Design and Optimization of Aerospike nozzle using CFD

Naveen Kumar K¹, Gopalsamy M², Daniel Antony², Krishnaraj R³ and Chaparala B V Viswanadh³

¹ Assistant Professor, Department of Aeronautical Engineering, Kumaraguru college of technology, Coimbatore, India.
² Assistant Professor, Department of Aerospace Engineering, Karunya University, Coimbatore, India.
³ UG Scholar, Department of Aerospace Engineering, Karunya University, Coimbatore, India.

E-mail: kraj220@karunya.edu

Abstract. New rocket designs are being adopted to increase the performance of the current satellite launch vehicles (SLVs). But, the aerospike (or plug) nozzle concept that has been under development since the 1950s is yet to be utilized on a launch platform. Due to its ability to adjust the environment by altering the outer jet boundary, the aerospike nozzle delivers better performance compared to present day bell nozzle. An aerospike nozzle is designed for 20 bar pressure ratio. In order to improve the performance of the aerospike nozzle for various conditions, optimization of the nozzle was carried out for some important design parameters and their performances were studied for cold flow conditions. Initially a model of an aerospike nozzle is created for certain parameters, then the optimization process is carried out for the nozzle (Truncated model & Base bleed model). Optimized model is designed by the software GAMBIT and the flow behaviour is analysed by the Computational Fluid Dynamics (CFD) software called FLUENT. Comparison also takes place between the full length and the optimized models.

Keywords- Aerospike nozzle, Contour, Base bleed, altitude compensation

1. INTRODUCTION
The most popular altitude-compensating rocket nozzle to date is the aerospike nozzle, the origin of which dates back to Rocket dyne in the 1950s. This type of nozzle was designed to allow for better overall performance than conventional nozzle designs.

1.1. Nozzle
A nozzle is a mechanical device of varying cross section which controls the direction and characteristics of the fluid (Air or Water) flowing through it. They are used in rocket engines to expand and accelerate the combustion gases, from burning propellants, so that the exhaust gases exit the nozzle at supersonic or hypersonic velocities.
When the fluid flows through the nozzle it exits at a higher velocity than its inlet velocity. This phenomenon occurs due to conservation of mass which states that the rate of change of mass equals to the product of density, area and velocity. 

\[ \frac{dA}{A} = \frac{dV}{V} (1 - N^2) \]

**1.2. NEED FOR NEW DESIGN**

The revolution in aerospace propulsion was increased greatly during World War 2. Faster, bigger and more efficient aerospace vehicles were required which led to the birth of Space research organizations like NASA. Speaking about the future, advanced rocket propulsion systems will require exhaust nozzles that perform efficiently over a wide range of ambient operating conditions. Most nozzles either lack this altitude compensating effect or they are extremely difficult to manufacture. In present day, only bell nozzles are used for launch activities. But, these bell nozzles have a major drawback of decreasing in efficiency as altitude increases. This is due to a phenomenon causing loss of thrust in the nozzle at higher altitudes called as separation of the combustion gases.

For conventional bell nozzles, loss mechanisms fall into three categories:

- geometric or divergence loss,
- Viscous drag loss,
- Chemical kinetics loss.

Geometric loss results when a portion of the nozzle exit flow is directed away from the nozzle axis, resulting in a radial component of momentum. In an ideal nozzle, the exit flow is completely parallel to the nozzle axis and possesses uniform pressure and Mach number. By calculating the momentum of the actual nozzle exit flow and comparing it to the parallel, uniform flow condition, the geometric efficiency is determined. By careful shaping of the nozzle wall, relatively high geometric efficiency can be realized. A drag force, produced at the nozzle wall by the effects of a viscous high-speed flow, acts opposite to the direction of thrust, and therefore results in a decrease in nozzle efficiency. The drag force is obtained by calculation of the momentum deficit in the wall boundary layer. This viscous drag efficiency is defined as:

\[ \eta_{drag} = 1 - \frac{\Delta C_{T}(\phi)}{C_{T}(ideal)} \]

The third nozzle loss mechanism is due to finite-rate chemical kinetics. Ideally, the engine exhaust gas reaches chemical equilibrium at any point in the nozzle flow field, instantaneously adjusting to each new temperature and pressure condition. In real terms, however, the rapidly accelerating nozzle flow does not permit time for the gas to reach full chemical equilibrium.

\[ \eta_{kin} = 1 - \frac{\Delta C_{T}(OD\xi)}{\Delta C_{T}(OD\xi)} \]
The overall nozzle efficiency is then given by the combined effects of geometric loss, viscous drag and chemical kinetics: \( \eta_{\text{overall}} = \eta_{\text{geo}} \times \eta_{\text{kin}} \times (1 - \eta_{\text{drag}}) \).

