Numerical analysis of slotted aerospike for drag reduction

S.Manigandan¹, K N Sruthisree¹, KaushaliAich¹, P.Gunasekar¹, S.Nithya¹, J.Devipriya¹, S.Venkatesh

¹Department of Aeronautical, Sathyabama Institute of Science and Technology, Chennai-600119, India.

E-mail: manisek87@gmail.com

Abstract. One of the important criteria for designing high speed vehicles is drag reduction and aerodynamic heating. There are plenty of methods available however finding an economical and simple method for drag and heat flux reduction is very challenging. In this paper, the forward facing aerospike for blunt-nosed bodies is introduced at supersonic and hypersonic Mach numbers and tested for drag reduction. Initially, the flow fields are studied around the blunt cone with and without aerospike. In addition, the different shapes and L/D ratios flying at supersonic Mach 2 are computed numerically. The computational simulations is carried out to solve the three-dimensional steady Reynolds-averaged Navier-Stokes equation along with the k-ω turbulence model in computation solver. After comparing the flow properties around the various aerospike models, we have found that the flat aerodisk experiences a drastic pressure reduction on the blunt nose cone but it has relatively less heat flux reduction. Consequently, a slotted aerodisk modification design has been analysed which uses convection flow to reduce heat flux.

1. Introduction

Generally, blunt shape geometry is used in high-speed missile, rockets, capsules, etc. in order to accommodate larger payload and to withstand severe thermal expansion exerted by the forebody which leads to the formation shock wave. The aerodynamic drag and aerodynamic heating lead to the reduced performance and erosion of the surface. The methods like breathing nose, an energy deposition technique, spike-tipped nose, adopted to suppress the aerodynamic heating problems. Among all the methods an application of forward facing aerospike for blunt-nosed bodies at supersonic and hypersonic Mach numbers is most economical [1]. Aerospike leads to transformation of strong bow or detached shock wave into weak shock waves followed by recirculation or dead air region. This dead region depends on which will depend on the shape of the tip of the spike. The forebody of missiles usually faces lesser pressure due to the presence of recirculation of air and hence the reduction of drag is clear. The flow field region between the tip of the aerospike and the main body depend on freestream flow conditions, the shape of the...
body and the geometry of the spike. The study of different length of the sharp spike at hypersonic Mach number 6.8 over a configuration shows the traces of reduction in drag and heat flux [2]. Further, several notable works made on the hemispherical body with different spikes at hypersonic speeds and saw low drag [2, 3]. Computational results are done using the Baldwin-Lomax turbulence model, where k-ω turbulence model was used [6-8]. The flow field around the body with aerospike will be different for hypersonic Mach number over speed owing to the shock strength and the separation zone between the spike. Pieces of literature reported that the flow field around blunt bodies with different kind of aerospike in supersonic speed. The domain shape is elliptical [9-16]. Manigandan et al. recently proved with his notable work stating that elliptical has high advantage than other profile. Major studies were carried out on sharp or blunt aerospike. Studies with spherical tip aerodisk at the hypersonic speed which shows a reduction in drag in comparison to sharp spike is very limited. Implementing k-ω turbulence model at Mach 2 with sharp and hemispherical blunt head spike reduces the drag up to 68%. The combinational jet with aerospike concept at supersonic speed was proposed to be more efficient in drag and heat reduction. Rahul et al. studied the simulation over the ballistic missile at Mach 6. They found the performance of the conical aerospike for the different length to diameter ratio to examine the effect of aerodynamic forces. They found the angle of attack is proportional to the drag on the body and pitching moment. Schnabel et al. studied the pressure distribution on aerospike nozzle on the rotating engines. The nozzle had been designed to examine the effect of pressure distribution. Dumitrescu et al. studied the plug nozzle to estimate the nozzle performance across the sea level. They have chosen three types of the plug of truncated length 40%, 50% and 60%. From the results they concluded the altitude and temperature are very important parameters for optimal performances. From the above studies, we have come to know that drag depends on the shape and size of the spike. Moreover, the material property is neglected for the simulation reference purpose we have used composite material as the spike material [17-24].

