Design and Analysis Aerospike Blunt Body at Hypersonic Speed

Dr. Muzeer.S[1], Umerpasha Jamilpasha Patil[2], Rakesh Vejendla[3]
[1]Assistant Professor, Department of Aerospace, Sanjay Ghodawat University, Kolhapur, Maharashtra, India.
[2,3]Department of Aerospace, Sanjay Ghodawat University, Kolhapur, Maharashtra, India.
Emails: saiyed.muzeer.30@gmail.com[1], vraki8686@gmail.com[2], patelumer77@gmail.com[3]

Abstract

Hypersonic vehicles have the drag and heating problem when they operate at high speeds, to avoid or minimize this problem, aero spikes are utilized. This investigation finds a solution for reducing the drag and heating. Using ANSYS fluent designed an aero spike, which has flow around a blunt body at hypersonic Mach number 6 with angle of attack zero. It is analyses for L/D ratio of 1.5 and 2. The front of the body is replaced by an aero spike which has weak oblique shocks and well-built detached shock. This develops a circular region between shock and blunt body. This works as a streamlined profile to reduce heat as well as drag.
Keywords: Aero spike, Blunt body, Hypersonic Mach number, ANSYS, Bow shocks.

1. Introduction

A rapidly developing area of technology called hypersonic flight holds the potential to completely transform both air and space travel. The design and development of vehicles that can sustain these circumstances, however, is significantly complicated by the high speeds and temperatures associated with hypersonic flight. Aero spike blunts body designs are available strategy for reducing the consequences of hypersonic flight. The aero spike blunt body is a type of hypersonic vehicle design that has shown potential for reducing the heating and drag associated with high-speed flight. The design has blunt body and a spike at the rear of the body [1]. This extension is designed to generate a shock which again directs the hot gas flow out from the aircraft or hypersonic vehicle, which reduces the heat on the surface [2]. "Heat transfer is a critical aspect of the aerospike blunt body design because it affects the thermal protection system required to protect the vehicle from the extreme temperatures generated during flight". Heat transfer occurs through several mechanisms, including conduction, convection, and radiation. In hypersonic flight, the dominant the at transfer mechanism is convective heating, which occurs when the hot gas molecules collide with the surface of the vehicle, transferring heat to the surface. The design and analysis of heat transfer over an aero spike blunt body at hypersonic speeds is a complex and challenging task that requires the use of advanced simulation techniques [3]. Computational fluid dynamics (CFD) simulations are commonly used to model the flow of hot gases around the vehicle and predict the heat transfer rates [4]. These simulations take into account the complex interactions between the vehicle and the surrounding air, including shock waves, boundary layer effects, and turbulence [5]. To design an effective thermal protection system for an aero spike blunt body, it is necessary to understand the heat transfer mechanisms involved and how they affect the surface temperature of the vehicle. The design must also take into account the structural requirements of the vehicle, including weight, strength, and durability. Overall, the design and analysis of heat transfer over an aero spike blunt body at hypersonic speeds is a critical area of research that has the potential to enable practical hypersonic flight [6][7]. By developing effective thermal protection systems, it may be possible to overcome the technical challenges associated with hypersonic flight and unlock new frontiers in air and space travel [8]. An aero spike blunt body is a type of hypersonic vehicle design that has been proposed as a means to reduce the heating and drag associated

with hypersonic flight. The aero spike design consists of a blunt body with a spike-shaped extension at the rear of the body. This extension is designed to form a shock wave that redirects the flow of hot gases away from the vehicle, reducing the heating on the surface. Heat transfer is a critical aspect of aero spike blunt body design because it affects the thermal protection system required to protect the vehicle from the extreme temperatures generated during flight. Heat transfer occurs through several mechanisms, including conduction, convection, and radiation. In hypersonic flight, the dominant heat transfer mechanism is convective heating, which occurs when the hot gas molecules collide with the surface of the vehicle, transferring heat to the surface. The design of the aero spike blunt body must take into account the heat transfer characteristics of the vehicle to ensure that the thermal protection system can withstand the extreme temperatures generated during flight. Computational fluid dynamics (CFD) simulations are commonly used to model the flow of hot gases around the vehicle and predict the heat transfer rates [9]. In summary, heat transfer is a critical factor in the design of an aero spike blunt body for hypersonic flight. Understanding the mechanisms of heat transfer and using CFD simulations to predict heat transfer rates is essential to develop effective thermal protection systems for hypersonic vehicles [10].