A long nozzle is needed to maximize the geometric efficiency; but simultaneously, nozzle drag is reduced if the nozzle is shortened. If chemical kinetics is an issue, then the acceleration of exhaust gases at the nozzle throat should be slowed by increasing the radius of curvature applied to the design of the throat region. The optimum nozzle contour is a design compromise that results in maximum overall nozzle efficiency. Nozzle contours can also be designed for reasons other than for maximum thrust. Contours can be tailored to yield certain desired pressures or pressure gradients to minimize flow separation at sea level. A nozzle contour designed to produce parallel, uniform exit flow, thereby yielding 100% geometric nozzle efficiency, is called an ideal nozzle.

This ideal nozzle is extremely long and the high viscous drag and nozzle weight that results are unacceptable. Some design approaches consider truncating ideal nozzles keeping in mind the weight considerations. Most companies have a parabolic curve-fit program, generally used to approximate Rao optimum contours, which can also be used to generate desired nozzle wall pressures. For nozzles at higher altitudes, vacuum performance is the overriding factor relating to mission performance and high nozzle area ratio is therefore desirable. However, nozzle over-expansion at sea level does result in a thrust loss because the wall pressure near the nozzle exit is below ambient pressure. If the nozzles exit area could be reduced for launch and then gradually increased during ascent, overall mission performance would be improved. The ideal rocket engine would make use of a variable-geometry nozzle that adjusted contour, area ratio and length to match the varying altitude conditions encountered during ascent. This feature is referred to as Altitude Compensation.

### 1.3. Aerospike Nozzle

The aerospike nozzle is a bell nozzle with its nozzle profile turned inside out. Flow of combustion gases is directed radially inward towards the nozzle axis. In the annular aerospike nozzle, flow issues from an annulus at a diameter located some radial distance from the nozzle axis. Flow is directed radially inward toward the nozzle axis. This concept is the opposite of a bell nozzle which expands the flow away from the axis along diverging nozzle walls. In an aerospike, the nozzle expansion process originates at a point on the outer edge of the annulus which is referred to as the "cowl-lip." In a standard bell nozzle, flow expansion continues regardless of what the ambient pressure is, and the flow can continue to over-expand until it separates from the nozzle walls. The linear aerospike, spike consists of a tapered wedge-shaped plate, with exhaust exiting on either side at the "thick" end.

#### 1.3.1. Working Principle of Aerospike nozzle:

The function of a rocket nozzle is to direct all gases, generated in the combustion chamber of the engine and accelerated by the throat, out of the nozzle. The key feature of the aerospike engine is that, as the launch vehicle ascends during its trajectory, the decreasing ambient pressure allows the effective nozzle area ratio of the engine to increase. An aerospike nozzle is often referred to as an altitude-compensating nozzle, because of its specific design capability of maintaining aerodynamic efficiency as altitude increases and thus throughout the entire trajectory. At the outer cowl lip, the gas expands to the atmospheric pressure immediately, and then causes serious expansion waves propagating inward at an angle through the gas stream. At the location where the last expansion wave intercepts the spike, the gas pressure is equal to the atmospheric pressure. For the over expanded case, the spike changes the gas to be directed outward, and thus compression waves form and propagate outward at an angle and reflect off the jet boundary as expansion waves. This process then begins again. The aerospike features a series of small combustion chambers along the ramp that shoot hot gases along the ramp's outside surface to produce thrust in a spike-shaped plume, hence the name "aerospike."
The ramp serves as the inner wall of the bell nozzle, while atmospheric pressure serves as the "invisible" outer wall. The combustion gases race along the inner wall (the ramp) and the outer wall (atmospheric pressure) to produce the thrust force.

1.3.2. Flow around Aerospike Nozzle
The main advantage to the annular aerospike nozzle design (both full length and truncated spike) is its altitude compensation ability below or at its design altitude. More specifically, the aerospike will not suffer from the same overexpansion losses a bell nozzle suffers and can operate near optimally, giving the highest possible performance at every altitude up to its design altitude. Above the design altitude, the aerospike behaves much like a conventional bell nozzle. Figure below shows the exhaust flow along an aerospike at low altitudes, design altitude, and high altitudes for a full spike and a truncated spike. Multiple expansion and compression, or shock, waves are evident in the flow in Figure these waves lead to losses in thrust. The outer flow boundary of the aerospike is the atmosphere itself. Unique to aerospike engines operating at their design altitude, engine geometry at the throat determines the expansion ratio of the aerospike nozzle and thus the corresponding engine performance.