Figure 1. Aerospike model

2. Simulation Methodology

The simulation is carried on the blunt hemispherical body of base diameter (D) of 14mm and length 1.5D which is shown in figure 1. Three different types of spikes were used for the analysis whose basic stem diameter was 2mm. The Semi-cone angle of 10 - 12 degrees was given to sharp aerospike and the flat head aerodisk (Aerospike) having a diameter of 0.3D with a flair angle of 120 degrees. Three different cases of L/D ratio (0.75, 1, 1.5) of individual model were studied to uncover the optimum solution.
Flathead aerodisk configuration is believed to be the optimum solution. Hence a slot was made around the stem throughout the disc having a diameter of 0.2 mm as shown in figure 1. In order to save time, 2D computations have been used. The platform Star CCM+ used to simulate the problem which uses a finite volume approach to solve compressible Reynolds Averaged Naiver Stokes (RANS) equations. The assumptions made for the simulation is a steady-state solution, axisymmetric computations explicit coupled using "k-ω" turbulence model. The k-ω turbulence model has been arrived after carrying out the sensitivity tests, convergence and comparison of the experimental and numerical results. A flow enclosure of hemi circular was made around the designed model. Unstructured grids were made with uniformly distributed quadrilateral cells as shown in figure 2. The mesh consists of 100 divisions of the cell and 8 layers in the surface of the body around the boundary of the body in order to capture flow pattern with high accuracy around the body. Meanwhile, the boundary conditions are named as inlet, wall and outlet. The pressure far field boundary values are used during simulation which is computed from literature. Further, the wall conditions are assumed to be no slip walls with near wall treatment for turbulent flows.

3. Results and Discussion

Different types of aerospike are studied for Mach number 2 is shown in figure 3 to figure 9.

3.1 Effect of spike on flow field

The flow field characteristics around a model with and without aerospike or slotted aerodisk are studied. Due to the presence of an adverse pressure gradient, the flow separation takes place. In addition, a large region of recirculation fluid zone is formed along the spike length. And consequently, the flow can be supposed to be a superimposition of two flows, which are called as external free stream flow and a recirculation flow. The regions like recirculation zone always depend on the L/D ratio of the geometry and aerospike of the aerospike models. In our study, we have taken 3 geometries for aerospike which are a sharp spike, flat disk aerospike and hemispherical head aerospike. Different kinds of shocks are formed on various models. For example, oblique shock wave formation on a sharp spike whereas bow shocks on flat head disk [25-27]. The extent of recirculation is also dependent on the nature of shock waves. The modification of slot in aerodisk increases the number of recirculation fluid regions. The free stream flow passes from side
to side of the slotted hole which then forms a series of wake and vortices right behind the rear part of the slotted aerodisk followed by a primary recirculation region.

As the length of the slotted aerospike increases, the degree of the wake is increased this may result in structural fatigue. Figure 3 to figure 7 shows the variation of the flow field. Among the several slots, figure 7 reports better mixing compared to other spike.

**Figure 3.** Flow visualization of blunt nose

**Figure 4.** Flow visualization of aerodisk
Figure 5. Flow visualization of hemisphere aerospike

Figure 6. Flow visualization of sharp aerospike
3.2 Effect of spike on surface pressure

The tip of aero spike forms the shock wave and leads to the formation of recirculation region around the roots of aero spike. Figure 8 and figure 9 shows the pressure distribution. This recirculation region acts as a streamlined body which reduces drag. Pressure drop between the aero spike and the blunt body is directly proportional to the ratio of L/D ratio of the spike.
In our study, we have taken 3 geometries for aerospike which are a sharp spike, flat disk aerospike and hemispherical head aerospike. In comparison with blunt nose cone, the peak pressure drop for the sharp spike, hemispherical spike and flat disk aerospike are 12.74%, 28.79% and 82.72% respectively.

The largest pressure drop is found using flat disk aerospike which is 82.72% as shown above. Hence, we have chosen flat disk aerospike as the best design which is further modified into slotted flat disk aerospike due to an aerodynamic heating problem. The peak pressure drop on slotted aero disk is 63.81% on comparing with blunt nose design. Thus, due to the introduction of a slot over flat disk aerospike, there is 18.91% increase in pressure in comparison with flat disk spike. This increase in pressure of slotted disk aerospike is due to the formation of series of wake or vortices right behind the rear part of the slotted aerodisk followed by a primary recirculation region [24-25].

