SURVEY ON BALISTIC MISSILE FIN CONFIGURATION DURING SUPERSONIC FLIGHT

Saddam Hussain[1], Anil Verma[2], Priyanka Jhawar[3], Dr. G.R.Selokar[4]

[1]*Research Scholar, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA*
[2]*Assistant Professor, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA*
[3]*Assistant Professor, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA*
[4]*Professor, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA*

ABSTRACT

The focus of this paper is to study of heating and thermo-Structural analysis of fin of a missile structures at supersonic flight. The heat due to friction at high Mach number between the flow and body, i. e. viscous heating must be taken into account due to the velocity field is coupled with the temperature field. The flow field around the fins of the missile and especially the temperature distribution on its surface, as well as thermal-structural analyses are numerically modeled in ANSYS FLUENT.,

Keyword: - *Missile Fins, Supersonic Flight, Thermodynamic Heating, and Thermal-Surface Interaction*

---

1. INTRODUCTION

1.1 Aerodynamic Heating

Aerodynamic heating is defined as the heating when an Aerospace vehicle or missiles flying at very high speed such as supersonic or hypersonic speed regimes due to shock waves compression as well as frictions within the boundary layer around the missile body or aerospace vehicle. Aerodynamic heating produced due to a cause of temperature rise because of the roughness of the surface and friction between surface air interactions at a high speed. However the aerodynamic heating can be reduced due to installation of thermal protection system in short TPS. Reduction in aerodynamic heating is based on optimal flight path or optimal shapes. And hence due to optimization of shape of a vehicle an amount of aerodynamic heating can be minimized. Since as we know that missile are moving at very high speeds as a result aerodynamic heating produces a major problem which is to be considered for the design of the missiles. However this problem is much larger near to the stagnation regions, such as near the missile fin leading edge [1, 2, 3]. Also at the supersonic flight with higher Mach numbers there is one of the major issues arises over the fin that is the fins can become substantial subjected to damage due to excessive heating of temperatures and structural loading. Therefore, the investigation of aerodynamic heating which is naturally coupled with aerodynamic forces which is acting on the structure of the fins, which represents the one of the major challenges in the design and analysis of supersonic & hypersonic vehicles, such as rocket, missile, and other aerospace vehicle.

Aerodynamic-thermal-structural analysis methods has an important role in the de-signing and analysis of high speed aerospace flight vehicles and modeling and designing of computational aero-thermo-elasticity phenomenon, therefore it represents the necessary tool in aerospace design, analysis as well as optimization. However the numerical prediction of induced deformations and stresses developed due to thermal and heat radiation [7] has been accomplished for better designing and analysis of aerospace vehicles [8].

In fact, the heating problem due to high speed requires aero-thermo-elastic analysis that is based on multi-disciplinary fluid-structures interaction between the surface and air heating as well as due to the aerodynamic & inertial forces acting on the body which results in the behavior and responses during the suction. In order to design the best possible design criteria aerodynamic, thermal as well as structural modeling and analyses must have to be met.

In this thesis multidisciplinary study for numerical aerodynamic thermo-structural analyses were carried out on short and long range aerospace vehicles well as ballistic missile fin model. That was developed by scientific internal experimental as well as CFD & computational-structural-mechanics (CSM) for the purposes of testing and calibration. The best design process of the missile fin mode including primarily static-structural experimental analyses to achieve the maximal strength durability and safeness. In this thesis results of the experiments will be used for validation & verification of the CFD & CSM analysis of our missile fin model. Validation & verification procedures for the numerical aero-thermodynamic and structural analysis based on the static structural and thermal experiments were the part of necessary designed trend line routine, so that the numerical analysis of aero-thermo-structural analyses would be carried out with an acceptable accuracy.

The main aim of this thesis is to represent the numerical aero-thermo-structural analyses of fin of ballistic missile during supersonic and hypersonic flight regimes by using of few designing and analysis softwares. In order to investigate the effects of thermal heat transfer on the fin structures of the missile, therefore for safety and reliability purposes a numerical approaches are required to improve and quicken the overall design and analysis processes.