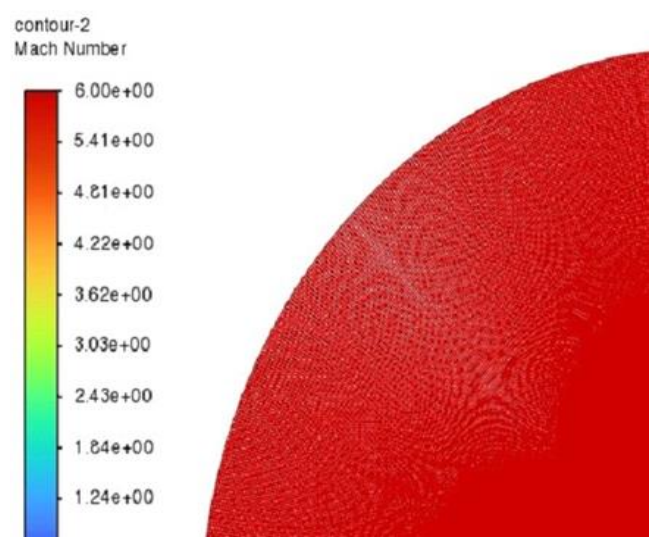## 2. Methodology

2.1 Meshing and Boundary Condition
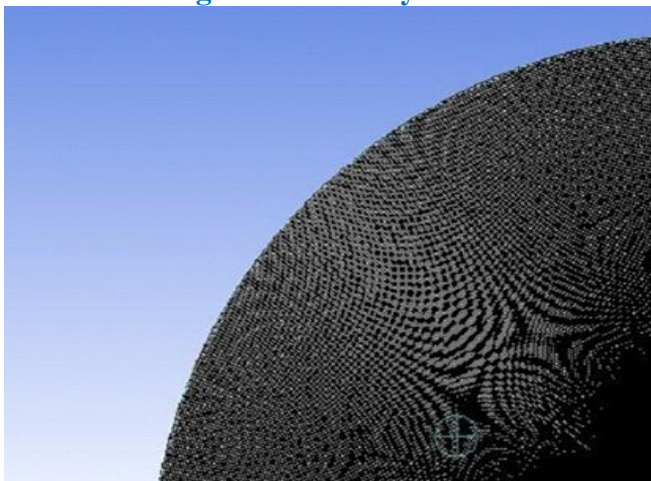
Figure 1 Blunt Body Mesh Without a Spike
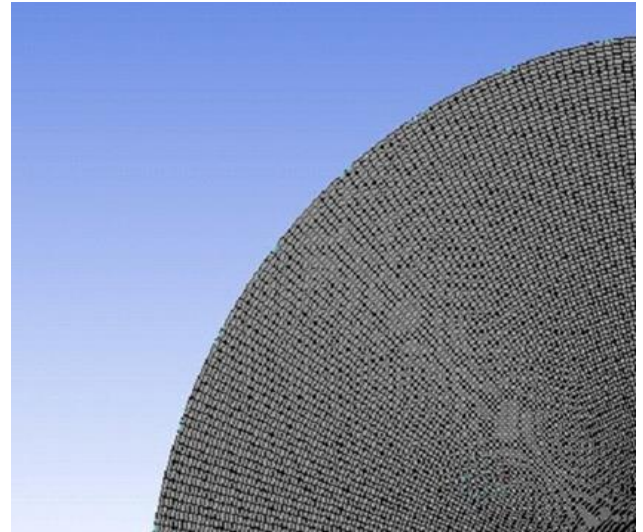
Figure 2 Blunt Body Mesh with Spike

For inner and outer free stream condition were given. At Mach 6, the free stream conditions are utilized for inner layer boundaries [11]. The temperature is 300k, stagnation pressure is 8.3 bars and stagnation temperature is 450k. For 60mm length, Reynolds number is 5*105. For the second it ions the mesh is created. The above figures 1 & 2 shows the L/D 1.5 with no spike [12].
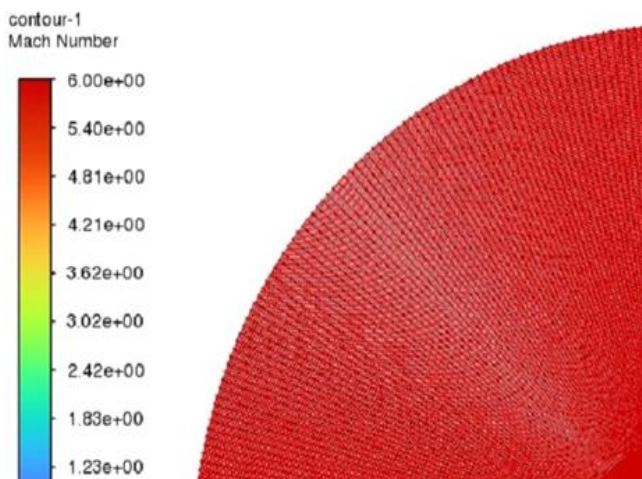
2.2 CFD Analysis