![Figure 2. Model of aerospike with flow field](image-url)
At the design altitude of the nozzle, the exhaust flow at the chamber exit lip will follow a parallel path to the centreline to the exit plane. Therefore, the expansion ratio for a full-length spike at design altitude is equivalent to the chamber exit lip area divided by the throat area. As the ambient pressure decreases, the hot gas/ambient air boundary expands outward changing the pressure distribution along the spike; as a result, the expansion ratio increases. As the ambient pressure increases (low altitudes), the higher ambient pressure compresses the hot gas/ambient air boundary closer to the spike resulting in an expansion ratio decrease. The pressure distribution change along the spike and the location of the hot gas/ambient air boundary is automatic thus permitting altitude compensation up to the design altitude of the nozzle. Above the design altitude of the nozzle, the pressure distribution along the nozzle wall is constant. The expansion of the flow exiting the combustion chamber is governed by the Prandtl-Meyer turning angle at the throat.

**Figure 3.** Exhaust Flow from a Full and Truncated Spike

**Figure 4.** Exhaust Flow along a Truncated Aerospike Nozzle

According to the aerospike nozzle numerical analyses the results of the altitude compensation capabilities of an aerospike up to the design altitude are undeniable. Furthermore, the aerospike
performs worse at high altitudes compared to bell nozzles with equal expansion ratios (exit area divided by throat area); therefore, to get the benefit of the aerospike, the design pressure ratio and the expansion ratio should be chosen as high as possible. The design pressure ratio is the ratio of the chamber pressure to the ambient pressure; ambient pressure is based on the chosen design altitude. If the spike is truncated, the aerospike advantage at higher altitudes (orbit transfer missions) includes shortened nozzle length and lower mass as compared to an equivalent performance bell nozzle design for orbit transfer missions.

1.3.3. Advantages and Disadvantages of Using an Aerospike Nozzle:
The aerospike nozzle has 90% overall better performance than the conventional bell shaped nozzle. The efficiency at low altitudes is much higher because the atmospheric pressure restricts the expansion of the exhaust gas. A vehicle using an aerospike nozzle also saves 25-30% more fuel at low altitudes.

At high altitudes, the aerospike nozzle is able to expand the engine exhaust to a larger effective nozzle area ratio. An aerospike nozzle with an expansion ratio of 200:1 to 300:1 can increase the thrust and specific impulse by five to six percent. Specific impulse is the total impulse per unit weight of propellant. As of now, the most widely used nozzle type is the bell-shaped nozzle. It has a high-angled expansion section, usually 20-50°, right behind the nozzle throat, which is then followed by a gradual reversal of nozzle contour slope so that the nozzle exit divergence angle is small, usually less than a ten-degree half angle. The drawback to using this type of nozzle is that its optimum design is for a specific altitude, and thus is better suited for multi-stage to orbit. The aerospike design is suitable for Single Stage to Orbit (SSTO) flight. There are rarely some disadvantages on using an aerospike nozzle.

The after body induces heat, and to cool means that the performance reduces along with the pressure against the nozzle. Another issue is weight, which as previously stated can be resolved through truncation. During flight a transonic and supersonic regime, generally between Mach 1 and 3, the slipstream effect reduces the aerospike nozzle’s performance due to the external flow over the vehicle because the plume tends to draw in the air flow and thus alters the aerodynamics at the aft end of the vehicle. Finally, the performance is more difficult to evaluate because of the complex flow field and the turbulence involved.
1.4. Objective

This thesis designs, analyses and optimizes the aerospike nozzle contours using CFD tool (FLUENT) and the prototype is fabricated from the design and tested in the ISRO (Mahendragiri) lab. Considering that the rockets are designed for high-altitude applications (since most of its flight time is at high altitude) and the effects of density changes in the air will be negligible, the annular nozzle will probably be more appropriate for this application since the automatic altitude adjustment characteristics of the aerospike nozzle will not be necessary. However, in the interest of knowledge and the application of the data to future proposals, the aerospike design will be valuable and advantageous.

2. DESIGN METHODOLOGY

2.1. From Requirements to Preliminary Design:

Since the cold flow is considered, the values of Specific Heat Ratio ($\gamma = 1.4$), gas constant of exhaust ($R = 287$) and chamber temperature ($T_i = 300$K) are taken of the values of ideal gas. For every design there has to be minimum one parameter will be considered constant to determine the other design parameter values. Here, for the aerospike nozzle design, Mass Flow Rate ($2$Kg/S) and Nozzle Pressure Ratio ($NPR = 20$) were considered as constant. Based on the value of Mass flow rate and ideal gas constant values, other basic design parameters were determined by 2D Isentropic Flow Equations.

2.1.1. Plug Nozzle Contour Design:

Basic Design Methods:

The design of spike contour of the aerospike nozzle is the most important step in the overall design of the nozzle, which varies according to the operating conditions and application. However, the design procedure, including the basic physics behind the spike design remains the same. The design of spike can be done using the following methods

i. Rao’s method based on calculus of variation

ii. Simple approximate method

iii. B-Spline Method

Simple Approximation Method:

Here, Simple approximation method is chosen to design spike contour. The simple approximate method assumes a series of centred isentropic expansion waves occurring at cowl lip of the spike nozzle. Using this method, the annular spike contour for a given pressure ratio, area at throat, and ratio of specific heats can be determined.