It should be noted in our thesis that our study addresses a relevant aspect of missile aero-thermo-elasticity, of the fin structures. Because, this kind of very complex and robust analysis, only few test were conducted on missile fin structures that can be seen from the literature reviews. However they have used either custom made codes which generate aerodynamic and thermal database, based on the curve fits of experimental data, 2-D, and 3-D simplifications or semi-empirical methods such as missile dotcom, or the data reduced from theoretical methods, those are based on in viscid aerodynamic. Even though if they used a commercial codes and softwares they have been used only developed data transfer interfaces.

When a missile travels into the air, the surface of the missile gets heated due to the friction with the air. Therefore at microscopic level the air molecules will be retarded by the surface (because of roughness of the surface) and hence heat is generated due to temperature differences thought the small particles that hits the missile fin unrealistically, therefore the kinetic energy of the fluid particles are converted to heat. This aerodynamic heating is not in any way the realistic model of the actual phenomena for 3 reasons:

1. The air flows around the missile, and the temperature of the air-flow depends upon local conditions. The surface of the missile does not reach absolute temperature rather than a temperature distribution.
2. The aerodynamic heating does not necessarily originate from friction, but from heat transfer from heated air-flow to the missile surfaces, and this heat transfer from the air-flow to the surface of the missile is not perfectly smooth.
3. The surface heating is the dynamic process due to motion of the flow, where the capacity of heated surface raises a lag in the temperature on the surfaces with respect to the flow.

2. LITERATURE SURVEY

H. Julian et al. [4] Explained about the missiles which is developed in earlier periods featured pointed nose on a slender body, which has also had extreme temperature at the nose cone due to attached.

D. Sahoo et al. in 2016 [5] studied the flow field characteristics of various aerospike with different shapes and length experimentally and computationally, at Mach number of 2.0 and at zero angle of attack. The different geometric shape of the aero spike investigated were sharp tip aero spike, flat tip aerospike and spherically blunted tip aero spike. It was found that the changes in the spike length and shape result in considerable change in the drag. The maximum drag reduction observed was 46.80% .For an l/D =1.5, a drag reduction of 45.40% is observed for sharp aerospike. They also observed unsteadiness in the flow, and found that change in the aerospike shape from sharp aerospike to blunt aerospike for a particular length of the aerospike reduces unsteadiness in the flowfield.

Humieres and Stollery [6] did experimental investigation on the 1:100 scaled down model of Apollo re-entry vehicle. The diameter of the projected body was kept to be 39 mm and the spherical cap radius was equal to 36 mm while the diameter of the cylindrical aerospike was 2 mm which was fitted with a conical tip. The l/D ratio of aerospikes investigated were 0, 0.125, 0.25, 0.5, 0.75, 1, 1.25, 1.5, 1.75 and 2.125 for a freestream Mach number of 8.2 and a Reynolds number of 9.01 x 106. The experimental study revealed that no unsteadiness in the flow was detected for all the aerospike lengths investigated and a reduction in drag was observed for the shorter aerospikes. It was further suggested that drag produced by the spiked body can be approximated by the pressure drag generated by a solid cone, with some correction.