contour-2
Mach Number

6.00e+00
5.41e+00
4.81e+00
4.22e+00
3.62e+00
3.03e+00
2.43e+00
1.84e+00
1.24e+00

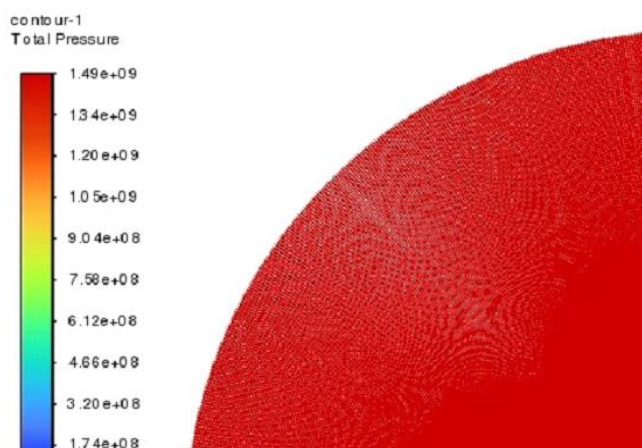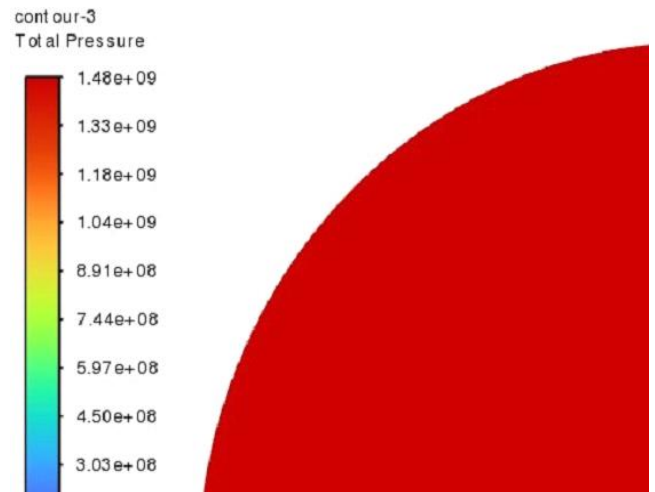**Figure 3** Blunt Body Without Spike for Mach Number

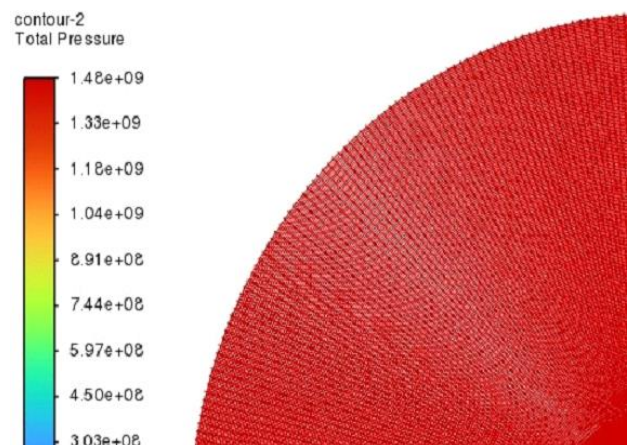**Figure 4** Blunt Body with Spike at L/D 1.1.5

Figure 5 Blunt Body with Spike at L/D 2

Figure 6 Total Pressure Contour for Blunt Body Without Spike

Figure 7 Total Pressure Contour for Blunt Body with Spike at L/D Ratio 1.5

Figure 8 Total Pressure Contour for Blunt Body with Spike at L/D Ratio 2

The Mach Number Contour and Total Pressure contour for all the three cases– Blunt Body without spike, Blunt Body with spike at L/D ratio 1.5 and Blunt Body with spike at L/D ratio 2are shown as the above images [13] [14]. These results are obtained after giving the boundary condition for the model in the ANSYS Software [15].

3. Results and Discussion

Highly strengthened bow shocks are formed, which are very near the spiked blunt body. Shocks deviate on spike and create a recirculation zone which was close to the body surface. "Due to this heat and pressure drops in zone ahead of the blunt body. The aero spike introduces bow shocks with low capable

oblique shocks; this can decrease the pressure on surface of a body. The above images show the without aero spike with different Mach number flow through a blunt body". Aero spike of L/D 1.5 has better variations for different Mach numbers and forms a bow shock in front of the blunt body. L/D ratio 2 has also having good variation with aero spike blunt body with various Mach numbers. The surface of the blunt body was cooled by the temperatures in the region of the recirculation. This recirculation depends upon the length and diameter of the aero spike. These all conditions lead the blunt body; it reduces the drag as well as pressure in front of the aero spike body. From the above figures 3 & 4 L/D1.5 shows better pressure reduction of the aero spike blunt body as compared to L/D ratio 2. Above all the figures 5, 6, 7, 8 show the pressure for various cases. This figure shows total pressure at different Mach numbers. The above images show the without aero spike with different Mach number flow through a blunt body". Aero spike of L/D 1.5 has better variations for different Mach numbers and forms a bow shock in front of the blunt body. L/D ratio 2 has also having good variation with aero spike blunt body with various Mach numbers. The surface of the blunt body was cooled by the temperatures in the region of the recirculation. This recirculation depends upon the length and diameter of the aero spike [16]. These all conditions lead the blunt body; it reduces the drag as well as pressure in front of the aero spike body. From the above figures L/D1.5 shows better pressure reduction of the aero spike blunt body as compared to L/D ratio 2. Above all the figures show the pressure for various cases. This figure shows total pressure at different Mach numbers.