A brief description of this is given below:

The expansion ratio is determined for the corresponding pressure ratio from the relation which specifies the variation between the two. Since the flow is assuming to be parallel to the nozzle exit the thruster angle is given by,

$$\theta_t = \theta(M_e)$$

Where $M_e$ is Mach number at exit and the Prandtl-Meyer function is given by

$$\theta = \frac{\gamma + 1}{\gamma - 1} \tan^{-1} \frac{\gamma - 1}{\gamma + 1} (M^2 - 1) - \tan^{-1} \sqrt{M^2 - 1}$$

The Throat area is,

$$A_t = \frac{\pi (r_2^2 - r_1^2)}{\cos \theta}$$
And exit area of the spike is,
\[ A_e = \pi (r^2 - r_i^2) \]

Where \( r_e \) denotes radius of cowl lip and \( r_b \) denotes radius of base pressure, the chamber pressure \( F \) thrust and \( C_f \) the thrust coefficient. Once the expansion ratio is calculated \( r_e \) and \( r_i \) can be determined for a fixed \( A_e \).

The radial co-ordinate of spike nozzle is given by,
\[ r^2 = r_e^2 - \frac{(r_e^2 - r_i^2) A \sin(\mu + \theta)}{A_e \sin(\mu) \cos(\theta_e)} \]

The axial co-ordinate is given by,
\[ x = \frac{r_t - r_i}{\tan(\mu + \phi)} \]

Where \( \theta = \theta_e - \phi \) and Mach angle \( \mu = \sin^{-1}(\frac{1}{M_e}) \).

The relationship for Mach number and area ratio is that given for isentropic expansion of a one-dimensional flow through varying area conduit. With the help of a FORTRAN program, the radial and the axial co-ordinates of the spike contour were found, thus obtaining the perfect shape of the spike satisfying the given problem statement.

2.2. The FORTRAN Program

Outline of External Expansion Plug Nozzle Design

INPUT:
- Estimate exit Mach number (which can be obtained from isentropic flow tables based on the expansion ratio of specific heats).
- Expansion ratio.
- Number of contour points
- Gas constant
- Exit temperature
- Atmosphere pressure ratio
- Constant of proportionality in Newton’s second law
- Ratio of specific heats (constant or variables)

OUTPUT:
- Angle between plug axis and sonic line
- Width of throat gap
- Optimum thrust coefficient
• Mach number distribution
• Co-ordinates of plug contour
• Pressure ratio at each point
• Cumulative vacuum thrust coefficient
• Cumulative specific impulse
• Cumulative vacuum specific impulse

2.2.1. Flow chart of External Expansion Plug Nozzle Design:

C.C. Lee (1963) created and verified the FORTRAN program for external plug nozzle based on the simple approximation method which is used here.
2.3. Governing equations

To simulate gas flow in the primary nozzle and on the outside of the plug, the compressible Reynolds-averaged Navier-Stokes (RANS) equations are employed. The following assumptions are made in the derivation on the appropriate form of the equations: compressible flow, Newtonian fluid, negligible gravity effects, no slip condition, 2-dimensionality, no chemical reactions, and thermal equilibrium. The equations are solved using an implicit, coupled, three-dimensional finite volume method. A second-order accurate, upwind discretization scheme is used for the convective term and multigrid scheme is used to accelerate the rate of convergence. Fluent 6.3 is used to perform these calculations. Considering that the velocity of the flow is in supersonic, inviscid turbulence models are employed.
The ambient pressure does not affect the flow in the primary nozzle. Therefore, to simplify the computation, in this study the flow fields over the plug wall are considered as the environment and the atmospheric boundary condition is given to it throughout the calculation.

3. CFD SETUP AND SOLUTION
The designed aerospike nozzle was analyzed using commercially available CFD software Fluent 6.3. The modeling of the domain was made using Gambit 2.3 software. It uses Finite volume technique to solve the Navier Stokes equation. Proper meshing was done to ensure the accuracy of the solution.