Ahmed and Qin in 2010 [7] did an extensive research in the area of drag reduction mechanism for hemispherical body flying at hypersonic speed. He found that a blunt body, when flying at hypersonic speed experiences excessive aerodynamic heating and large increase in pressure drag which could possibly be reduced by deploying an aerodisks at the stagnation point of the main body. The l/D ratios of the aerospikes investigated were varied up to 2.5 and the diameter of the aerodisk selected was 0.4 times the body diameter. Based on the numerical investigation, they explained that the dividing streamline of the shear layer was assumed to be the outline of the effective body which replaces the original spiked hemispherical body that effective body shape and the flow stability depends on the energy level of the dividing stream. They concluded that the aerodisks is more effective in reducing aerodynamic drag than aerospikes when the separation point is shifted on the aerospike axis, reducing the downstream pressure by the formation of expansion fans at the shoulder of the aerodisk. It was also observed that the drag reduction mainly depends upon the size of the aerodisk and the length of the aerospike. An optimum aerodisk size for a specific aerospike length produced minimum drag which was found to be inversely proportional to the aerospike length.
Mehta R C [8] in 2000 for a freestream Mach number of 6.8. The geometry studied was a hemispherical cone with a pointed aerospike ahead of it and the flow was assumed to be laminar. The studies were conducted on conical aerospike and hemispherical aerospike of various l/D ratios of 0.5, 1.0 and 2.0 suggests that a conical shock wave generated at the tip of the spike in a separated region ahead of the main body and extend up to the reattachment point. The peak heating and local maximum pressure was found to be at the reattachment point because of the shear layer created by the spike, which passes through the reattachment shock wave. It was also observed that as compared to an aerospike the aerodisk performed well in terms of drag reduction and aerodynamic heating reduction.
Allen and A.J Eggers, Jr [9] in 1953. They made a significant contribution to hypersonic aerodynamics by proposing the use of blunt nose for re-entry vehicles and established that although a blunt body offers a high pressure drag, it has a detached shockwave which minimizes heat transfer to the vehicle as compared to a slender body with attached shock wave. Allen and Eggers obtained an expression for heat transfer rate to a blunt body, and proposed that the stagnation point heat transfer rate is inversely proportional to the square root of the leading edge radius of the blunt body. In comparison to a slender body design, the blunt body produces more drag; hence slender bodies are preferred over blunt bodies, keeping aerodynamic drag as a main parameter. It has been established that although pointed bodies generates less drag, the amount of aerodynamic heat produced is tremendously high and it varies with the cube of the freestream velocity. Therefore for high speed flow slender bodies are not utilized, as a huge thermal gradient can lead to the failure of the sensitive electronic equipment employed in the vehicle. These thermal gradients can also lead to the ablation of the material used for the structure which can cause undesirable aerodynamic performance and may even cause disintegration of the vehicle itself.
The severe increase in pressure drag for blunt is caused by the detached shock which is formed just ahead of the nose and can be reduced by changing the flowfield ahead of stagnation region. One such mechanism to do this is the use of an aerospike or an aerodisk. An aerospike replaces a strong bow shock with a series of oblique shock and promotes the separation of flow and creates shear layer. This shear layer is propagated in the flow direction and reattaches at forebody to form a zone of recirculating flow. There is a high local heat flux and pressure across the zones where reattachment occurs, but owing to the reattachment zone, the pressure and heat flux decrease ahead of the blunt nose.
The effectiveness of the aerospike is further increased by introducing a blunt face aerospike known as aerodisk and sizable research is oriented towards the findings of aerospike and aerodisk shapes and sizes.
Alexander SR [10] at the Langley Pilotless Research Division carried out numerous firing tests on blunt and slender nosed bodies at various Mach number ranging from to. He proposed that mounting a cylindrical thin rod with large conical tip on blunt forebody may alter the fineness ratio and effect of
increase in weight. It was also observed that although there was a small drop in the drag of the bodies using windshield but it was still more than that of the pointed forebody at Mach number 1.37.
Mair W.A [11] did a comprehensive study on spiked blunt bodies which is considered as a landmark study in this field. He did experimental investigation of the flowfield around spiked blunt body for hemispherical and flat cylindrical models at a freestream Mach number of 1.96 and Reynolds number of 1.65 x 105. He experimentally observed the flowfield around spiked hemispherical and flat cylindrical bodies in relation to the effect of spike length. Mair was the first one to observe the flow instability around spiked blunt bodies. He also observed that shock wave at the probe tip changed its shape in a self-sustained manner for a certain range of its length. He observed that the flow oscillation with respect to pressure variance between the flow inside the recirculation zone and the aft of the reattachment shock was substantial for certain range of probe length on a cylindrical model.

