ISSN No:-2456-2165

Review on Hypervelocity Impact over Different Blunt Body

Suraj Sambhajirao Patil
Lovely Professional University

Sahil Chandel
Lovely Professional University

Abstract:- At hypervelocity flows the design place a major role. To sustain such impact the design, need to be more aerodynamic. Blunt body is key factor to achieve and sustain those hypersonic velocities that is mainly focused beyond Mach number 5. Blunt body design in aerodynamic and structural aspects become more crucial factors in hyper velocity vehicles. The review mainly focuses on effective blunt body design. In order to achieve the effectiveness various radius of curvature taken into account and comparative study were done. Along with that flow compressibility, chemical reaction, aerodynamic heating, stagnation conditions were discussed. Extensive review was done to select most efficient blunt body design in both aerodynamic and structural aspects.

I. INTRODUCTION

Hypersonic region means, one is travelling five times faster or more than that in compare with speed of sound. The flow compressibility, chemical reaction, stagnation conditions, aerodynamic heating plays important role during high speed flight. Further increase in Mach number, produces more strong shock around body and its start moving closer to the body surface. The ionization of gas occurs and the radiative heat transfer starts dominate the heat transfer in flow. All occurs at hypersonic region. The paper will discuss the variation in design parameters of nose cone and its effect in hypersonic region. Stagnation point, aerodynamic heating calculation as well as detail study of the both aerodynamic and structural parameter will be presented in paper. In order to find the efficient nose shape design for the hypersonic region, experimental study has been done by comparing different nose shape design. The flight conditions have set to be at sea-level reference. This efficient design was chosen by altering the nose cone radius of curvature simultaneously. Bluntness ratio are set to be varying according to the design parameters

At hypersonic region, gas dynamics plays important role. There is certain flow characteristic occur at hypersonic velocities. It can be characterized by certain physical phenomena i.e. Shock Layer, Entropy Layer, Aerodynamic heating, Low Density. As the Mach number increases strong shock wave formation occur on the body. It starts moving closer to the body resulting high rise in the stagnation temperature and other stagnation flow condition such as pressure and density. It is also important to study the flow behavior on the downstream of the shockwave as flow properties behave very differently in the downstream of the shockwave in compare to the upstream of the shockwave. The aerodynamic effects can be simulating in the software to validate the result and to visualize the hypersonic flow over body. This In structure, blunt bodies are the perfect shape to any hypersonic craft. Blunt shape nose is mostly used nose of the re-entry vehicle. As the larger surface leads to decrease in the temperature and also leads to form a shockwave far from the body of hypersonic craft. To design the blunt nose cone for any hypersonic craft, studies related to the field of aerodynamic and structures needed. Blunt nose cone structure provides the strength to nose to sustain the high temperature during hypersonic speed. Structural part also covers the study of the material required for the outer parts of vehicle along with the heat transfer field. As most of the metals and higher aluminum alloys have melting point around 2000 K, the stagnation temperature can easily rise up to melting temperature of the most metal and higher aluminum alloys. So it is important to study the structural and fabrication part for the design of hypersonic craft along with aerodynamics. Structural Analysis can be carried out using the Finite Elemental Analysis.

There are many researches going on in the field of hypersonic region. In depth studies in field of aerodynamics and structure are ongoing presently.

II. REVIEW OF LITERATURE

[1] Influence of Nose Radius of Blunt Cones on Drag in Supersonic and Hypersonic Flows

The paper deals with influence of nose cones design in supersonic and hypersonic flows, varying its nose radius in every case. The free stream conditions of flows are sent to be at Altitude 25000-meter, Temperature 221.65 K, Pressure 2.51102E+3, Density 3.94658E-2, Viscosity 1.40644E-5. The geometrical modelling of the nose cone is varying with respect to AOA (Angle of Attack), Bluntness Ratio. The angle of attack is varying with the factor of 2.5. Different blunt shapes were considering during the experiment setup. The paper also provides the numerical approach to perform the simulation and to select the efficient nose design for drag reduction in supersonic and hypersonic field.

