Performance Analysis of Linear Aerospike Nozzle Using Computational Study

C. Ted Solomon[1], Dr. P. Vasanthakumar[2]
[1]Dept of Aeronautical Science
[2]Professor, Dept of Aeronautical Science
[1, 2] Hindustan Institute of Technology & Science,
Padur Campus, Kelambakkam, Chennai – 603 103, Tamil Nadu, India

Abstract- *As we all know the importance and need for space transportation in the upcoming days. the requirement for launch vehicle has increased drastically. So, SSTO based launch vehicles are required in order to achieve our missions in a cost effective and affordable access to space. The space race has began and everyone has started its development for future transportation launch vehicle. The purpose of this project work is to model an linear aerospike nozzle to study the performance and flow characteristics of an truncated aerospike nozzle with consideration to major problems such as weight and heat which affects the efficiency of the nozzle. Also the performance comparison with respect to various altitude is done. The results of this study are done using the computational fluid dynamics software ANSYS-Fluent. The flow parameters such as pressure, velocity will be used to the analysis the performance of the nozzle.*

Keywords- SSTO, Launch vehicle, Linear aerospike, ANSYS.

I. INTRODUCTION

1.1 Introduction

It was shortly ago when the aerospike nozzle was unknown to any or all but rocketry experts. except for a spate of research within the early 1970s, the concept of the plug nozzle looked as if it would be nothing quite a footnote in aerospace history. Though explored by the Germans as early as war II to be used in jet engines, it absolutely was not until the revealing of Lockheed Martin's winning X-33 concept that this sort of engine has received widespread attention. The nozzle is that the component of a rocket or air-breathing engine that produces thrust. This is often accomplished by converting the thermal energy of the recent chamber gases into mechanical energy and directing that energy along the nozzle's axis, as illustrated below.

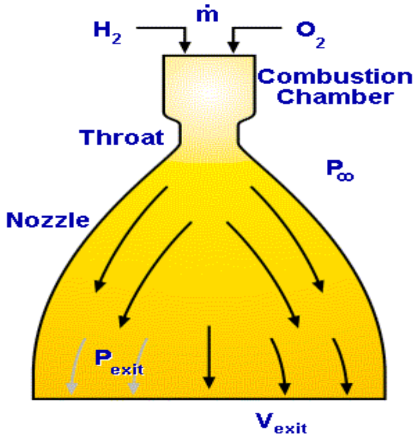


Fig 1.1 Simple representation of a rocket nozzle [from Rocketdyne, 1999]

This simple figure illustrates how a rocket nozzle works. The propellant consists of a fuel, typically liquid hydrogen (H2), and an oxidizer, typically atomic number 8 (O 2). The propellant is pumped into a combustion chamber at some rate (mdot) where the fuel and oxidizer are mixed and burned. The exhaust gases from this process are pushed into the throat region of the nozzle. Since the throat is of less cross-sectional area than the remainder of the engine, the gases are compressed to a air mass. The nozzle itself gradually increases in cross-sectional area allowing the gases to expand. because the gases do so, they push against the walls of the nozzle creating thrust.

Mathematically, the nozzle is to expand the gases as efficiently as possible so as to maximize the exit velocity (v_{exit}). This process will maximize the thrust (F) produced by the system since the two are directly related by the equation

$$F= mdot\ Vexit + (P_{exit} - P_{\infty})\ A_{exit}$$

F = thrust force

mdot = mass flow rate

v_{exit} = exhaust gas velocity at the nozzle exit

p_{exit} = pressure of the exhaust gases at the nozzle exit

p_∞ = ambient pressure of the atmosphere

A_{exit} = cross-sectional area of the nozzle exit

- ### 1.1.1 Expansion Area Ratio:

In theory, the sole important parameter in rocket nozzle design is that the expansion area ratio (ε), or the ratio of exit area (A exit) to throat area (A throat).

$$\varepsilon = A_{exit} / A_{throat}$$

Fixing all other variables (primarily the chamber pressure), there exists just one such ratio that optimizes overall system performance for a given altitude (or ambient pressure). However, a rocket typically doesn't travel at just one altitude. Thus, an engineer must remember of the trajectory over which a rocket is to travel so an expansion ratio that maximizes performance over a spread of ambient pressures is selected. Nevertheless, other factors must even be considered that tend to change the planning from this expansion ratio-based optimum. a number of the problems designers must accommodate are nozzle weight, length, manufacturability, cooling (heat transfer), and aerodynamic characteristics.

1.2 Rocket Nozzle Shapes

Not all rocket nozzles are alike, and also the shape selected usually depend on the applications. This section discusses about the fundamental characteristics of the key classes of nozzles used today.

1.2.1 Nozzle Comparisons:

To date three major varieties of nozzles, the cone, the bell or contoured, and also the annular or plug, are employed. Each class satisfies the previously discussed standard to that of the varying degrees. Examples of these nozzle and its types are seen below.

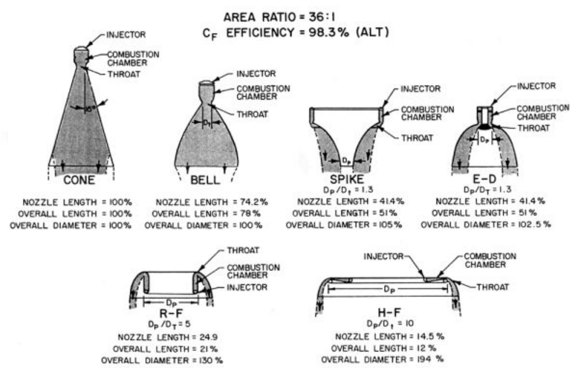


Fig 1.2 Size comparison of optimal cone, bell, and radial nozzles for a given set of conditions [from Huzel and Huang, 1967]

1.3 Aerospike Nozzles

We discussed methods of reducing the length of a nozzle center body by replacing the perfect spike with a conical spike. While this method does indeed result during a much shorter nozzle length, we are able to go even further by removing the pointed spike altogether and replacing it with a flat base. This configuration is understood as a truncated spike, an example is shown below.

