Flow Field Characteristics of an Aerospike Nozzle in Over-expansion and Under-expansion Conditions

Ava Shahrokhi and Sahar Noori**

** Faculty member*

Aerospace Research Institute,
Ministry of Science, Research and Technology, Tehran, Iran

Abstract
The flow properties of an aerospike nozzle are investigated in different back pressures utilizing k-ω turbulence model. The shape of the plug is determined through approximate method for design conditions. The flow field is simulated using Navier-Stokes equations for compressible flows. A finite volume cell centered scheme is used to discretize the flow field equations. To accelerate the solution convergence rate, the flow field is divided into several zones. Each zone is facilitated with proper unstructured grid and the appropriate initial conditions.

1. Introduction

Conventional bell nozzles suffer from reduced engine performance i.e. overexpansion and under expansion at high and low altitudes correspondingly. To achieve relatively better performance, the aerospike engine has been variously studied from early 60^{th}. The gas expansion process within the aerospike nozzle is largely controlled by the ambient pressure (or back pressure) of the surrounding air. In theory, the optimal thrust for a given jet or rocket engine occurs when the nozzle exit pressure equals surrounding pressure.

The aerospike engine, maintains its aerodynamic efficiency across a wide range of altitudes through using an aerospike plug. Various investigations conducted by scientists on aerospike nozzles. The influence of annular conical spike nozzle was studied by researchers and the performance of aerospike nozzles was compared with conical plug [1]. Computational heat transfer analyses have been conducted to study the effect of the fence on the base heating of an X-33 aerospike engine [2]. As documented in the aerospace literature, the flow in supersonic nozzles is substantially influenced by viscous effects which may reduce the performance of the propulsion system [3]. Koutsavdis applied several turbulence models for computation of a plug nozzle flow field and compared the results with experimental observations [3]. According to Koutsavdis studies about the influence of the flow separation around the boot tail of the truncated nozzle, the k-ω turbulence model appears to be the model of choice for prediction of most of the characteristics of the flow on the full nozzle. Authors' previous studies on different turbulent schemes in aerospike nozzles proved the influence of turbulence consideration in the accuracy of the thrust calculation [4].

The focus of the present study is on the flow field computation of an Aerospike nozzle in design, over expansion and under expansion conditions using k-ω turbulent model. The thrust force is determined for each case as well as pressure and Mach contours and the flow characteristics are investigated in all cases.

2. Geometry of the plug nozzle

The contours of the aerospike nozzle have been designed by approximate method. The approximate method drives a set of analytical relations, which can be used to calculate radius and flow properties at each longitudinal station of the plug in the case where no shockwave occurs along the spike. Figure (1) shows the aerospike nozzle components which include rocket thruster, cowl, aerospike nozzle and plug contour.

The method presented in this section is implemented to design an aerospike nozzle in the 4-5 KN thrust class, with specifications originally introduced in [5], chamber pressure is 2067857 N/m^2 and the optimum design altitude is 3657.6 m in which the optimum thrust of 4443.9 N is achieved. The specific impulse and mass flow rate are 235 s and 3.25758 kg/s respectively. The propellant is ethanol- oxygen with the specific heat ratio of 1.21 and C_P = 1286.68 J/kgK.

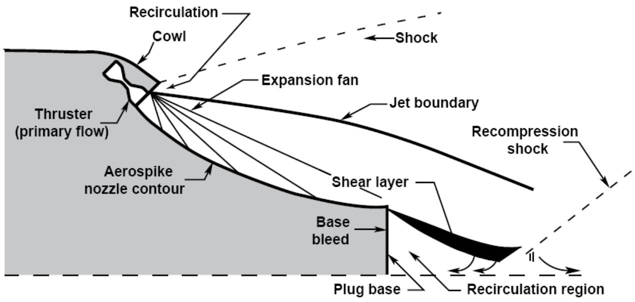

Figure 1: Schematic of the aerospike nozzle components

3. Numerical method

The numerical analysis of the internal and external flow of the aerospike nozzle is performed using Navier-Stokes equations considering turbulent flow. The viscose terms are evaluated by the central differencing and the eddy viscosity is modelled by k-ω method. The flow is assumed to be axisymmetric although three-dimensional equations are used in this study. The computational methodology utilizes a steady state density-based formulation and a finite volume cell centered scheme to discretize the flow field equations. Each zone is facilitated with proper unstructured triangle grid and the appropriate initial conditions. Providing the control over the grid density and initial conditions in different zones is helpful in accelerating the solution convergence.