- The selected free stream conditions are Pressure 2.51102E+3, Altitude 2500-meter, Temperature 221.63 K, Density 3.94658E-2.
- Th e nose radius is varying with diameter 0.1, 0.2, 0.3, 0.4 factor with original diameter.

- The difference between the actual values and the theoretical values for pressure and temperature are 90Pa and 4K simultaneously.
- Lowest drag was seen for the cone with 30-degree cone angle.

[2] Experiments on a Blunt Cone Model in a Hypersonic Shock Tunnel

The paper shows the experimental investigation on 60-degree of blunt body is subjected to study of the effect of drag and aerodynamic heating in the shock tube tunnel. The aerodynamic drag and heating is to be considering in the following experiment. The experimental setup consists 60-degree apex-angle blunt cone model along with an accelerometer balance system and platinum thin film gauges for simultaneous drag and heat transfer measurements. The shock tunnel used during the experiment comprises of a shock tube, a conical nozzle, square test section and a dump tank along with a high efficiency vacuum pumping system. Figure 7 in the paper shows schematic diagram of blunt cone model with electrode arrangement for flow visualization experiment which can be taken as reference while designing the blunt nose cone

- Different gases (air, carbon dioxide, argon and helium) are injected into the hypersonic flow field at different enthalpy levels.
- Stagnation point aerodynamic heating varies inversely as the square root of nose radius.
- The heating problems are reduced to some extent by large angle body configurations, but the drag increases.

[3] Hypersonic flow past nose cones of different geometries: A Comparative Study

The work presents the comparative study of the different type of nose cone design at hypersonic region, typically at Mach 5.8. The main focus is to determine the geometry and parameters of the nose cones that provide minimum aerodynamic drag and heating. Studies on spherically blunted and parabolic nose cones was performed for different fineness ratios at zero AOA (Angle of Attack). Detailed comparison of the shock features in the blunted and parabolic nose cones for different fineness ratios is also shown in order to determine its influence on aerodynamic drag in hypersonic velocity region.

- Detail comparison of bluntness ratio, fineness ratio is given to design the efficient geometry for the nose cone. Studies shows that the fineness ratio less than 1.2 gives drag reduction values for blunt nose cone while for the parabolic nose cone the fineness ration greater than 1.2 gives superior drag reduction value.
- Total drag reduction coefficient is mainly function of fineness ratio and pressure coefficient.
- Comparison of Mach no. for parabolic and blunt nose cone was given at fineness ratio of 4.7, 3.6, 1.2, 1 respectively.

[4] Aerothermodynamics Design and Optimisation of Hypersonic Research Re-entry Vehicle

The spacecraft travelling at hypersonic speed results rapidly increase in the aerodynamic heating value. Blunt body travelling at higher speed region creates the bow shock wave around body. This paper discusses the effects of aerodynamic drag and heating on two design of nose cone that is one with spike and other without it. Total design and optimisation was discuss and shown in the paper. The experimental setup was at Mach 9.0 to analyse the effect of aerodynamic heating and drag.

- Aerospike nose design shows greater reduction in the aerodynamic drag and heat generation value due to shock detach from surface up to certain distance.
- Pressure is set at 50 Bar, Mach number 9.0 and mesh and analysis is done at Gambit software.
- Length of the spike on nose cone vary temperature according to the length, as result decrease overall temperature on the body.

[5] Computational Flow Analysis of Hypersonic Re-entry Blunt Body Using Fluent and Gambit

This paper discusses the aerodynamic effect of nose cone at hypersonic velocity region. Also discuss the effect of blunt design on aerodynamic heating and drag. The analysis was done with help of powerful tool like ansys and gambit. During the analysis setup the velocity mode is selected at inlet and exit as far field pressure value. The Mach number is set at 8.0. The mesh is generated in gambit and then imported in ansys to performance simulations.