Conclusion

This investigation utilizes the ANSYS fluent for hypersonic Mach number 6, which is having aero spike with different L/D ratio, that is 1.5, 2 respectively at 00angle of attack. The pressure difference sand Mach number contour are investigated. Due to the spread of recirculation, pressure drops in stagnation zone. This research analyses two studies that is with spike and without spike. Without spike has very poor pressure values and with spike has observed as high-pressure values at L/D ratio of 1.5, and 2. The total pressure drops for L/D 1.5 ahead of blunt body; this indicates the reduction in the drag. This study is help full for further development of drag reduction and experimenting with different drag reduction techniques with different L/D ratios.

Bibliography

[1]. S. A. Shahab Esmailzadeh Vali, "Hypersonic drag and heat reduction mechanism of a new hybride method of spike," September 2022.

[2]. S. A. Shahab Esmailzadeh Vali, "Hypersonic drag and heat reduction mechanism of a new hybrid method of spike, multi-row discs and opposing jets aerodynamic".

[3]. S. F. J. G. Z. N. Y. X. Ziyu Wang, "Investigation of influence factors to the reduction of hypersonic blunt-body drag and aeroheating based on a spike - aerodisk - Channel concept," Acta Astronautica., vol. 211, pp. 716-733, 2023.

[4]. T. C. L. Ami Vyas, "Porosity development and reactivity changes of coal–biomass blends during co-pyrolysis at various temperatures. Ami Vyas, Thiago Chellappa , Jillian L. Goldfarb , Journal of Analytical and Applied Pyrolysis,," vol. 124, pp. 79-88, March 2017.

[5]. S. F. J. G, Z. W, Y. x. Zijian Ni, "Three-dimensional numerical study on reducing hypersonic blunt body drag and aero heating with spike-aero disk-bleed air channel," Acta Astronautic., vol. 215, pp. 674-688, 2024.

[6]. S. F. J. G. Z. N. Y. X. Ziyu Wang, "Investigation of influence factors to the reduction of hypersonic blunt-body drag and aero heating based on a spike - aerodisk - Channel concept," Acta Astronautic, vol. 211, pp. 716-733, 2023.

[7]. H. A. G. B. T. L. T. T. Uma singh, "Cyan metabolites: molecules with immense antiviral potential".

[8]. H. A. G. B. T. L. T. T. Uma Singh, " Cyan

metabolites: molecules with immense antiviral potential".

[9]. T. C. J. L. G. Ami Vyas, "Porosity development and reactivity changes of coal–biomass blends during co-pyrolysis at various temperatures.," Journal of Analytical and Applied Pyrolysis, vol. 124, pp. 79-88, march 2017.

[10]. T. C. J. L. G. Ami Vyas, "Porosity development and reactivity changes of coal–biomass blends during co-pyrolysis at various temperatures," Journal of Analytical and Applied Pyrolysis, vol. 124, pp. 79-88, 2017.

[11]. S. E. J. A. S. Jiss J. Sebastian, "Computational Study of Hypersonic Flow Past Spiked Blunt Body Using RANS and DSMC Method7 Computational Study of Hypersonic Flow Past Spiked Blunt Body Using RANS and DSMC Method.".

[12]. S. E. J. A. S. Jiss J. Sebastian, "Computational Study of Hypersonic Flow Past Spiked Blunt Body Using RANS and DSMC Method.".

[13]. R. Mehta, "Flow Field Computations Over Conical, Disc and Flat Spiked Body at Mach 6.," 2012.

[14]. S. I. I. M. P. a. L. N. M. Edwin, "Computational Study of Aero-Spike On Blunt Body," Materials Science and Engineering, 2022.

[15]. V. Tekure, P. S. Pophali and K. Venkatasubbaiah, "Numerical investigation of aerospike semi-cone angle and a small bump on the spike stem in reducing the aerodynamic drag and heating of spiked blunt-body: New correlations for drag and surface temperature," 2021.

[16]. S. M. T. P. L. Mohamad Kazemi Esfeh, "Evaluation of aerospike for drag reduction on a blunt nose using experimental and numerical modeling.," Acta Astronautica., vol. 160, pp. 656-671, 2019.