walls of the nozzle](image)

The texture of the inner walls of the nozzle is shown below which clearly shows that there are no sharp grooves.

4. **The flow of exhaust plume**

At high velocity, the Mach angle should be greater than 150 to get a swirling action in the nozzle. This is that the half-angle of the nozzle is 150 so the plume must have an angle of 150 and above to swirl through the nozzle. Various Mach Cone at various pressure drops and altitude are displayed in the below in Fig 8. Various plumes at various levels Mach angle can be obtained from the formula:

\[
\sin (\alpha) = \frac{1}{M} \quad (7)
\]

Where \( \alpha \) is the Mach angle and \( “M” \) is the Mach number.

**Table 3. Mach angle at inlet and exit**

| Mach number | Mach angle (In Degree) |
|-------------|-----------------------|
| 3.5         | 16.38040              |
| 12          | 21.190908             |

5. **Computational Fluid Dynamics**

ANSYS Fluid flow (Fluent) is selected because the nozzle is kept stationary in the rocket and the fluid in the nozzle is going to be dynamic inside the nozzle.
6. Meshing
Mesh generation is that the apply of generating a plane figure or solid mesh that approximates a geometrical domain. The term "grid generation" is often used interchangeably. Typical uses area unit for rendering to a monitor or for physical simulation like finite part analysis or process fluid dynamics. The Mesh view is displayed below in Figure 4.

![Meshed view](image)

**Figure 4.** Meshed view

7. Solution and results
The results are obtained and the graphs can be plotted for the results and the obtained results are shown below in Figure 5 and Figure 6.

![Temperature distribution in the nozzle](image)

**Figure 5.** Temperature distribution in the nozzle

The solution is done using Iterative method. The iterative method is used for extracting the finest results in the nozzle.

![Temperature distribution in the inner walls of the nozzle](image)

**Figure 6.** Temperature distribution in the inner walls of the nozzle

8. 3D printing
The method of 3D-printing used for printing the nozzle is Selective Laser Sintering. Selective laser sintering (SLS) is a technique of additive manufacturing (AM) which uses a laser as the power source for sintering powdered material (typically nylon/polyamide), automatically focusing the laser at space points identified by a 3D model, binding the material together to create a solid structure. An additive layer technique, SLS requires the use of a high-power laser (e.g. a carbon dioxide laser) to fuse small particles of plastic, steel, ceramic or glass powders into a mass that has a desired 3D form.
The laser selectively fuses powdered material by scanning cross-sections formed on the surface of a powder bed by a 3-D digital description of the component. The powder bed is lowered by one-layer thickness after each cross-section is inspected, a new layer of material is added on top, and the process is repeated until the portion is complete.

The results of the above-designed nozzle are designed and the results are taken by manual calculation and from the Computational fluid dynamics done in the ANSYS and the results are obtained and it is displayed below.

![Figure 7. Extreme pressure drop](image)

The plume is diffracted to the ramp of the nozzle and the results are displayed in Figure 7, Figure 8 and Figure 9.

![Figure 8. Diffraction of the plumes at the ramp of the nozzle](image)

![Figure 9. Thermal distribution in the nozzle](image)

9. Conclusion
Single expansion ramp nozzle is a developing nozzle in the jet industry. Single expansion ramp nozzle is developed only in the Boeing. Single expansion ramp nozzle is developed only in the supersonic nozzle. The results of the nozzle are designed and the analytical solutions are verified and validated the Computational Fluid Dynamics Analysis.

Acknowledgement
Sincere thanks to Sri Ramakrishna Institute of Technology, Coimbatore, Tamil Nadu, India.
References

[1] Sutton G P, Stepped Nozzle, U.S. Patent 5,779,151, 1998; M. Ferlin, Assessment and benchmarking of extendible nozzle systems in liquid propulsion. *AIAA Paper 4163* 2012

[2] Hoffman J D 1987 Design of compressed truncated perfect nozzles. *J. Propul. Power* 3 150–156

[3] Rao G V R 1961 Recent developments in rocket nozzle configurations, *ARS Journal* 31 1488–1494

[4] Rao G V R 1958 Exhaust nozzle contour for optimum thrust jet propulsion, *ARS Journal* 28 377–382

[5] Yu K K, Xu J L, Zhang X F and Liu S 2004 Inverse design of shock wave distortion for a demonstrators- overview, status and future plans *Acta Astronaut.* 55

[6] Quan Z B, Xu J L, and Jian-wei J W M, *Design of nonlinearly compressed SERN profile*, College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, China.