- Simulation approach was mention in the paper. Meshing was done in Gambit, imported in fluent for analysis.
- Designing criteria for choosing the efficient nose design for providing less

[6] Influence of Nose-Tip Bluntness on Conical Boundary Layer at Mach 10

The paper describes the boundary layer formation on the specific blunt nose cone design. The Von-Karman nose cone design is subjected in the low enthalpy hypersonic wind tunnel. During the experiment, the flow is accelerated hypersonic speed for fraction of time. The free stream Mach number is set in between the $9.5<=M<=12$ range. The length of the nose cone is about 800 mm long with variation in radius. The main aim of the experiment is to validate the different results on variation of the radius of the nose cone. The radius of nose cone is varying in between the range of 0.05 and 10 mm. Flow parameter are then studied with help of the thermostats and the pressure gauge. Along with the flow conditions, flow visualization is done using the schlieren technique. It also validates the shockwave formation at hypersonic speed. The turbulence in the flow leads to the vorticity in the flow. Higher the turbulence more over the vorticity formation in the flow at hypersonic region.

- Flow compressible parameters are well explained in the paper on the different flow region. From the supersonic to the hypersonic the flow conditions are stated along with that the flow behavior explained in the hypervelocity's region
- The boundary layer formation can be also seen in the hypersonic region, noting that the behavior can be different from normal boundary layer at subsonic region

- The free stream Mach number for the experiment are hold in range of 9.5 to 12 Mach which is hypervelocity region
- Schlieren images also shows that wave packets break down to turbulence near the middle of the transition region

[7] Rapid Prediction of Hypersonic Blunt Body Flows for Parametric Design Studies

The work in the paper shows the utility and the practical advantage of using the numerical and computational method to derive the result of the hypersonic flow over the blunt body. The collective information was used as a case study to simulate the hypersonic flow over the blunt body. The different numerical approach is compared to discuss the various results and efficient method is selected further to solve the governing equation and to reduce the computational cost. Along with that the improvement method to solve the Navier-Stokes equation is determining by the comparing the result of computational analysis. The results were collected for accuracy and computing time for each method including under solved Navier-Stokes equation. During the experiments the adiabatic circular cylinder is been used as hypersonic blunt body.

- Numerical method is implicated to solve and validate the result in the computational fluid dynamics software. Code is very well implicated and tested for the hypersonic flow region.
- The code is able to solve the Navier-Stokes equation for the hypersonic speed
- The simple method was use for the weak viscous inviscid interaction between the boundary layer and free stream Mach number were discussed.
- The pressure distribution, Shear stress, Heat flux were calculated using computational support for the adiabatic circular cylinder

[8] Aerodynamic Design Analysis of The Hexafly-Int Hypersonic Glider.

This paper deals with the aerodynamic performance studies of the experimental flight test vehicle currently under development. The vehicle is design by considering the Aerodynamic, thermal, mechanical loads in flight conditions. This helps to design the vehicle conceptually. The flight will experience the hypersonic flow at the altitude of 50 km. The model is set to be experience the schedule Mach 8 speed at given altitude During flight different experiments conducted to study the aerodynamic effect, mechanical-thermal loads. The main project highly depended on the existing data of pervious project of the glider. Along with the aerodynamic effect, the structural and thermal loads are simultaneously calculated in flight. The design is inspired by the previous European community funded projects ATLLAS I & II, LAPCAT I.

- Numerical result shows the compliance of vehicle aerodynamic performance with flight envelope expected for the glider
- Aerodynamic result highlighted that vehicle aero shape features a lift to drag ratio greater than 4 for Mach number ranging in between 6 to 8.
- No significant effect of sideslip on aerodynamic efficiency, total pitching moment and on trim ability.
- Viscous effect seems significant at lower Mach number region.