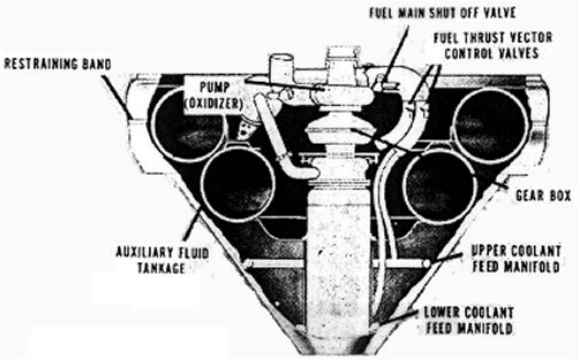


Fig 1.3 Example of a truncated, conical spike [from Berman and Crimp, 1961]

As any fluid dynamists recognizes, the numerous disadvantage of the "flat" plug is that a turbulent wake forms aft of the bottom at high altitudes leading to high base drag and reduced efficiency. However, this problem is greatly alleviated in an improved version of the truncated spike that introduces a "base bleed," or secondary subsonic flow, into the region aft of the bottom.

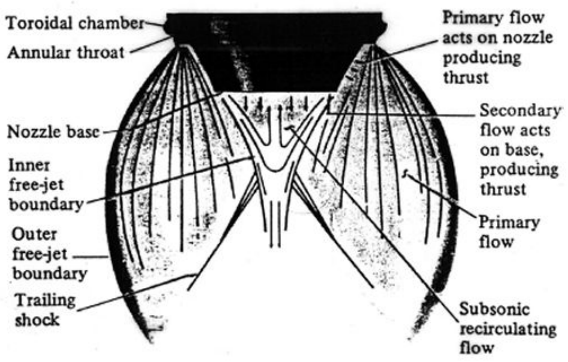


Fig 1.4 Flow visualization in truncated aerospike nozzle

The circulation of this secondary flow and its interaction with the engine exhaust creates an "aerodynamic spike" that behaves very similar to the best, isentropic spike. additionally, the secondary flow re-circulates upward pushing on the bottom to provide additional thrust. it's this artificial aerodynamic spike that the aerospike nozzle.

1.4 Linear Aerospike

All of the nozzles we've studied to this point are annular, or circular when viewed from below. Still another variation of the aerospike nozzle isn't an annular nozzle the least bit. A second approach, pioneered by the Rocketdyne company (now a division of Boeing) within the 1970s, places the combustion chambers during a line along two sides of the nozzle:

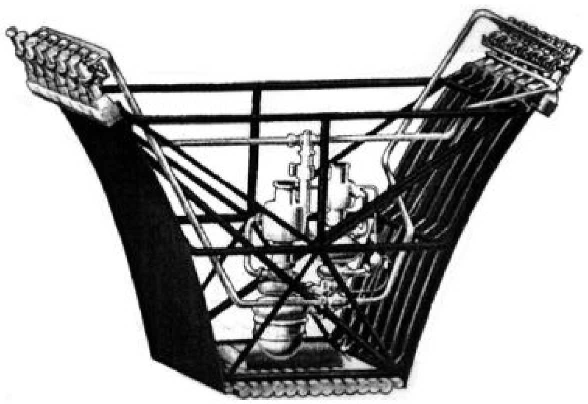


Fig 1.5 Rocketdyne RS-2200 linear aerospike engine [from Flinn, 1996]

This approach results in a more versatile design allowing the use of lower-cost modular combustors. These modules can be combined in varying configurations depending on the application.

II. LITERATURE REVIEW

Mehdi Nazarini et al., (2005) The objective of the present study to address practical aspects of using an aerospike nozzle, effects of geometric parameters including base curvature, and different values of plug truncation ranging from 0 to 75%, on thrust and base temperature distribution of the nozzle have been studied..Based on the observed behavior of the thrust delivered by aerospike nozzles with different amounts of plug truncation, it can be concluded that selection of the amount of plug truncation depends on the flight regime of the vehicle which will use this propulsion system.

Vinay Kumar Levaka et al., (2014) The paper simulation results shows that the base pressure compensates the loss of thrust in under-expansion conditions, plug truncation has minor effect on the loss of thrust in these conditions. But in over-expansion, thrust loss will increase with the increase of truncation. Base pressure thrust is closely related to variation of base pressure with atmospheric pressure. Base pressure is constant in under expansion conditions, but increase with the increase of the atmospheric pressure in over-expansion conditions. Based on the observed behavior of the exhaust flow, it can be concluded that the 40 % truncated nozzle is recommended.

Naveen Kumar K et al., (2017). A comparison between the results of experimental and computational analysis of aerospike nozzleand 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 nozzleis 3.54 & 4.47 respectively.