3.1. Design Parameters:
The input parameters for the design of the annular aerospike nozzle are,
- Exit Area ratio \( \frac{A_e}{A_t} \) = 2.8998
- Exit diameter \( \text{Re} \) = 18mm
- Chamber Pressure = 20 bar
- Exit Pressure = 1 bar
- Mass Flow Rate = 1.988Kg/Sec

From the given interval of design parameters, some of the values that are used for the design are,
- Length of the nozzle = 60mm
- Thrust = \( 1.268 \times 10^3 \) N
- Coefficient of thrust \( C_T \) = 1.446
- Mach No at the end of the ramp = 2.601

3.2. COMPUTATIONAL DOMAIN CREATION (GAMBIT):
The computational domain of the annular Aerospike nozzle modelled in Gambit. Boundary conditions are imposed in this Computational domain. The distance from the tip of the plug to the end of the imaginary boundary is 10 and 5 times of the nozzle outer diameter with respect to X and Y axis. The nozzle geometry and flow is axisymmetric, so upper half of the nozzle only designed for flow analysis and thus reduce the computational time. The modelled nozzle and the nozzle with the computational domain is shown below,

![Figure 7. Designed aerospike nozzle in GAMBIT.](image-url)
3.2.1. *Grid Generation:*

Quad-map meshing element is used to mesh the entire domain. The intensity of the grid is more near the wall and the regions where the gradients are high. The grid independency study was carried out and it was found that a minimum of 27000 nodes were needed. The total numbers of nodes on the domain are 27998 and cells are 27396.

![Figure 8. Aerospike nozzle design with far field boundary](image)

![Figure 9. Aerospike nozzle design after mesh](image)
3.3. **FLOW PARAMETERS AND TURBULANCE MODEL (FLUENT):**

After modelling and meshing the nozzle in GAMBIT 2.3, CFD analysis made by using FLUENT 6.3 to solve the Navier-Stokes Equations. Air is used as the fluid medium. The material property of the gas is ideal gas. Since, the Mach No > 0.3, The ambient conditions are supposed at the far field Boundary Conditions.

FLUENT Boundary Conditions used for simulations are given below,

i. **Solver** = Density based
ii. **Space** = Axisymmetric
iii. **Viscous model** = Spalart – Allmaras, RNG k-ε
iv. **Operating Pressure** = 0 Pa

*Boundary conditions*

**Pressure Inlet**

i. **Gauge total pressure** = 8 bar
ii. **Total temperature** = 300 K
iii. **Direction Specification** = Normal to boundary

**Pressure Outlet**

i. **Gauge total pressure** = 1 bar
ii. **Backflow total temperature** = 300 K

**Far Field Conditions**

i. **Gauge pressure** = 1 bar
ii. **Mach Number** = 0.4
iii. **Temperature** = 300 K

*Figure 10.* High density grids on the areas where high flow gradient occurs.
Solution Control

i. Discretization flow = Second order upwind

ii. Solver parameters = Courant No (3)

3.3.1. Contours of Full Length Nozzle:

Mach Contour

Figure 11. Mach Contour of the Full Length Aerospike Nozzle

Figure 12. Streamlined Mach Contour of the Full Length Aerospike Nozzle
3.4. Optimization studies

Need of Optimization for Aerospike Nozzle:

The wake formation at the end of the plug is studied. The formation of wake at the surface of the nozzle may affect the thrust production of the nozzle. To overcome this problem, the various optimized models of the same aerospike nozzle is designed. Various models were considered & compared. The Performance of Full length & optimized models were compared.

Nozzle design parameters considered for optimization:

i) Effect of truncation:
10%, 20%, 40%, 60% & the point on the axial length where the Mach No starts to decrease considered for truncation.

ii) Effect of base bleed:
Base bleed systems combined with the above mentioned truncated models (20%, 40%, 60% &the point where the Mach No maximum) are considered for optimization.

iii) Effect of thruster Area Ratio:
Current nozzle design has the thruster area ratio (TAR) of 1.2. In order to obtain better performance, designs having Thruster area ratio of 1.3, 1.4 and 1.5 are designed and their analyzed results were taken for comparison.

3.4.1. Designs of Optimized models:

1. Truncated model:
In the effect of truncation criteria, the truncated models of 10%, 20%, 40%, 60% & the point on the axial length were the Mach Number starts to decrease are designed and analysed for the same flow conditions specified above.
The table below shows the highest Mach number attained on truncated models:

| TRUNCATED MODEL | MACH NO ATTAINED IN S-A TURBULANCE MODEL | MACH NO ATTAINED IN K-Ε TURBULANCE MODEL |
|-----------------|------------------------------------------|------------------------------------------|
| Full Spike      | 3.82                                     | -----                                    |
| 10%             | 3.82                                     | -----                                    |
| 20%             | 3.56                                     | 3.57                                     |
| 40%             | **3.54**                                 | **4.47**                                 |
2. Base bleed model:
In the truncated model, the formation of wake on the base of the nozzle creates pressure drops which cause reduction in its thrust producing capability. So, by providing additional flow through the nozzle to its base will reduce the wake formation. For the additional flow a hole has created in the nozzle’s solid part from the pressure inlet portion to the base of the nozzle.

The only drawback of this model is it requires excess amount of mass flow rate for base bleed operation.