[9] Numerical Study of Hypersonic Flow Over Re-entry Configuration with Different Chemical Non-Equilibrium Models

Paper consist the two different chemical reaction model on hypersonic reentry flow simulations that are numerically investigated. There are two typical models examined, whose thermos-chemical non-equilibrium flow field are computed by multi-block finite volume code using two temperature model. The FIRE II & RAM-C II model was used to conduct the test. Result show the FIRE II case, the two models yield significantly different distribution profiles of ions and electron. The shock wave studies are being done on the model. The paper also consists the chemical reaction code to carry out the chemical equilibrium analysis of the vehicle at hypersonic velocities.

- Gupta's and Park's 11-species chemical reaction rate model is being utilize to demonstrate the hypersonic flow over RAM-C II vehicle model.
- Gupta's model gives more piece wise result for the reentry vehicle travelling at hypersonic field
- Reentry vehicle speed is considering as 10 km/sec during the simulations.

[10] Multi-Objective Design Optimization of Blunt Body with Spike and Aerodisk in Hypersonic Flow.

Blunt body travelling at hypersonic speed will lead to experience the aerodynamic drag and aerodynamic heating during the flight. In order to reduce the overall aerodynamic drag and heating the aero disk and spikes are applied to the blunt body. In this paper, the influence of the aero-disk on aerodynamic drag and heating of the hypersonic blunt body are studied numerically. The paper also derived the advantage of using the aero disk blunt body over non-aero disk one by numerical method. The results also indicate the advantage of aero disk blunt body. The two methods which were used for numerical solution are weighting method and NSGA-II method

- The result shows that the drag coefficient and maximum heat flux of the blunt body for the configurations with the aero-disk are reduced by 34.09% and 51.06% respectively compared with the configuration without aero-disk.
- In order to reduce the aerodynamic drag and heating, the aero disk is installed in effective way.
- Practical prototype shows that the aerodynamic drag decreases with increase in length of spike.
- The drag value has inverse relation with total heat flux value.

[11] A Validation Study for a Hypersonic Flow Model

Paper consist the in-depth knowledge of hypersonic re-entry simulation effectively at various Mach number region. The also consist the reentry code for simulating the atmospheric reentry of spacecraft at hypersonic speeds. The development roadmap consists higher-order discretization including shock capturing and more extensive physics. Paper will help while designing the blunt nose for the reentry vehicle as well as to validate the result during the simulations. This paper also consists depth information related vehicle performance prediction, aerodynamic forces, radiative heat transfer, thermochemical non-equilibrium model. This paper will help while performing the simulation for the blunt body at hyper velocities region.

- ➢ Heat transfer and pressure on the attached flow region of the first cone are well resolved even on coarse meshes-numerical error estimates are very small
- ➢ The simulation model will have predicted well using a perfect gas model

[12] Large Scale Hypersonic Separated Flows at Moderate Reynolds Number and Moderate Density

In many aerospace applications, such as spacecraft which operated at hypersonic region, for them the typical Reynolds and Knudsen number under these circumstances are typically of order of the 10^5. The flow is simulated using CFD. The paper considers the large laminar hypersonic low enthalpy, moderate density flow. The main advantage of such configuration is that is largely eliminate the presence of the boundary layer. The paper further discusses the methods to solve N-S models to validate the result.

- ➢ A code to code comparison between different models and time accurate N-S solution yield almost same slow after reaching steady state.
- ➢ Density Variation at hypersonic region are clearly visible during simulation process.
- ➢ Code validation, time-accurate solutions of CFD is needed.

[13] Stagnation Point of Blunt body in Hypersonic Flow.

The paper purpose was to present a method of calculation devised to yield all the important information on symmetric inviscid hypersonic flow in the stagnation point region of blunt body. Equations are presented giving velocity, pressure, detachment distance and vorticity. The values of shock detachment distance and body pressure are coefficient are compared with the experimental data. The pressure comparison shows about the better result of hypersonic flow. Different theories are compared to get the better result for the hypersonic region on blunt body. It also demonstrates Hayes result are valid in the stagnation point region.

