# Efficiency Analysis of an Aerospike Nozzle

Dipak J. Choudhari*, Uday V. Asolekar**

*(Department of Mechanical Engg, Vidyavardhini's College of Engg. & Tech Vasai Road (W) 401202
Email : c_dipak@yahoo.com)
** (Department of Mechanical Engg, Vidyavardhini's College of Engg. & Tech Vasai Road (W) 401202
Email: udayasolekar@rediffmail.com)

## ABSTRACT

**This paper explains a new design of nozzles used in aerospace applications called as an aerospike nozzle. We examined analytically the performance of aerospike nozzle with conventional bell shaped nozzles, currently used in all aerospace applications. The result of this study showed that the efficiency of an aerospike nozzle outweighs the efficiency of the bell nozzle at different working altitudes, making it a better option for aerospace applications.**

*Keywords*- **Aerospike nozzle, Computational Fluid Dynamics.**

## NOMENCLATURE:

| | |
|---|---|
| m | Mass flow rate |
| A | Area of flow |
| V | Velocity of flow |
| M | Mach Number |
| C | Velocity of body |
| $C_f$ | Drag Force |
| η | efficiency |
| ε | Expansion ratio |
| θ | Thrust angle |
| $r_b$ | Radius of base of cowl |
| $r_e$ | Radius of cowl lip |
| μ | Mach angle |

## I. INTRODUCTION

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.

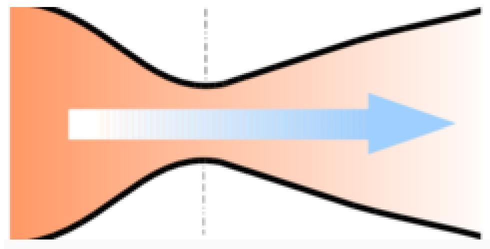

Fig. 1 C-D nozzle.

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.

$$m = \rho * A * V \text{ ----------------(1)}$$

Solving this equation using differentiation, we get the equation,

$$\frac{dA}{A} = \frac{dV}{V}(1 - M^2) \text{ ----------------2)}$$

According to this equation the change in the area (dA/A) changes the sign of the value of Mach number M.

*Mach number* (M): is defined as the ratio of velocity of the body(c) in the fluid medium to the velocity of sound (v) in the fluid medium

$$M = \frac{c}{v} \text{ ----------------(3)}$$

The relation gives the following results:

- For the converging section the change in area is negative, M < 1 (subsonic flow).
- For the flow near the throat change in area is zero, M = 1 (sonic flow).
- For the diverging section the change in area is positive, M > 1 (supersonic).

## II. 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.

Bell nozzles are currently used for all aerospace applications. As stated earlier, the main drawback of this design is the decrease in efficiency with the increase in altitude as there is a loss of thrust in the nozzle. This occurs due to a phenomenon called "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:

$$\eta_{drag} = 1 - \frac{\Delta C_{f(drag)}}{C_{f(ideal)}} \text{ ----------------(4)}$$

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.

$$\eta_{kin} = 1 - \frac{\Delta C_{f(ODK)}}{\Delta C_{f(ODE)}} \text{ ----------------(5)}$$

The overall nozzle efficiency is then given by the combined effects of geometric loss, viscous drag and chemical kinetics: $\eta_{kin\,=}$ 0.99(approximate)

$$\eta_{overall} = \eta_{geo} * \eta_{kin} * (1 - \eta_{drag}) \text{-------(6)}$$

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 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.

## III. 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.

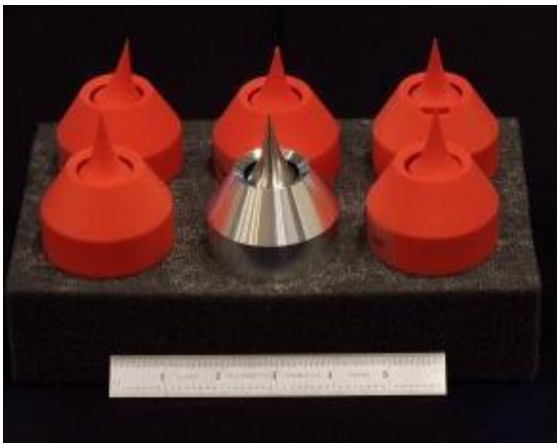

Fig 2 Annular Aerospike nozzle

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.