Md Saquib Reza et al., (2017) This objective of this Paper has been excavating and investigating two different types of Aerospike nozzles such as external aerospike nozzle and internal-external aerospike nozzle. Analysis has been carried out for both types of nozzles with different truncations. However, in this paper the fuel was taken as Papi 94 and Hydroxyl-terminated poly butadiene (HTPB) with N_2O as an oxidant and the combustion products were entered in FLUENT by selecting species transport. The results shows the efficiency.

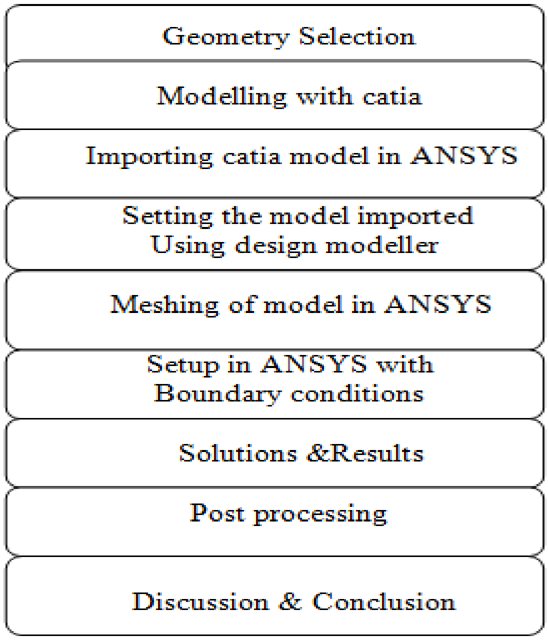
Chang-HuiWang et al., (2009) In this paper both the experimental and numerical studies were performed on a 6-celltile-shaped aerospike nozzle, a1-cell linear aerospike nozzle and a 3-cell aerospike nozzle with RTR primary nozzles designed by method proposed in the paper. In cold-flow tests, 6-celltile-shaped aerospike nozzle and 1 cell linear aerospike nozzle obtained high thrust efficiency at design altitude. Employing GH2/GO2 as propellants, hot-firing tests were carried out on a 3-cell aerospike nozzle engine with RTR primary nozzles. Near NPR = 50, Efficiency is 92.0-93.5% and efficiency is 95.0–96.0% near NPR = 350. The promising efficiency has been achieved more than 98.0% at design point can be expected.

III. METHODOLOGY

3.1 Introduction

The methodology adopted and research design taken under consideration will be discussed in this chapter. The details of the computational studies adopted are discussed below

3.2 Methodology Used



3.3 Geometry

Geometry of the model are the intial values which are used to model the aerospike nozzle and the design softwares such as catia software is used to model in this work. The geometry of the model are taken from the Nasa datasheet report of XRS-2200 aerospike engine. The geometry values are given in the below section.

The following specification are used for designing the aerospike nozzle are below:

Table 3.1 Geometry Sizing

Parameters	Values
Length of spike	1811.01mm
Exhaust Radius	1701.8mm
Base Radius	533.4mm
Length of thruster	400mm
Radius of thrusters	200mm
Throat of radius	200mm
Throat angle	30.3°

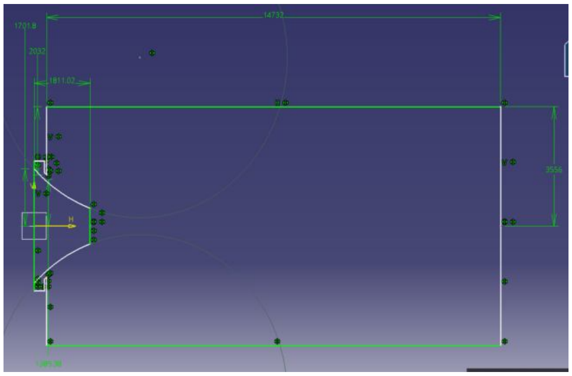


Fig 3.1 2D Aerospike model with all dimensions

3.4 Boundary condition

The following design parameters are considered for geometry

Table 3.2 Boundary conditions

Boundary Name	Boundary Type
Coaxial jet surface	Wall(Domain)
Circular coaxial jet	Symmetry
Geometry	3-D Model
Model	Viscous K-epsilon Model
Material	Fluid – Air Solid - Aluminium
Boundary Conditions	Primary Fluid Secondary Fluid Wall (Domain)
Chamber pressure	2067857 N/m^2
Design Altitude	3657.6 m
Mass flow rate	3.25758 kg/s
Temperature	300K

3.5 Catia model with Geometry

By using the design specification a 2D aerospike model has been modelled using catia and it is as below:

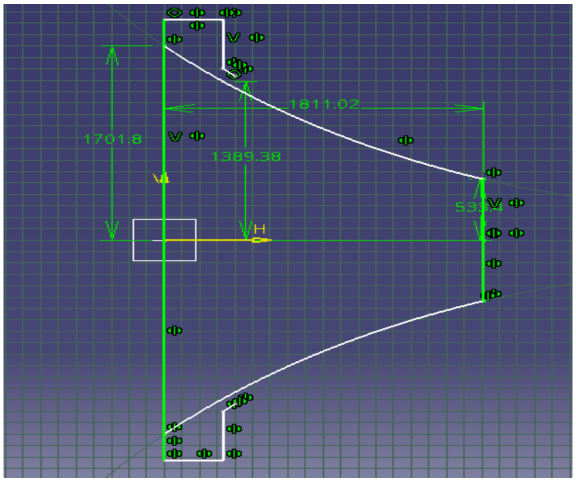


Fig 3.2 2D Aerospike model with dimensions

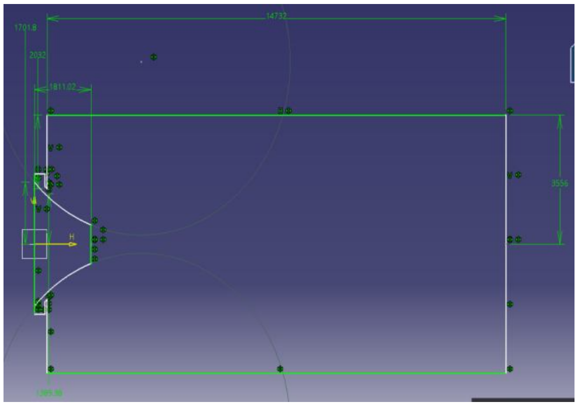


Fig 3.3 2D Aerospike model with domain

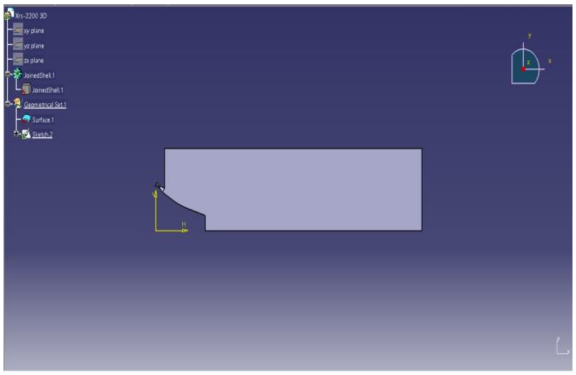


Fig 3.4 2D Axisymmetric Aerospike model with domain

3.6 Ansys Import Geometry

Ansys import of the aerospike model with the geometry is imported into the ansys and it is as below:

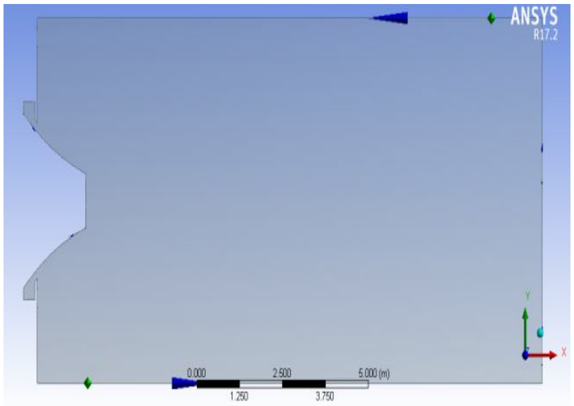


Fig 3.5 2D Aerospike model imported into Ansys

Fig 3.6 2D Axisymmetric Aerospike model imported into Ansys

3.7 Mesh Generation with ANSYS

Mesh Generation of the aerospike model which is designed in catia is meshed as below:

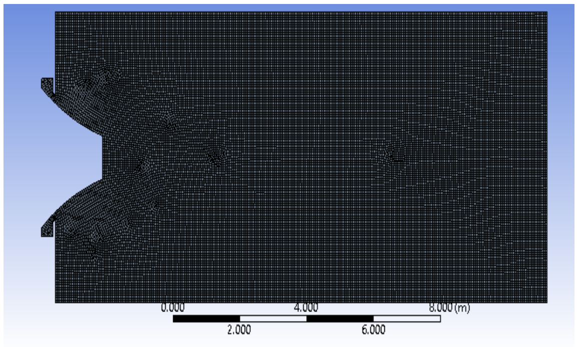


Fig 3.7 Mesh generation of model

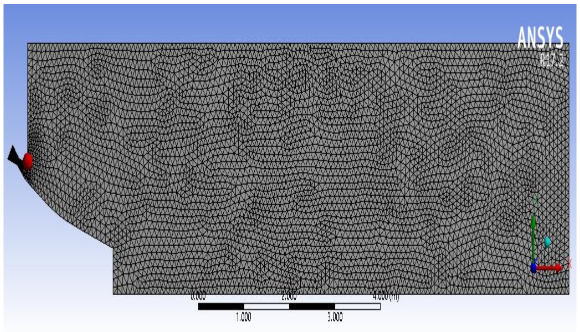


Fig 3.8 Mesh generation of 2D Axisymmetric model

Table 3.3 Number of Nodes & Elements

Geometry	Nodes	Elements
Aerospike nozzle	654981	753305
Axi Aerospike nozzle	91996	464378

IV. RESULTS

4.1 Numerical simulation using Ansys Fluent

In this section, flow pattern of the Linear aerospike nozzle with different working boundary conditions in this section. The boundary conditions such as Inlet, far field & outlet conditions are given. The flow parameters which is produced by the aerospike nozzle at the outlet are studied using the pressure, velocity and temperature using which the performance of Linear Aerospike nozzle is obtained. With the numerical simulation of the Ansys fluent using the post processing the contours for the flow parameters to study performance of the aerospike nozzle is determined.

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.1, the maximum velocity flow is 2.368x10^{3}m/s.

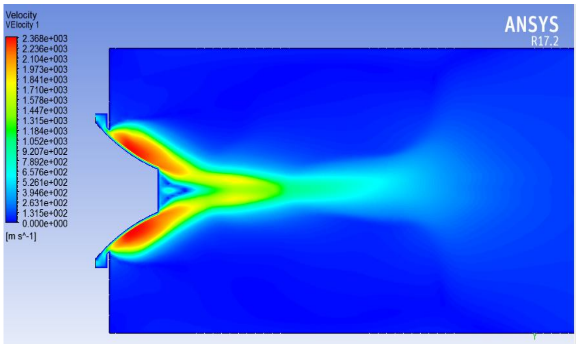


Fig 4.1 Velocity Contour 2D Aerospike model (Non C-D thruster)

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.2, the maximum velocity flow is 4.23x10^{3}m/s.