- ➢ The local velocities are varying linearly with the value of x. This value of vorticity remains constant along the surface. The vorticity is function of two factors mainly, one is density of flow and Reynolds number.
- ➢ Experiments shows that the boundary layer thickness is negligible near the nose cone.

[14] **Investigation of Mach Number Effect on Aerodynamic Loading of Nose Cone of Missile**

A combined experimental and numerical studies was completed to determine the effect of Mach number on aerodynamic loading produced by articulating a nose cone. The articulation was modeled as bending of the centerline of the nose cone over supersonic region of Mach number 4.3 and subsonic region 0.5 of missile. The result is compared.

To previous experiments using a Mach 2.0 missile. The angle of attack from -15 to 15 degree. It also consists the in-depth field studies of the aerodynamic load while at flight speed of supersonic region.

- ➢ For Mach number 0.5, nose cone significantly produces less moment in comparing with other two Mach number values.
- ➢ Drag forces are computed in both ways by numerical methods as well as CFD analysis. The comparison of two results shows the formation of the induced drag on individual design of nose cone.

[15] Design Optimization and Analysis of Rocket Structure for Aerospace Applications

For designing a rocket structure, different rocket parts are individually designed and later assembled together in a computer assisted software. After that, the rocket design is analyzed on computer software that facilitates structural and aerodynamic analysis of the rocket design. It provides the different drag and pressure values on rocket design and the point at rocket where that is acting upon. For the optimization of rocket design, pressure and aerodynamic loads are balanced as to maintain aerodynamic shape and to ensure structural safety of rocket.

- ➢ Design of nose and nose cone are made in such a way that the aerodynamic shape of rocket is maintained and nose-tip experience very low air resistance.
- ➢ While designing the main body, it is very important to keep sufficient space for fuel storage.
- ➢ The fins are designed in such a way that centre of gravity is balanced and rocket is stable on the lateral axis.
- ➢ The stress is maximum in the fin and blade regions. So, these two parts are optimized by varying the thickness in these regions.

[16] Design and Structural Analysis of Solid Rocket Motor Casing Hardware used in Aerospace Applications

Solid Rocket Motor is generally preferred because of its simple arrangement and it can be easily manufactured. It is designed in way as to bear extreme temperature and pressure. Durability, toughness and availability are criteria while choosing the material for manufacturing. The proper arrangement of motors in rocket is important to maintain the stability of rocket. Design of motor casing is analyzed using computer assisted software to look for pressure and aerodynamic loads acting on the structure. This is very essential as to ensure high fatigue strength of the structural frame. The main advantage of using solid propellant in rockets and missiles is the very high durability. Also, it can be prepared to launch in a short span of time.

- ➢ The motor casing is designed in a way such that it can bear high temperature and pressure. Thus, it is constructed using robust metals and composites.

- The motor casing is insulated using material with high heat tolerance.
- Generally, Converging Diverging nozzle is preferred since rocket usually travels at supersonic speed.

[17] Structural Design and Fabrication of a Rocket

To design a single- stage rocket, having high structural strength and less weight. For the fabrication of structural frame, different materials were analyzed on the basis of different parameters like physical properties, availability of material and price. Initially, the aerodynamic design for single stage rocket was finalized. It was followed by considering several deployment techniques to deploy the payload. Then, a suitable material is chosen for the fabrication of the rocket.

- For the designing of rocket, circular cross section is preferred. Since it enhances stability of rocket while in air and also circular tubes are easily available for the fabrication of the rocket.
- Choose appropriate material for the rocket fabrication that has high robustness and less price as well.
- Surprisingly, the base of rocket creates highest amount of drag force.
- For the payload deployment, top deployment technique is preferred since it has very simple arrangement and has a positive contribution in rocket stability.

[18] Structure optimal design of wrap-around fins of the Rocket

Structural optimal design of the wrap-around fins is subjected to aerodynamic loads of the rocket. Six geometry parameters of the wrap-around fins were chosen which include fin length, thickness, leading edge sweepback as the design variables, its intensity and distortion were taken as the restrictions, the finite element method as analytical tool and non-linear program as search algorithm.