3.3 Effect of spike on surface heat flux

There is a minor change in heat flux around the root of aerospikes. In our study, we have taken 3 geometries for aerospike which are sharp spike, flat disk aerospike and hemispherical head aerospike. In comparison with blunt nose cone, the heat flux drop for sharp spike and hemispherical spike flat disk aerospike are 0.12% and 0.18% respectively. When the bow shockwave formed in front aerodisk which results in a higher value of stagnation point heat fluxes which is higher than those for hemisphere-cylinder without aerodisk. There is an increase of 2.91% in heat flux for flat disk aerospike on comparing with a blunt nose. We have chosen flat disk aerospike in spite of an increase in heat flux by 2.91%. This design is taken as an optimum design by considering the fact that there is a maximum pressure drop in the flat disk aerospike when compared to other models. Hence, the slot is introduced on the flat disk aerospike.

The free stream flow passes through the slotted hole which then forms a series of wake or vortices right behind the rear part of the slotted aerodisk followed by a primary recirculation region. This free stream flow acts as a convection medium that passes through the slot. Consequently, there is a decrease of heat flux on slotted aerospike in comparison with blunt nose cone due to convection taking place. 5.12% heat flux is decreased with the application of slotted aerospike. The overall decrease in heat flux of slotted aerospike in comparison with flat disk aerospike is 8.03%.
Conclusion

The pressure and reattachment heat flux of a blunt body with different models of aerospikes is tested for results under k-ω turbulent conditions. The operating Mach number is of Mach 2. The numerical simulation is conducted on aerospikes of three different L/D ratios of 0.75, 1.00 and 1.5 suggested that there is an increase in pressure drop with increase in L/D ratio of aerospikes. In this study, we have found that for flat disk aerospike which has a maximum pressure drop of 80.72%, the local heating at the reattachment is always higher than the peak heating of the base model about 2.91%. Hence, we introduced a slot at the tip of the flat disk aerospike concentrically. This resulted in 62.81% of pressure drop and 6.12% of heat flux reduction due to convection taking place through the slot from free stream flow to the recirculation region. Conclusively, our slotted flat disk aerospike model is the best model design in which the reduction of pressure and peak heat flux has been compromised.

References

1. Ma Y, Liu X, Ou P. Numerical investigation of hypersonic unsteady flow around a spiked blunt-body. Procedia Engineering. 2015 Jan 1; 126:163-8.

2. Huang W, Liu J, Xia ZX. Drag reduction mechanism induced by a combinational opposing jet and spike concept in supersonic flows. Acta Astronautica. 2015 Oct 1; 115:24-31.

3. Lohner R, Baum JD. On the Drag Efficiency of Counterjets in Low Supersonic Flow. In53rd AIAA Aerospace Sciences Meeting 2015 (p. 0069).

4. Schnabel MC, Brophy CM. Pressure distribution and performance impacts of aerospike nozzles on rotating detonation engines. In2018 AIAA Aerospace Sciences Meeting 2018 (p. 1626).

5. Dumitrescu O, Gherman B, Dragan V. Study of full and truncated aerospike nozzles on performances at different working conditions. InIOP Conference Series: Materials Science and Engineering 2018 Aug (Vol. 400, No. 4, p. 042017). IOP Publishing.

6. Manigandan S, Gunasekar P, Devipriya J, Saravanan WS. Reduction of greenhouse gases by the effect of window position and its size in isolated building. Journal of chemical and pharmaceuticals sciences. 2016.

7. Manigandan S, Gunasekar P, Sruthisree N, Aich K, Sathya K, Selvan A, Nithya S. Visualization of vortex flow field around a flat plate with noncircular hole. InIOP Conference Series: Materials Science and Engineering 2018 Feb (Vol. 310, No. 1, p. 012135). IOP Publishing.

8. Manigandan S, Kumar KG, Gunasekar P, Nithya S. Experiment Evaluation of Skin Friction Drag by Surface Tailoring. InIOP Conference Series: Materials Science and Engineering 2017 Aug (Vol. 225, No. 1, p. 012062). IOP Publishing.