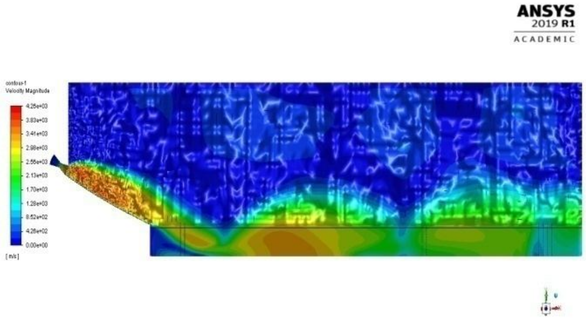


Fig 4.2 Velocity Contour 2D Axisymmetric Aerospike model (with C-D thruster)

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.3, the maximum velocity flow is 4.40x10^{3}m/s.

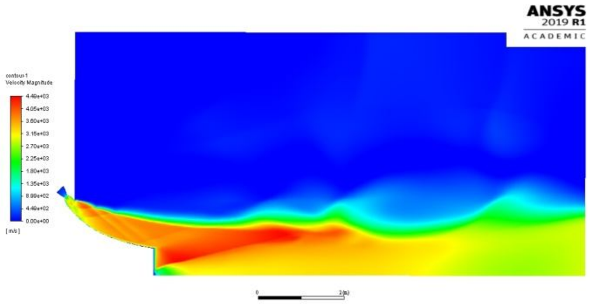


Fig 4.3 Velocity contour C-D section in the nozzle with 0.04m throat

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.4, the maximum velocity flow is 3.82x10^{3}m/s.

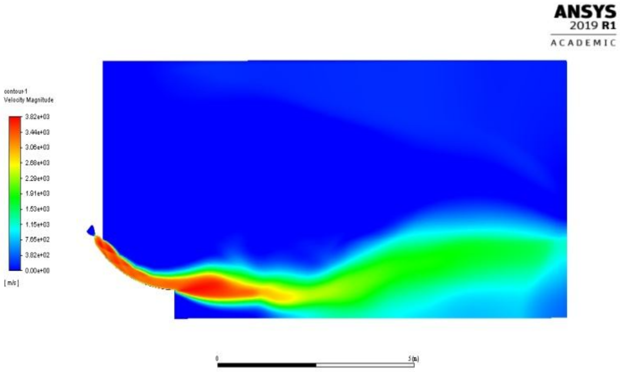


Fig 4.4 Velocity contour C-D section in the nozzle with 0.08m throat

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.5, the maximum velocity flow is $4.17x10^3$m/s.

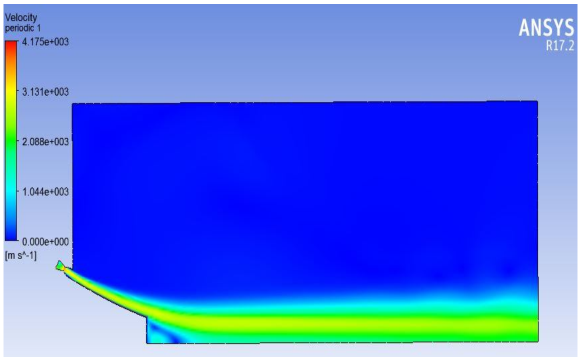


Fig 4.5 Velocity contour C-D section in the nozzle with 0.10m throat

The Velocity Contour for the symmetry Aerospike nozzle is shown in Figure 5.6, the maximum velocity flow is $4.51x10^3$m/s.

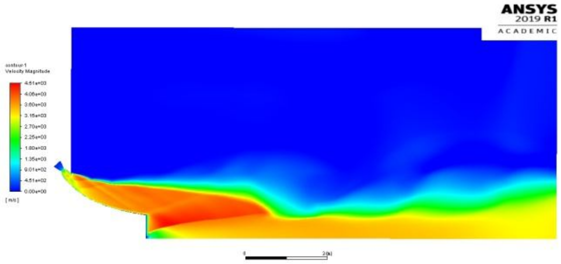


Fig 4.6 Velocity contour C-D section in the nozzle with 0.12m throat

The Pressure Contour for the symmetry Aerospike nozzle is shown in Figure 5.7, the maximum velocity flow is $5.94x10^6$pascal.

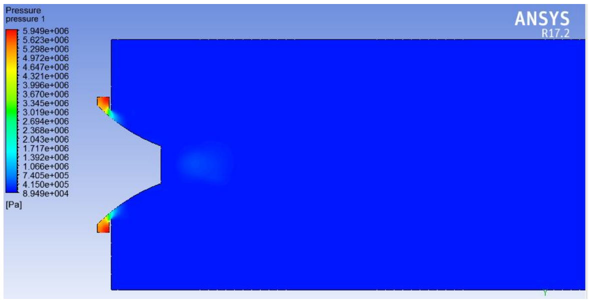


Fig 4.7 Pressure Contour 2D Aerospike model (Non C-D thruster)

The Pressure Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.8, the maximum velocity flow is $6.12x10^6$pascal.