## IV. WORKING PRINCIPLE OF AEROSPIKE NOZZLE

In an aerospike nozzle, ambient pressure places a limit on the expansion process and the thrust loss associated with nozzle over-expansion does not materialize. Since ambient pressure controls the nozzle expansion, the flow area at the end of the aerospike changes with altitude. A key advantage of an aerospike is that a very high area ratio nozzle, which provides high vacuum performance, can also be efficiently operated at sea level. The primary exhaust can be seen expanding against the center body and then around the corner of the base region. The interaction of this flow with the re-circulating base bleed creates an inner shear layer. The outer boundary of the exhaust plume is free to expand to ambient pressure. Expansion waves can be seen emanating from the thruster exit lip, and these waves reflect from the centerbody contour to the free jet boundary. At low altitude, the free boundary remains close to the nozzle causing the compression waves to reflect onto the center body and shear layer themselves. The waves impacting the center body increase pressure on the surface, thereby increasing the center body component of thrust. The waves impacting the shear layer, on the other hand, increase the circulation of the base flow thereby increasing the base component of thrust.

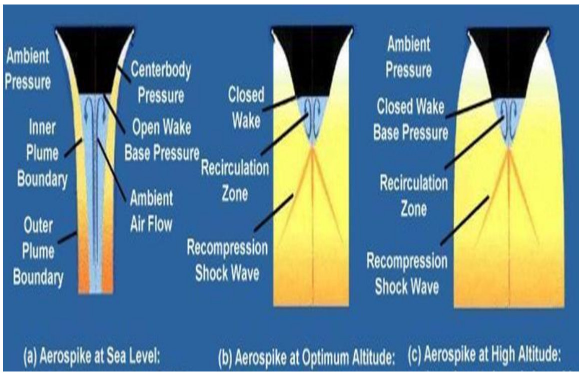


Fig 3 Flow regimes at different altitudes

Ideal behavior results from the fact that the outer plume boundary of the primary flow is acted upon only by the ambient pressure of the atmosphere. From Aerospike thrust characteristics the high ambient pressure at low altitudes forces the exhaust inward increasing the pressure on the "centerbody" and the centerbody component of thrust. In addition, the base region is open to high ambient pressure resulting in a greater "base" thrust component. At design pressure, the flow becomes column shaped, much like a bell nozzle, for maximum efficiency.

## V. DESIGN OF AEROSPIKE NOZZLE

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 a simple approximate method or Rao's method based on calculus of variation. The simple approximate method assumes a series of centered 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:

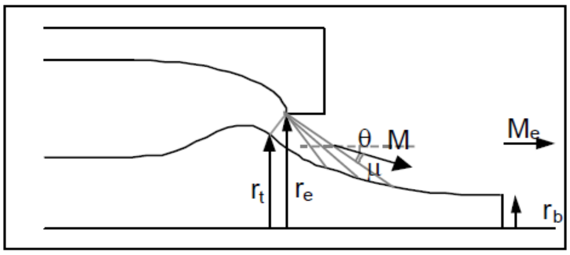


Fig 4 Area at different position.

The expansion ratio is determined for the corresponding pressure ratio from the relation which specifies the variation between the two.