The table below shows the highest Mach no attained on truncated models:

| TRUNCATED MODEL | BASE DIA (mm) | BLEED HOLE DIA (mm) | MASS FLOW RATE (Kg/S) | EXCESS % OF MASS FLOW RATE FOR BASE BLEED |
|-----------------|--------------|---------------------|-----------------------|-----------------------------------------|
| 20%             | 3.918        | 2                   | 1.2699E-3             | .6356                                   |
| 40%             | 9.176        | 4.588               | 6.6833E-3             | 3.345                                   |
| 60%             | 17.026       | 8.512               | 23.004E-3             | 11.5135                                 |
| @ Max Mach no attained pt. | 23.462 | 8.512 | 23.004E-3 | 11.5135 |

3. Effect of Thruster Area Ratio:
Current model has the thruster area ratio of 1.2. By increasing the thruster area ratio, nozzle can produce more thrust. When the Thruster Area Ratio changes, mass flow rate, thrust and Mach No at the exit of the thruster also changes. These changing values based on the thruster area ratio are tabulated below,
Table 3. Optimized Various Aspect ratio models Mach Number Values

| MODEL BASED ON THRUSTER AREA RATIO | MACH NO AT THE END OF THE THRUSTER | THRUST (KN) | MASS FLOW RATE (Kg/S) |
|-----------------------------------|------------------------------------|-------------|-----------------------|
| 1.1                               | 1.372                              | 1.403       | 2.02                  |
| 1.2                               | 1.534                              | 1.268       | 1.988                 |
| 1.3                               | 1.659                              | 1.167       | 1.949                 |
| 1.4                               | 1.763                              | 1.089       | 1.915                 |
| 1.5                               | 1.854                              | 1.027       | 1.887                 |

3.5. Selection of Optimized model

The three aspects shown below are considered before concluding the optimized model:

- Thrust Production.
- Weight reduction.
- Low Mass Flow Rate increment due to optimization.

Based on the points mentioned above and with reference to the Tables which has analysed results of the optimized models, the 40% truncated aerospike nozzle and 40% truncated base bleed is chosen to compare with the Full-length aerospike nozzle.

3.5.1. 40% Truncated Model:

For the 40% length external aerospike nozzles, the outlet boundary of the nozzle was created with one endpoint having the coordinates of the last point on the full-length nozzle far field contour and the other endpoint having the x-coordinate of the last point on the 100% length contour and an y-coordinate equal to the y-coordinate of the expansion point.

![Figure 13. Designed 40% truncated aerospike nozzle in GAMBIT](image-url)
Contours of 40% Truncated Nozzle:
Mach Contour

Figure 14. High density grids on the areas where high flow gradient occurs

Figure 15. Mach Contour of the 40% Truncated Aerospike Nozzle

Figure 16. Streamlined Mach Contour of the 40% Truncated Aerospike Nozzle.
3.5.2. 40% Truncated Base Bleed Model:

Since the 40% truncated model is taken as the perfect optimized model, the base bleed model also designed on the 40% truncated model.

![Figure 17. Mach Number Vs Ramp wall plot of the 40% Truncated Aerospike Nozzle](image17)

![Figure 18. Designed 40% truncated base bleed aerospike nozzle in GAMBIT.](image18)

![Figure 19. High density grids on the areas where high flow gradient occurs](image19)
Contours of 40% Truncated base bleed model:

Mach Contour

Figure 20. Mach Contour of the 40% Truncated base bleed Aerospike Nozzle

Figure 21. Streamlined Mach Contour of the 40% Truncated base bleed Aerospike Nozzle.

Figure 22. Mach No Vs Ramp wall plot of the 40% Truncated base bleed Aerospike Nozzle.
4. MANUFACTURING

4.1. FABRICATION MATERIAL:

Stainless steel is an iron-containing alloy. It is a substance made up of two or more chemical elements used in a wide range of applications. It has excellent resistance to stain or rut due to its chromium content, usually from 12 to 20 percent of the alloy. Stainless Steel 304 type material is used to fabricate the design because of its properties listed below.

Raw Material:
Stainless steels are made of some of the basic elements found in the earth: iron ore, chromium, silicon, nickel, carbon, nitrogen, and manganese. Properties of the final alloy are tailored by varying the amounts of these elements. Nitrogen, for instance, improves tensile properties like ductility. It also improves corrosion resistance, which makes it valuable for use in duplex stainless steels.