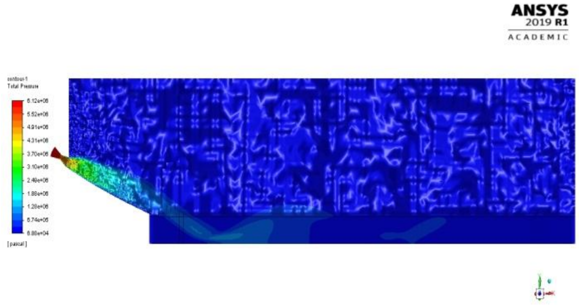


Fig 4.8 Pressure Contour 2D Axisymmetric Aerospike model (with C-D thruster)

The Pressure Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.9, the maximum velocity flow is $6.76x10^6$pascal.

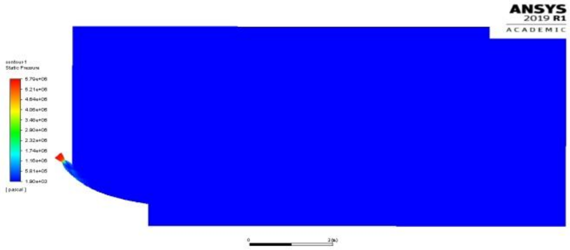


Fig 4.9 Pressure contour C-D section in the nozzle with 0.04m throat

The Pressure Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.10, the maximum velocity flow is $6.79x10^6$pascal.

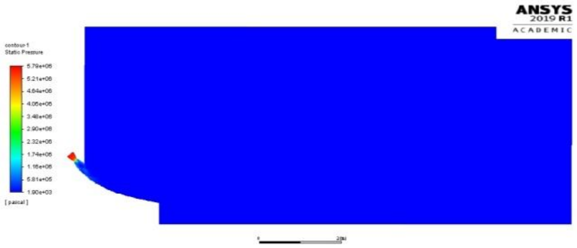


Fig 4.10 Pressure contour C-D section in the nozzle with 0.08m throat

The Pressure Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.11, the maximum velocity flow is $5.79x10^6$pascal.

Fig 4.11 Pressure contour C-D section in the nozzle with 0.10m throat

The Pressure Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.12, the maximum velocity flow is 6.79x10^{6}pascal

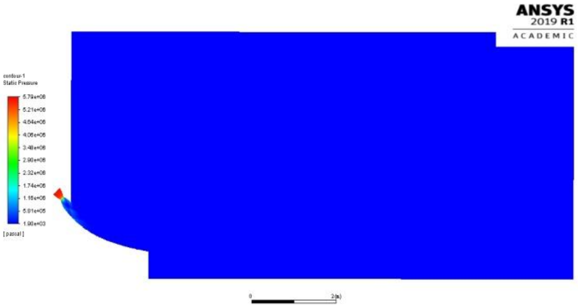


Fig 4.12 Pressure contour C-D section in the nozzle with 0.12m throat

The Temperature Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.13, the maximum temperature is 5.23x10^{3}K

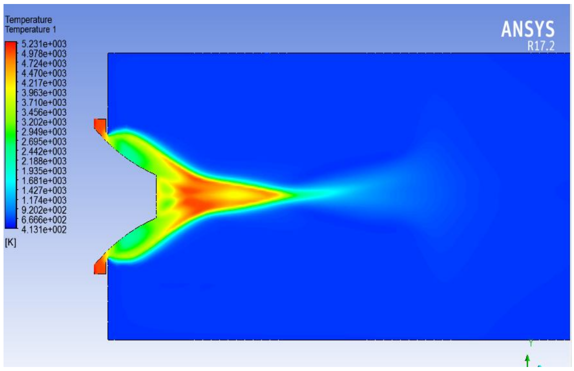


Fig 4.13 Temperature Contour 2D Aerospike model (Non C-D thruster)

The Temperature Contour for the Axi symmetry Aerospike nozzle is shown in Figure 5.14, the maximum temperature is 3.86x10^{3}K

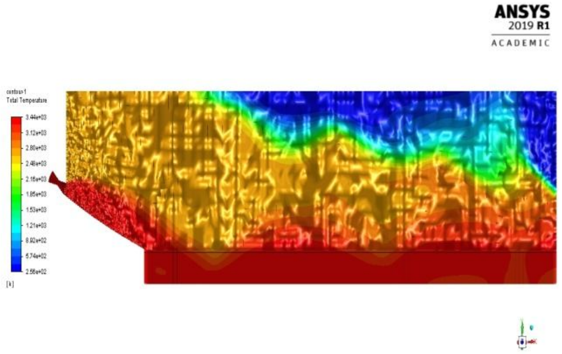


Fig 4.14 Temperature Contour 2D Axisymmetric Aerospike model (with C-D thruster

The Velocity Contour for C-D Thruster section of Aerospike nozzle is shown in Figure 5.15, the maximum velocity flow is 4.49x10^{3}m/s.