Since the flow is assume to be parallel to the nozzle exit the thruster angle is given by

$$\theta_t = \vartheta(M_e) \quad \text{----------------(7)}$$

Where $M_e$ is Mach number at exit and the prandtl-meyer function $\vartheta$ is given by

$$\vartheta = \sqrt{\frac{\gamma+1}{\gamma-1}} \tan^{-1} \sqrt{\frac{\gamma-1}{\gamma+1}(M^2-1)} - \tan^{-1} \sqrt{M^2-1} \quad \text{-----(3)}$$

The throat area is

$$A_t = \frac{\pi({r_e}^2 - {r_t}^2)}{\cos\theta_t} \quad \text{----------------(8)}$$

And exit area of spike is

$$A_e = \pi({r_e}^2 - {r_b}^2) \quad \text{----------------(9)}$$

Where $r_e$ denotes radius of cowl lip and $r_b$ denotes radius of base $p_c$, the chamber pressure F thrust and $C_f$ the thrust coefficient. Once the expansion ratio is calculated $r_e$ and $r_t$ can be determined for a fixed $A_t$.

The radial co-ordinate of spike nozzle is given by

$$r^2 = {r_e}^2 - \frac{({r_e}^2 - {r_t}^2) A \sin(\mu+\theta)}{A_t \sin(\mu)\cos(\theta_t)} \quad \text{----------------(10)}$$

The axial co-ordinate is given by

$$x = \frac{r_e - r}{\tan(\mu+\theta)} \quad \text{----------------(11)}$$

Where $\theta = \theta_t - \vartheta$ and Mach angle $\mu = \sin^{-1}(\frac{1}{M})$

The relationship for Mach number and area ratio is that given for isentropic expansion of a one dimensional flow through varying area conduit[1].

With the help of a C++ 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.

## VI. DESIGN AND ANALYSIS

The design thus obtained is simulated using computational fluid dynamics software ANSYS FLUENT 12.01 and the post processing of the results, for the first design reveals the flaws in the design and thus the design is modified .The pictures below show the old design, its CFD analysis and the new design.

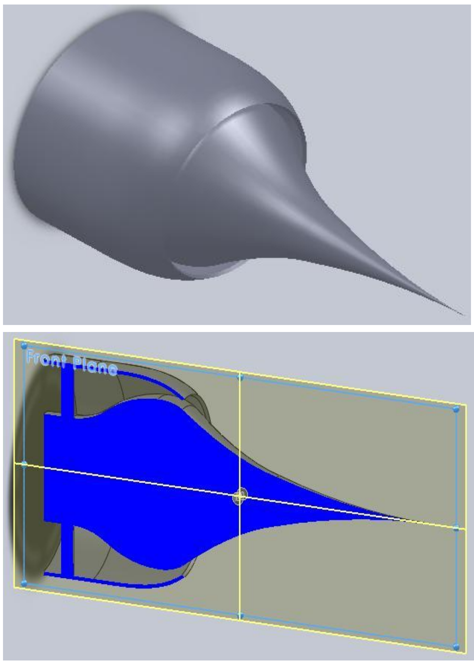


Fig 5 Artistic view of the Nozzle in Design

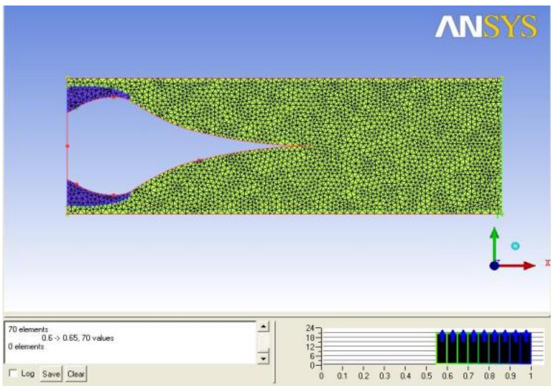


Fig 6 Meshing of the CAD model in ANSYS ICEM 12.01.