SS304 Notes:
Type 304 stainless steel is a T300 Series Stainless Steel austenitic. It has a minimum of 18% chromium and 8% nickel, combined with a maximum of 0.08% carbon. It is defined as a Chromium-Nickel austenitic alloy. These are some of its characteristics:
- Forming and welding properties
- Corrosion/oxidation resistance thanks to the chromium content
- Deep drawing quality
- Excellent toughness, even down to cryogenic temperatures which are defined as very low temperatures.
- Low temperature properties responding well to hardening by cold working.
- Ease of cleaning, ease of fabrication, beauty of appearance

Applications:
- Architectural panelling, railings & trim
- Chemical containers, including for transport
- Heat Exchangers
- Woven or welded screens for mining, quarrying & water filtration
- Dyeing industry
- In the marine environment, because of it slightly higher strength and wear resistance than type 316, it is also used for nuts, bolts, screws, and other fasteners.
### Properties

| Property                        | Value               |
|--------------------------------|---------------------|
| Density                        | 8.03 g/cc           |
| Tensile Strength, Ultimate     | 621 MPa (=90100psi) |
| Tensile Strength, Yield        | 290 MPa (=42100psi) |
| Elongation at Break            | 55%                 |
| Modulus of Elasticity          | 193 GPa             |
| Modulus of Elasticity          | 78 GPa              |
| Melt temperature               | 1371- 1399 °C       |

#### 4.2. MANUFACTURING PROCESS:

The whole design of the full length aerospike nozzle was drawn by using CATIA v5 software and the machine draft design chart was prepared by using AutoCAD 2009.

The complete Assembly Model of the aerospike nozzle was shown below,

![Figure 23. Assembly design of Aerospike nozzle.](image)
1. **Spike:**

   • The Spike part of the nozzle was created using CNC machine by installing the machine coding which includes coordinates of the spike.
   • Additional length of 10mm of 12.75mm (1/2”) dia was given for the model to fix with the plate. Clockwise male tapper threading was given on its surface.
   • To create the spike 100X40mm piece of SS304 metal was taken.

2. **Plate**

   • The important part of the nozzle is the plate; it is used to hold the spike.
   • Clockwise female tap thread was created inside the centre hole.
   • similar size 100 degree holes with the outer dia of 35mm. The holes allowed the air to enter the nozzle inlet.
   • Hot wire cutting method is used to cut the metal for inlet holes.
3. **Cowl and plate**

- Initially the cowl was created using CNC machine in order to create the cowl lip arc.
- The disc with the fitting holes created as shown in fig.
- Finally, the cowl is forged with the plate.
4. Flange:
   • 1inch 300# flange: The flange is readily available in all classes. Based on the size of the inlet valve of the wind tunnel and the nozzle operating pressure 300# (class) flange is used here.
   • Since the flange has lesser Outer Dia (34.5mm) than the wind tunnel pressure inlet valve dia (38.4mm). The excess material will be removed from the flange.
Figure 27. Design parameters of flange

5. TESTING OF PROTOTYPE

The fabricated prototype was tested in the pressure wind tunnel and the results were taken out for comparison.

5.1. Wind Tunnel Configuration:

| Operating Parameters | Value |
|----------------------|-------|
| Air storage capacity  | 8000 cubic feet |
| Max. Storage Pressure | 100 Psi |
| Air Compression Rate  | 16 to 18lb/sec |
| Run time              | 15 to 80 sec |
| Nozzle                | 5 feet long |
| No. of adjustable jacks | 12 |
| Mach Number Range     | Transonic .3 to 1.8 |
Supersonic 1.6 to 4.8
Dynamic Pressure Range 150 to 5000 PSF

5.2. Testing

Since it is a small model there are some difficulties on the insertion of the pressure taps. The removal of material inside the spike with the prescribed thickness became tough and the size of the taps available at the labs are 2mm so the number of taps were decreased into three.

1. At the beginning of the spike to measure the pressure at the combustion chamber (here from the pressure tunnel).

2. At the point of 40% of the ramp curve axis length (35.1402mm from throat).

3. Near the end of the spike (the point where the wake started in the computational analysis).

Due to some restrictions the images of fabricated model testing cannot be shown. But the results taken from the experiments were allowed to use and it is used to compare the results taken from the computational methods.

6. RESULTS AND DISCUSSION

Numerical modelling and analysis of internal and external flow of different aerospike nozzle shapes introduced in previous sections are described here. The objective of the analysis is to verify the prescribed exit flow properties of the nozzle design and compare the flow parameters between the nozzle designs.

6.1. Mach Number Plots:

Figure 29 shows typical Mach No plots of Experimental result of the full spike model and FLUENT generated graphs of 100% and 40% truncated and 40% base bleed aerospike nozzles designed for an exit Mach number of 2.601.
Figure 29. Mach No Vs Position of the ramp contour of the Full length & optimized nozzle models.

The maximum Mach number attained of full length aerospike nozzle is 3.82 and 40% truncated aerospike nozzle exit Mach number is 3.54 and the 40% truncated base bleed aerospike nozzle exit Mach number is 4.47.

Although agreement between desired exit Mach number and simulated exit Mach number are below expectations, it is evident that the code is still relatively valid and that adjustment and refinement is needed to more accurately design an aerospike contour for a desired exit Mach number.