Jones J. J. [12] did another investigation for flow around a hemispherical nose with pointed rod protruding from the nose at Mach number 2.72. The length of the protruding rod varied from 1 to 6 times the radius of the hemispherical cylinder. Jones pointed out that there exist only one streamline that stagnate on the reattachment point of the fore body surface, and this streamline divided the flow between the recirculating zone and the non-circulating main flow.

Mehta RC and Jayachandran T [13] numerically studied the effect of viscous flow around a heat shield of launch vehicle. The studies were performed for both hemisphere-cylinders with and without forward facing spike. Reynolds Averaged Navier-Stokes equation was discretised using cell-centred finite volume and solved with a time marching approach. The freestream Mach number ranged from transonic to supersonic flow i.e. 0.9 to 3.0 and Reynolds number ranged from 33.5 x 106 to 46.75 x 106 per meter. Both experimental and computed results obtained for typical heat shield without forward facing spike showed good agreement with each other. It was observed that for spiked blunt body the flow separation zone depends on the Mach number and an increase in the Mach number results in separation conical angle to become sharper. It was also found that the bow shock angle increase because of the presence of spike and deployment of forward facing spike gives a reduction in drag.

Feszty et al. [14] in 2000 carried out numerical simulation over axisymmetric spiked blunt body in supersonic and hypersonic flow regimes of Mach numbers 2.21 and 6 respectively and at Reynolds number to 0.12 x 106 for supersonic flow and 0.13 x 106 for hypersonic flow, based on the diameter of the blunt body. The diameter of the base body was set as unity while length of the spike was made equal to the diameter of the blunt body with spike diameter is 0.06D and cone angle at the tip of the spike of 30°. An implicit upwind algorithm based on Roe's scheme was used in conjunction with time marching procedure to solve the transient, compressible Navier Stokes equations so as to study the nature of pulsating flow over spiked blunt body. It was found that the experimental and computed results were almost identical in terms of predicted time periods, while the computed pressure amplitudes were marginally underestimated in supersonic flow and overestimated in the hypersonic flow. Qualitatively both the computed and experimental data showed that the flow features were in good agreement. It was also confirmed that annular jet is formed at the foreshock-aftershock interaction which inflated the separation region to move extra downstream.

Srinivasan and Chamberlain in 2004 [15] studied the drag reduction by means of heat addition to the separated region of turbulent boundary layer by combustion of hydrogen gas. Two geometries were considered viz. a hemispherical nosed body with an spike and an ogive nosed body with an aerospike at a Mach number of 2.2. It was found that spiked ogive nose cylinder and hemispherical nose cylinder had shown 35%-40% reduction in drag, beyond that produced by conventional aerospike, because of hydrogen combustion. Numerical simulations were also carried out for hydrogen injection without combustion and it was found that reattachment shock was not eliminated. The additional drag reduction was found to be a result of the main shock wave getting diffused and also the elimination of shear layer reattachment shockwave on the blunt body.

Muhammad Asif et al. [16] in 2004 did numerical simulations on supersonic and hypersonic flow around forward facing aerospikes attached to a blunt body. Aerospike of different geometries and l/D ratios were investigated to study the flow around the forebody and its influence on static aerodynamic coefficients. The influence of the aerospike shapes on the aerodynamic coefficients was investigated at Mach number 1.89 and Reynolds number of 0.38 x 106 per meter and angle of attack between 0° to 10°. The second set of investigation were made to study the effect of l/D ratio on the aerodynamic coefficient at Mach numbers of 5, 6.8 and 8, and a Reynolds number of 1 x 106 per meter and angles of attack 0, 2 and 6 degrees. It was found that the shapes of the aerospike influence the aerodynamic parameter but the effect reduces with increase in the angle of attack. Aerospike with hemispherical nose and cylindrical body showed the minimum drag coefficient and maximum lift coefficient in the supersonic flow regime. In the case of hypersonic flows, an increase in l/D ratio results in reduction in drag but with a substantial loss of static stability. In both the cases the flow around the aerospike is featured by a conical shock emanating from the tip with a separated region in front of the forebody resulting in reattachment hockwave.