9. Manigandan S, Vijayaraja K. Acoustic and mixing characteristic of CD nozzle with inverted triangular tabs. International Journal of Ambient Energy. 2017 Jul 20:1-5.
10. Manigandan S, Vijayaraja K. Flow field and acoustic characteristics of elliptical throat CD nozzle. International Journal of Ambient Energy. 2017 Aug 17:1-6.

11. Manigandan S, Gunasekar P, Devipriya J, Nithya S. Determination of heat flux on dual bell nozzle by Monte carlo method. Journal of chemical and pharmaceuticals sciences. 2016.

12. Gunasekar P, Manigandan S, Devipriya J, Saravanan WS. Investigation of Dual Mode RJ Nozzle by Discrete transfer method. Journal of Chemical and Pharmaceutical Sciences, ISSN. 2016;974:2115.

13. Manigandan S, Vijayaraja K, Gunasekar P, Nithya S, Devipriya J, Ilangovan N. Mixing characteristics of elliptical throat sonic jets from orifice and nozzle. International Journal of Ambient Energy. 2017 Nov 16:1-3.

14. Manigandan S, Vijayaraja K. Energy decay characteristics of a non-circular supersonic jet. International Journal of Ambient Energy. 2018 Feb 22:1-4.

15. Manigandan S, Vijayaraja K, Revanth GD, Anudeep AV. Mixing and Entrainment Characteristics of Jet Control with Crosswire. InAdvances in Machine Learning and Data Science 2018 (pp. 121-128). Springer, Singapore.

16. Manigandan S, Vijayaraja K. Flow Mixing and Screech Characteristics of Elliptical Supersonic Jet. International Journal of Turbo & Jet-Engines.

17. Pawar RS, Gilke NR, Warade VP. Numerical Simulation Over Conical Aerospike at Mach 6. InProceedings of International Conference on Intelligent Manufacturing and Automation 2019 (pp. 15-26). Springer, Singapore.

18. Gunasekar P, Manigandan S, Anderson D, Devipriya J. Evaluation of Fe-Epoxy metal nanocomposite in glass fiber and Kevlar. International Journal of Ambient Energy. 2017:1-5.

19. Gunasekar P, Manigandan S. Computational analysis of frp composite under different temperature gradient. InIOP Conference Series: Materials Science and Engineering 2017 May (Vol. 197, No. 1, p. 012054). IOP Publishing.

20. Manigandan S, Gunasekar P, Nithya S, Revanth GD, Anudeep AV. Experimental analysis of graphene nanocomposite on Kevlar. InIOP Conference Series: Materials Science and Engineering 2017 Aug (Vol. 225, No. 1, p. 012061). IOP Publishing.

21. Deng F, Xie F, Qin N, Huang W, Wang L, Chu H. Drag reduction investigation for hypersonic lifting-body vehicles with aerospike and long penetration mode counterflowing jet. Aerospace Science and Technology. 2018 May 1;76:361-73.

22. Eghlima Z, Mansour K. Drag reduction for the combination of spike and counterflow jet on blunt body at high Mach number flow. Acta Astronautica. 2017 Apr 1;133:103-10.

23. Karimi MS, Oboodi MJ. Investigation and recent developments in aerodynamic heating and drag reduction for hypersonic flows. Heat and Mass Transfer. 2018 Jul:1-23
24. Ou M, Yan L, Huang W, Li SB, Li LQ. Detailed parametric investigations on drag and heat flux reduction induced by a combinational spike and opposing jet concept in hypersonic flows. International Journal of Heat and Mass Transfer. 2018 Nov 30;126:10-31.

25. Bibi A, Maqsood A, Sherbaz S, Dala L. Drag reduction of supersonic blunt bodies using opposing jet and nozzle geometric variations. Aerospace Science and Technology. 2017 Oct 1;69:244-56.

26. Sun XW, Guo ZY, Huang W, Li SB, Yan L. Drag and heat reduction mechanism induced by a combinational novel cavity and counterflowing jet concept in hypersonic flows. Acta Astronautica. 2016 Sep 1;126:109-19.