- After optimization, weight reduction is achieved.
- The weight factors analysis results show that the design variables which have the greatest influences on optimal target and restrictions are the central angle of wrap-around fins and the thickness.

[19] Structure of Reusable Hypersonic vehicles

This research paper deals with problems associated with reusable hypersonic vehicle. It discusses the methods for selecting the suitable material for flight envelope. It lists out several methods to prevent flight structure from heat loading.

- Technical and economical efficiency can be enhanced by increasing the payload mass fraction. It is achieved by increasing specific impulse of propellant or reducing the Airframe weight.
- Airframe weight is defined by: -

i) Properties of materials used
ii)Structural configuration
iii) Production technique

- Performance of structural material is estimated by value of its specific strength and specific stiffness.

III. CONCLUSION

As per the review it is clearly evident that the more the curvy the more the aerodynamic efficiency of bluff body but whenever we a making more aerodynamically we will face the problem of lack of response time. So the prescribed radius needs to be carry forward to get the optimum solution. At the same without flow coupling the shock wave need to be analyzed for better aerodynamic efficiency, during that time material absorption and emission rate need to neglected.

REFERENCES

[1]. "A. Hemateja, B. Ravi Teja , A. Dileep Kumar , Rakesh S.G", "IOP Conference Series: Materials Science and Engineering", "2017".
[2]. "Niranjan Sahoo", "16th Australasian Fluid Mechanics Conference", "2007".
[3]. "Ashish Narayan, S Narayanan and Rakesh Kumar", "Simulation: Transactions of the Society for Modeling and Simulation International", "2017".
[4]. "Manjunatha H, C. N. Nataraj", "International Journal of Science and Research (IJSR)", "2014".
[5]. "N. S. Harshavardhan .R; Sanjana .K; Sai Sharan .K ; Srinivas .G", "International Journal of Scientific & Engineering Research, Volume 5, Issue 5", "May-2014".
[6]. "Guillaume Grossir, Fabio Pinna, Olivier Chazot", "AIAA Journal", "1999".
[7]. "W. Schuyler Hinman, Craig T. Johansen", "Aerospace Science and Technology", "2016".
[8]. "Giuseppe Pezzella, Macro Marini", "International Space Planes and Hypersonic system and Technology Conference", "2015".
[9]. "Jiaao Hao, Jingying Wang, Chunhian Lee", "Acta Astronautica ", "2016".
[10]. "Jie Huang, Wei-Xing Yao", "Aerospace Science and Technology", "2019".
[11]. "Brian Carnes, Derek Dinzl, Micah Howard, Sarah Kieweg, Jaideep Ray, William Rider, Tom Smith, Greg Weirs". "ASMEVVS2018 Conference", "2018".
[12]. "S.L.Gai, R.Prakash, A. Khraibut, L.Le Page, S. O'Byrne", "AIP Conference Proceedings","2019"
[13]. "Ting-Yi Li, Richard E. Geiger", "AIAA Conference", "2014".
[14]. "Eric J. Stephen, Gregg Abate, Tom McLaughlin, Manuel Figueroa, Steven Ringenbach", "AIAA Conference","2019".
[15]. "Anoop Thankachen, Santosh Kumar", "International Journal of Engineering Trends and Technology (IJETT)", "2015".
[16]. "Dinesh Kumar, Shishira Nayana and Shravya Shree", "Journal of Aeronautics & Aerospace","2016".
[17]. "Michael Lostoski, Matthew Schwenning, Jeffrey Szucs"," IdeaExchange@Uakron","2016".
[18]. "XUE Song-Hai, YANG Shu-Xing, ZHAO Liang-yu", "Journal of Solid Rocket Technology", "2009".
[19]. "V. Lazarev", "9th International Space Planes and Hypersonic Systems and Technologies Conference", "1999".