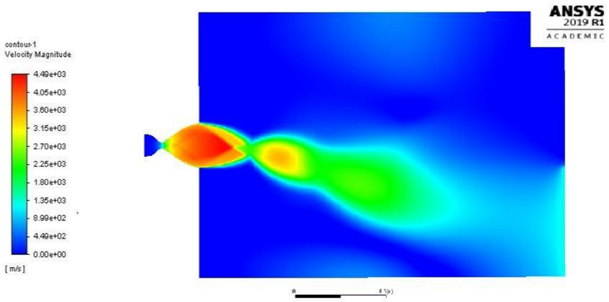


Fig 4.15 Velocity Contour for C-D Thruster section of Aerospike model

The Pressure Contour for C-D Thruster section of Aerospike nozzle is shown in Figure 5.16, the maximum pressure flow is 5.84x10^{6}pascal

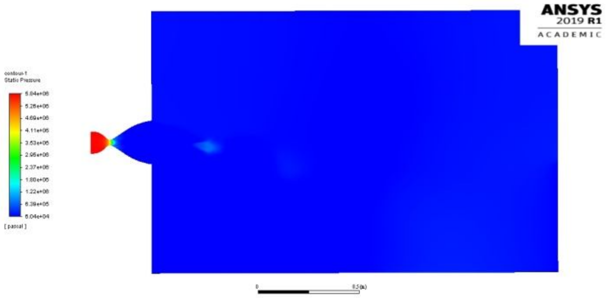


Fig 4.16 Pressure Contour for C-D Thruster section of Aerospike model

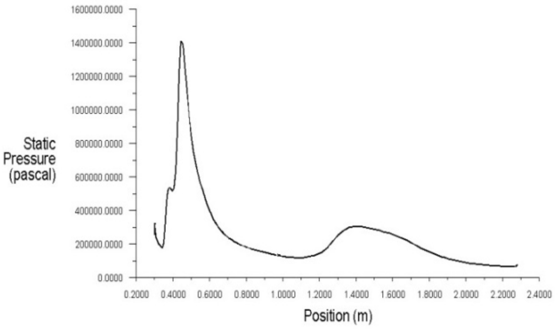


Fig 4.17 Pressure plot for C-D Thruster throat 0.04m

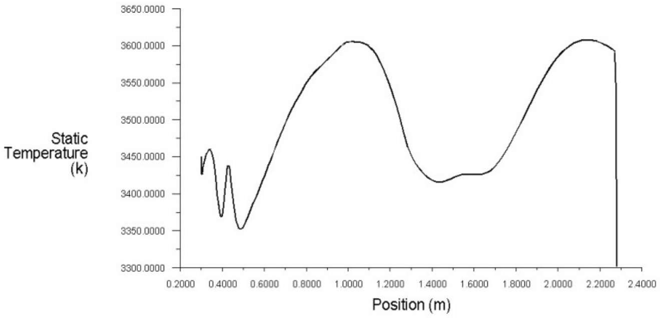


Fig 4.18 Temperature plot for C-D Thruster throat 0.04m

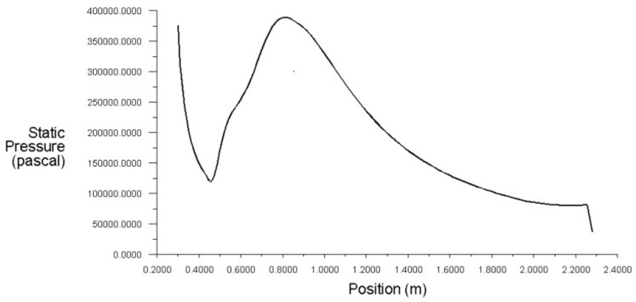


Fig 4.19 Pressure plot for C-D Thruster throat 0.08m

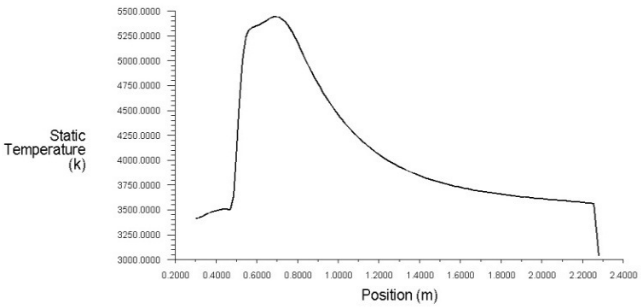


Fig 4.20 Temperature plot for C-D Thruster throat 0.08m

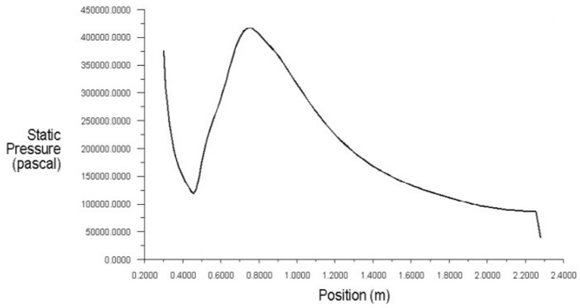


Fig 4.21 Pressure plot for C-D Thruster throat 0.12m

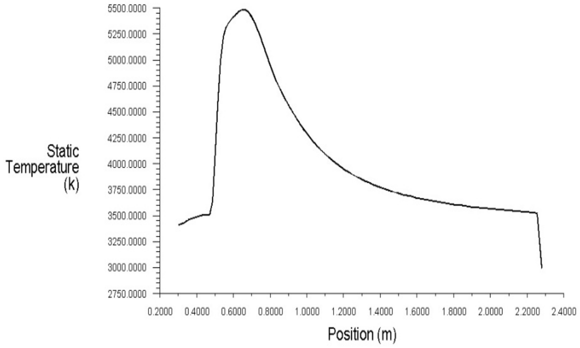


Fig 4.22 Temperature plot for C-D Thruster throat 0.12m

V. DISCUSSION

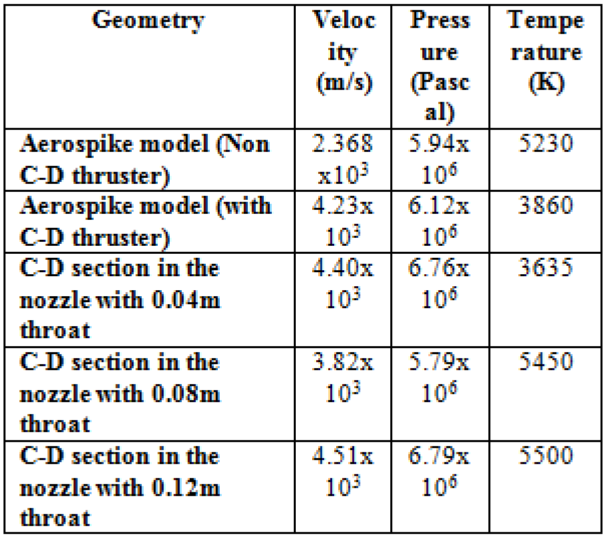
5.1 Introduction

This chapter discusses the comparison between the Aerospike model with various throat dimension for the C-D section of thruster for constant boundary condition. The numerical results of the aerospike model for pressure, velocity and the graphs is been plotted for the pressure and temperature for the models were done.

5.2 Comparison of Results obtained for the 2D Aeropike model

The pressure, velocity and temperature of 2D Aerospike model for the aerospike nozzle the numerical results obtained from the computational study are as tabulated below for the maximum value of the each parameter studied using the contours and graphs obtained. The results obtained from the 2D Aerospike model for the boundary condition and results obtained are as follows.

Table 5.1 Results of 2D Aerospike model obtained maximum Pressure, Velocity, Temperature



Geometry	Velocity (m/s)	Pressure (Pascal)	Temperature (K)
Aerospike model (Non C-D thruster)	2.368×10^3	5.94×10^6	5230
Aerospike model (with C-D thruster)	4.23×10^3	6.12×10^6	3860
C-D section in the nozzle with 0.04m throat	4.40×10^3	6.76×10^6	3635
C-D section in the nozzle with 0.08m throat	3.82×10^3	5.79×10^6	5450
C-D section in the nozzle with 0.12m throat	4.51×10^3	6.79×10^6	5500

5.3 Conclusion

As I started working on aerospike it was really challenging and helped to understand better about the performance of the aerospike nozzle. To know the basic knowledge required to model a aerospike nozzle in CATIA and analysis it using ANSYS with the existing Geometry, Inlet and Boundary condition.

Flow characteristics and the performance of truncated aerospike nozzles in design conditions has been compared with the existing results. The results are positive as per the inlet and boundary conditions given in the ANSYS. As the velocity increases at the exit the velocity increases in clearly understand from the results attained using the contours. The velocity and pressure are the parameters which was only considered to the know the performance of the aerospike nozzle.