4. Results and discussions

The results are obtained in three cases a) design conditions of the nozzle where the exit pressure equals the ambient pressure, b) over-expansion condition where exit pressure is greater than the ambient pressure c) Under-expansion condition where the exit pressure is smaller than the ambient pressure. Figures 2-4 illustrate the Mach contours for these cases.

As illustrated in Figure 1, the exhaust flow of the aerospike nozzle forms a series of expansion waves which originate from the upper lip of the convergent section. Since the exhaust flow is not bounded by a solid wall, these expansion waves adjust their intensity and domain to match the exhaust flow with the external flow.

In optimum condition the domain covered by expansion waves ends at the end of the plug. At that station, flow properties match with those of the external flow and no considerable further expansion or compression is encountered. The results for this case are shown in Figure 2.

In over-expansion conditions, the domain covered by expansion waves ends before the end of the plug. At that station, flow properties are close to those of the optimum condition, which usually involve a higher Mach number and a lower pressure compared to the external flow. From this station onwards, flow encounters reflection of the expansion waves in form of a series of compression waves, which increase the pressure and reduce Mach number to a value close to that of the external flow. The process mentioned above can be noticed in Figure 3 in the form of contours of Mach number where P_{atm}/P_{des} equals 4.0. After leaving the nozzle, flow is compressed through a series of compression waves originating from the edge of the exhaust surface, which resemble converging shock waves. These compression waves contract the flow and impose a radial velocity component which contributes to loss of thrust in over-expansion conditions.

For an aerospike nozzle in under-expansion conditions, the domain of the expansion waves covers an area larger than the plug itself, and the flow continues to expand after the plug. The effect of these waves on the exhaust flow can be seen in Figure 4 in the form of Mach number contours where P_{atm}/P_{des} equals 0.41.

Figure 2: Mach number Contours in design condition

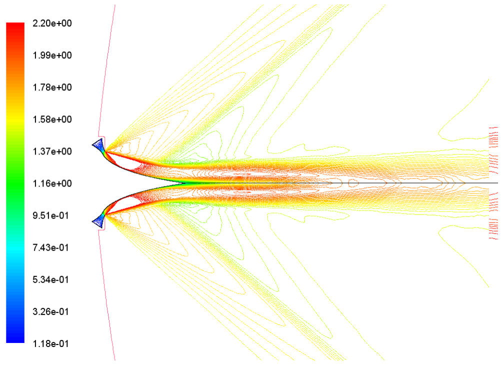

Figure 3: Mach number Contours in over-expansion condition

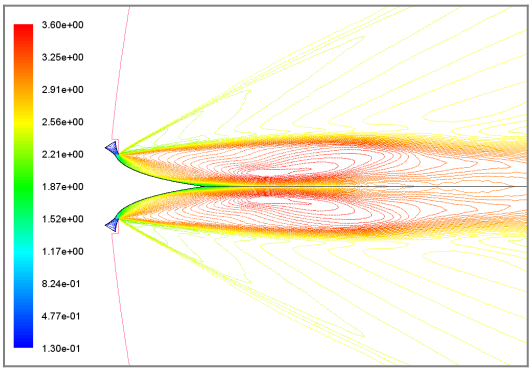

Figure 4: Mach number Contours in under-expansion condition

References

[1] Verma, S.B., Performance Characteristics of Annular Conical Aerospike Nozzle with Free stream Effect, *Journal of Propulsion and Power*, Vol. 26, NO.3, 2009.

[2] Wang, T., Effect of Fence on Linear Aerospike Plume-Induced Base-Heating Physics, *Journal of Thermo physics and Heat Transfer*, Vol.14, No.4, 457-463, 2000.

[3] Koutsavdis, E.k., A Numerical Investigation of the Flow Characteristics of Plug Nozzle Using Fluent, *Proceedings of the 40th AIAA meeting and exhibition*, Reno, NV, 2002.

[4] S. Noori and A. Shahrokhi The Influence of different Turbulence Models on the Flowfield Characteristics of an Aerospike Nozzle, *International Conference on Mechanical and Aerospace Engineering*, *Kuala Lumpur, Malaysia*, 26- 28 Nov. 2010.

[5] Besnard, E., H. H. Chen, T. Mueller and J. Garvey, Design, Manufacturing and Test of a Plug Nozzle Rocket Engine, *Proceedings of the 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, 2002.

.