3. CONCLUSION

We have observed that thermal heating is much more on the fin L.E as compare to the other portion of the fin surfaces and of course, the normal forces are very important parameter for the designers to design any high speed vehicles and hence we must design a fin in such a way that it should sustain the higher load or normal forces, keep in mind that it should have less projected area and high strength and low weight. Therefore geometrical properties of the initial design phase must be carefully selected by the designer to avoid excess wall shear stress which may cause the failure of the fin structures.

4. REFERENCES

1.Mazzoni, J. A., et al., Aerodynamic Heating on VSB-30 Sounding Rocket, Proceedings, 18th International Congress of Mechanical Engineering, Ouro Preto, Brasil, Nov., 2005

2. Cayzac, R., et al., Navier-Stokes Computation of Heat Transfer and Aero-Heating Modeling for Supersonic Projectiles, Aerospace Science and Technology, 10 (2006), 5, pp. 374-384

3. Bao, W., et al., Effect of Structural Factors on Maximum Aerodynamic Heat Flux of Strut Leading Surface, Applied Thermal Engineering, 69 (2014), 1-2, pp. 188-198.

4.Allen, H.J. & Eggers, A.J. (1953). A Study of the Motion and Atmosphere at Supersonic Speed. NACA TR-1381.

5.Sahoo, D., Das, S., Kumar, P., & Prasad, J. K. (2016). Effect of spike on steady and unsteady flow over a blunt body, Acta Astronautica, (pp. 521-533)

6.Humieres, D., & Stollery, J.L. (2010). Drag reduction on a spiked body at hypersonic speeds. The Aeronautical Journal, 14(1152), (pp. 113- 119).

7. Ahmed, M., & Qin, N. (2010). Drag Reduction using aerodisks for hypersonic hemispherical bodies. Journals of Spacecraft and Rockets, 47(1), (pp. 62-80).

8. Mehta, R.C. (2000). Numerical heat transfer study over spiked blunt bodies at Mach 6.8. Journals of Spacecraft and Rockets ,37(5), (pp. 700- 3).

9. Allen, H.J. & Eggers, A.J. (1953). A Study of the Motion and Aerodynamic heating of Ballistic Missile entering the earth Atmosphere at Supersonic Speed. NACA TR-1381.

10. Alexander, S.R. (1947). Results of Tests to Determine the Effect of a Conical Windshield on the Drag of a Bluff Body at Supersonic Speeds. NACA RM -L6K08a.

11.Mair, W.A. (1952). Experiments on Separation of Boundary Layers on Probes in Front of Blunt-nosed Bodies in a Supersonic Air Stream. Philosophy Magazine, Series 7, 43(243), (pp. 695 716).

12. Jones, J.J. (1952). Flow Separation from rods ahead of Blunt Nose at Mach number 2.7. NACA RM, L52E05a.

13. Mehta, R. C., & Jayachandran, T. (1997). Navier Stokes solution for a heat shield with and without a forward facing spike. Journals of Computers & Fluids, 26(7), (pp. 741-751).

14. Feszty, D., Badcock, K.J., and Richards, B.E. (2000). Numerical simulation of the hysteresis phenomenon in high-speed spiked body flows. AIAA, 0141

15. Srinivasan, G. R., & Chamberlain, R.R. (2004). Drag reduction of spiked missile by heat addition, AIAA Atmospheric Flight Mechanics Conference and Exhibit, 4714.

16. Muhammad A., Zahir, S., Nasir K., and Mahmood A. (2004). Computational investigations Aerodynamic forces at Supersonic/Hypersonic flow past a blunt body with various forward 187 | P a g e facing spikes. AIAA, 22nd Applied Aerodynamics Conference and Exhibit.