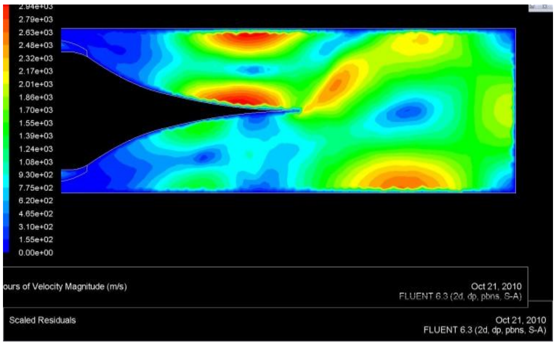


Fig 7 Velocity Contour obtained from FLUENT 6.3.26 shows the flaw in the boundary conditions for the design.

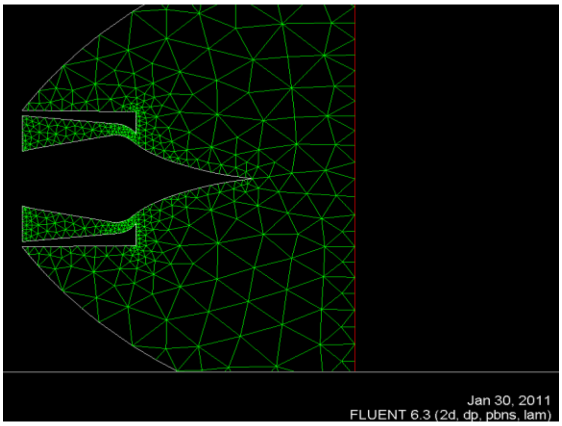


Fig 8 Meshing of the new design done in ANSYS ICEM 12.01.

Many new designs can be simulated through the software and an optimized curve can be obtained thus giving the highest efficiency nozzle design.

## VII. EXPERIMENTATION

To validate the results obtained from CFD analysis the nozzle undergoes a Cold Flow analysis using either Compressed Air or Nitrogen. The compressed air or nitrogen will enter a pressure chamber where the required pressure ratio for the nozzle will be maintained. The nozzle and chamber will be attached together by means of concentric transparent tubes to facilitate easy view of the flow. To increase the visual effects, a dye will be injected. Pressure sensors mounted along the setup will give us the required readings.

## VIII. FUTURE SCOPE

The simplicity of the aerospike design coupled with its altitude compensating properties makes it the best option in aerospace applications. With some more research done into these nozzles it can be seen as a means of propulsion for reusable vehicles in near decade.

## IX. RESULT AND CONCLUSION

Following results obtained from the simulation in Fluent at same boundary conditions.

| Altitude (feet) | Mach No (M) | |
|---|---|---|
| | C-D Nozzle | Aerospike Nozzle |
| 0 (Ground Level) | 1.96 | 1.96 |
| 20000 | 1.99 | 2.4 |

This proves that CD nozzle works best only for the particular designed altitude whereas Aerospike nozzle maintains its efficiency for the changes in the altitude.The results obtained from CFD analysis will be compared with theoretical and experimental values thus accomplishing the validation of the code.

## REFRENCES

[1] Erich Besnard, Hsun Hu Chen, Ton Mueller, *Design Manufacturing and test of a plug rocket nozzle*, Mechanical and Aerospace engineering department, California State University, Long Beach.

[2] J.J, Korte, *Parametric Model of an Aerospike Rocket Engine*, NASA Langley Research Centre, Hampton, Virginia.

[3] Klaus W.Gross, *Performance Analysis Of Aerospike Rocket Engines.*

[4] Brandon Lee Denton, *Design And Analysis of Rocket Nozzle Contours For Launching Pico-Satellites*, Department of Mechanical Engineering, Kate Gleason College Of Engineering, Rochester Institute Of Technology, NY.

[5] Maciej Matyka, *Prandtl-Meyer Expansion Wave*, Computational Physics Section of Theoretical Physics, University of Wroclaw.

[6] John Cipolla, AeroRocket and Warpmetrics http://aerorocket.com/MOC/MOC.html

[7] Chang Hui Wang, Yu Liu, Li-Zi Qin, *Aerospike Nozzle Contour Design and Its Performance Validation, Beijing* University Of Aeronautics and Astronautics, Beijing.