6.2. Contours of pressure:

6.2.1. Dynamic Pressure contours:

Figure 30. Dynamic Pressure Vs Position of the ramp contour of the Full length, 40% truncated & 40% truncated aerospike nozzle with base bleed.
Figure 30 shows the dynamic pressure variation throughout the nozzle. It seems that the dynamic pressure over the ramp up to truncated portion is having same value. The sudden decrement in dynamic pressure in 40% nozzle is due to the wake formation on the base and it was rectified by the 40% base bleed model.

7. Conclusion
The procedure to design an aerospike nozzle and design parameters that governs the aerospike nozzle design is discussed here. The designed value of the exit velocity (Mach No 2.5) is achieved with only 2.3% of error. The exit velocity of the air from FLUENT calculation is 2.44.

A comparison between the results of experimental and computational analysis of aerospike nozzle and also the performance of a full-length aerospike nozzle, a nozzle truncated at 60% of the full nozzle length & the same with the base bleed effect were done. For a single flow and boundary condition, the maximum Mach No attained at the end of the 40% truncated & Base bleed spike nozzle is 3.54 & 4.47 respectively.

i) The 60% truncated nozzle produces only a 10.8% lesser thrust than the full-length nozzle. But it reduces nearly 23% of the weight of the full-length nozzle. The performance of the truncated nozzle will be more effective at higher altitudes. Since, the length of the nozzle is very less than the full-length nozzle, it produces considerable amount of performance. These findings are important when it comes to designing air and spacecraft since a nozzle with a lighter weight and equal performance is more attractive than a heavier nozzle.

ii) On the other side, the 40% truncated base bleed produces 14.35% more thrust than the full-length nozzle and it also reduces around 29% of the weight of the full-length nozzle.

But the only constraints of the base bleeding models are its additional mass flow rate. These nozzles need more mass flow rate for their base bleed flow (shown in table 4.2). We can use the base bleed models for high speed rocket engine applications. The diameter of the base bleed model can be modified based on the speed requirement. The diameter changes can be done only in ascending order. Experiments can be done for the optimization models and compared with the FLUENT results.
References

[1] Angelino G., “Approximation Method for Plug Nozzle Design”, AIAA Journal, Vol. 2, No. 10, Oct.1964, pp. 1834-1835.
[2] Lee, C. C., “Computation of plug nozzle contours by the Rao’s optimum thrust method”, NASA CR-21914 R-61, 1963.
[3] Paul V. Tartabini, Roger A. Lepsch, Korte, J. J., and Kathryn E. Wurster, 2000, “A Multidisciplinary Performance Analysis of A Lifting-Body Single-Stageto-Orbit Vehicle”, AIAA-2000-1045.Trans. Roy. Soc. London, vol. A247, pp. 529-551, April 1955.
[4] Verma, S. B., 2009, “Study of Restricted Shock Separation Phenomenon in a Thrust Optimized Parabolic Nozzle”, J. Propulsion and Power.
[5] Shannon D. Eilers, Matthew D. Wilson, and Stephen A. Whitmore, 2010, “Analytical and Experimental Evaluation of Aerodynamic Thrust Vectoring on an Aerospike Nozzle”, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conf. & Exhibit, Nashville, TN.
[6] Eilers S.D., Matthew D., Wilson and Whitmore S.A. (2010), ‘Analytical and experimental evaluation of aerodynamic thrust vectoring on an aerospike nozzle’, Joint Propulsion & International Energy Conversion Engineering Conferences, AIAA-jpc-10.
[7] Li Junwei., Liu Yu., Liao Yunfei and Wang Changhui (2010), ‘Experimental and numerical study on two dimensional plug nozzle’, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA paper 2010-6659, 2010.
[8] Takeo Tomita., Nobuhiko Kumada and Akira Ogawara (2010), ‘A conceptual system design study for a linear aerospike engine applied to a future SSTO vehicle’, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA paper 2010-7060, 2010.
[9] Johnson G.R., Doyle Thompson H. and Hoffman J.D (1974), ‘Design of maximum thrust plug nozzles with variable inlet geometry’, Computers and Fluids, Vol 2, pp 173-190.
[10] Korte J.J. (2000), ‘Parametric model of an aerospike rocket engine’, AIAA paper 2000-1044, 2000.
[11] He Miaosheng, Qin Lizi, Liu Yu (2015), ‘Numerical investigation of flow separation behavior in an over-expanded annular conical aerospike nozzle’, Chinese Journal of Aeronautics, (2015), 28(4): 983-1002.
[12] Johnson G.R., Doyle Thompson H. and Hoffman J.D (1974), ‘Design of maximum thrust plug nozzles with variable inlet geometry’, Computers and Fluids, Vol 2, pp 173-190.
[13] Korte J.J. (2000), ‘Parametric model of an aerospike rocket engine’, AIAA paper 2000-1044, 2000.