Based on the observation it states that the results achieved are comparatively good than the currently used C-d and Bell nozzles. Thus the usage of aerospike nozzle in the launch vehicle gives the attiude compensation and can improve the fuel efficiency.

VI. SUMMARY

As I started working on aerospike it was really challenging and helped to understand better about the performance of the aerospike nozzle. To know the basic knowledge required to model a aerospike nozzle in CATIA and analysis it using ANSYS with the existing Geometry, Inlet and Boundary condition.

VII. FUTURE WORK

- To write a matlab code inorder to generate the contours for the Linear aerospike nozzle.
- To find the optimimum length of the aerospike nozzle with a better performance charsacteristics
- Analysing the performance of the aerospike nozzle at various attitudes.
- Comparing the performance with various parameters to understand more better performance.

REFERENCES

[1] A. Naghib-Lahouti, M. Nazarinia and E. Tolouei, **“**Design, numerical flow analysis and investigation of the effects of geometric parameters on performance of an aerospike nozzle in off-design conditions**”**, Technical Report No. ARI-83-21-ASN1-4-2-1, Aerospace Research Institute, Tehran, Iran, September 2004.

[2] Korte, J., J. Dunn, A. Salas, N. Alexandrov, W. Follett, G. Orient and A. Hadid, “Multidisciplinary Approach to Linear Aerospike Nozzle Optimization”, 33rd Joint Propulsion Conference, Seattle, USA, July 1997.

[3] Md Saquib Reza, Konark Arora, “Contour Design of Aerospike Nozzle and Comparison of Performance”, International Conference on Infocom Technologies and Unmanned Systems, Nov 2017

[4] Naghib-Lahouti, A., Nazarinia, M., and Tolouei, E., “Design and CFD Analysis of anAerospike Nozzle to Compare Its Off-Design Performance with a Conventional Nozzle”,Proceedings of IMEC2004, International Mechanical Engineering Conference, IMEC2004-FM014-CP, December 5-8, 2004, Kuwait.

[5] Vinay Kumar Levaka, Srinivasa Reddy K, “Design and flow simulation of truncated aerospike nozzle”, International Journal of Research in Engineering and Technology eISSN: 2319-1163| pISSN: 2321-7308 Volume: 03 Issue: 11 | Nov-2014

[6] Tomita, T., Tamura H. and Takahashi M., “An Experimental Evaluation of Plug Nozzle Flow Fields”**,** AIAA Paper 96-2632, 1996.

[7] Chang-Hui Wang, Yu Liu, Li-Zi Qin, “Aerospike nozzle contour design and its validation”, 403 Teaching and Research Section, Beijing University of Aeronautics and Astronautics, 100083 Beijing, People’s Republic of China, July 2009.

[8] Wang Chang-hui, LIU Yu, LIAO Yun fei, “Studies on Aerodynamic Behavior and Performance of Aerospike Nozzles”, Vol. 19 No. 1 Chinese journal of aeronautics, Feb-2006