International Journal of Research in Engineering, Science and Management
Volume-3, Issue-2, February-2020
www.ijresm.com | ISSN (Online): 2581-5792

Investigation On Drag Force Reduction by Aerospikes Using CFD

T. Banu[1], K. Ganesh Babu[2], N. Mohammed Raffic[3], A. J. Maha Boop Badhusa[4]
[1]*Assistant Professor, Dept. of Aeronautical Engineering, Nehru Institute of Technology, Coimbatore, India*
[2]*Principal, Chendhuran College of Engineering & Technology, Pudukottai, India*
[3]*Assistant Professor, Department of Mechanical Engineering, Nehru Institute of Technology, Coimbatore, India*
[4]*UG Scholar, Department of Aeronautical Engineering, Nehru Institute of Technology, Coimbatore, India*

***Abstract*: Drag force reduction over missiles has become a common problem of study in the present scenario. The presence of spikes in forward facing missile has shown a remarkable influence in reducing the drag force created with the base cone blunt shape. The present work considers the brahmos missile model for analyzing the presence of spikes in hemispherical and triangular form in reducing the drag through ANSYS CFD module by considering a free flow stream with a mach number of 2.75 at zero-degree angle of attack over the missile. The results obtained have shown that the hemispherical model has reduced the drag by 4% than the triangular spike model. The variation of spike geometry and dimensions may be considered for further study.**

***Keywords*: Aerospikes, Brahmos Missile, CFD, Drag reduction, Hemispehrical Spike, Triangular Spike.**

1. Introduction

The study of a moving body in the presence of air and its related response over the flow is an interesting and essential area of aeronautical and automobile engineers as the study has more valuable contribution towards the design of aircrafts, missiles and rockets. The aircraft and missiles are bodies that are heavier than air and so can support their weights only if they produce a force to counter it. This force can be either lift force generated by the flow of air over the wings and body or generated by means of an engine in the form of thrust. A moving body in the presence of air experiences an opposing force in motion called as drag which has to be generally countered by engine's thrust. The drag force created has depends upon fineness or bluntness and size of the body. To minimize the drag force one has to choose the aerodynamic shape without compromising the essential and desired functional requirements. Aerodynamic characteristics of various external components and their configuration aid their selection towards an optimum missile performance with respect to its lift and drag characteristics, aerodynamic stability, maneuverability, etc. Comprehensive and accurate data to enable a missile technologist to zero-in on a particular configuration is not readily available since much of the essential data is classified. The fundamentals of many technically specialized areas such as aerodynamics, thermodynamics (mainly heat transfer), kinematics, propulsion, structural design-are a necessity though it makes the task of the aeronautical design engineer rather complex. The body of the missile may be divided into three major sections the - fore body or the nose, the mid-section and the aft or boat-tail section. The Fig. 1 shows the schematic diagram of missiles.

1) Nose Section

Forebodies may have many varieties of shapes, most common of which are conical, ogival, power series or hemispherical. These shapes are used primarily on the missiles of supersonic speeds and are generally selected on the basis of combined aerodynamic, guidance and structural considerations. A hemispherical nose has very high drag from the aerodynamic drag or performance standpoint, but it is excellent from the standpoint of structural integrity, resistance to aerodynamic heating and amenability to certain types of guidance like infrared guidance. Since the pressure or wave drag may be several times that due to friction at supersonic speeds, careful selection of the nose shape needs attention to assure satisfactory performance of the overall system.

Fig. 1. Schematic diagram of missiles

Conical forebody has given way to other types because of relative disadvantages but the conical one is the basis for the study of aerodynamic characteristics due to its simplicity. Briefly some of the flow characteristics about which an aero engineer will have to be very familiar are the formation of a shock wave, the shock angle, streamlines or flow direction and air properties between the shock wave and surface of the body. The supersonic flow over a cone has characteristics which are similar in appearance as that of a conical one but are markedly different in nature from those corresponding to two-dimensional flow (i.e., flow over a wedge). The Fig. 2 shows the drag created on different sections based upon their shape at forward section.

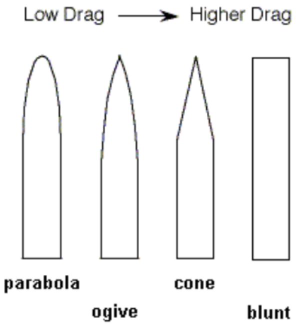

Fig. 2. Drag on different sections

2) *Mid-Section*

The mid-section in most missile configurations is cylindrical in shape. This shape is advantageous from the standpoint of drag, ease of manufacturing and load carrying capability. It is known that the total reaction of the missile at any instant has two components, the lift (components at right angle to the direction of airflow) and drag (those parallel to the direction of airflow). These may be positive or negative. It becomes desirable to have a greater lift than the drag and this can be done by using a curved surface. Angle of attack is the direction of the reaction force with respect to the free stream direction. Even at zero angle of attack, called as the zero-lift drag (x = O), some lift can be obtained by using what are called as airfoil sections.

3) *Tail Section*

Base drag is the drag resulting from the wake or “dead air” region behind the missile. Base drag is less of a problem during powered flight but during free flight it can account for as much as 50% of total drag. Base drag can be reduced by tapering the tail (boat tailing).

A boat-tail is the transition section at the tail of a rocket (or other vehicle) that gradually narrows the body down to the motor diameter. It thereby helps reduce base drag. Base drag is a component of aerodynamic drag caused by a partial vacuum in the missile's tail area. The vacuum is the hole created by the rocket's passage through the air. Base drag changes during flight. While the motor is firing, the drag is minimal since the tremendous volume of gas generated by the motor fills this void. The drag takes a sharp jump at burnout when this gas disappears.

4) *Drag Reducing Aerospike*

A drag-reducing aerospike is a device (see Nose cone design) used to reduce the fore body pressure aerodynamic drag of blunt bodies at supersonic speeds. The aerospike creates a detached shock ahead of the body. Between the shock and the fore body a zone of recirculating flow occurs which acts like a more streamlined fore body profile, reducing the drag. concept was used on the UGM-96 Trident I and is estimated to have increased the range by 550 km. The Trident aerospike consists of a flat circular plate mounted on an extensible boom which is deployed shortly after the missile breaks through the surface of the water after launch from the submarine. The use of the aerospike allowed a much blunter nose shape, providing increased internal volume for payload and propulsion without increasing the drag. This was required because the Trident I C-4 was fitted with a third propulsion stage to achieve the desired increase in range over the Poseidon C-3 missile it replaced. To fit within the existing submarine launch tubes the third-stage motor had to be mounted in the center of the post-boost vehicle with the reentry vehicles arranged around the motor.

At the same time (middle 1970s) aerospike was developed in KB Mashinostroyeniya (KBM) for 9M39 surface-to-air missile of 9K38 IGLA MANPADS (in order to diminish heating of infrared homing seeker fairing and reduce wave drag), giving the name to the whole system. Simplified Igla-1 version with a different kind of target seeker featured a tripod instead of 'needle' for the same purpose.

Further development of this concept has resulted in the "air-spike". This is formed by concentrated energy, either from an electric arc torch or a pulsed laser, projected forwards from the body, which produces a region of low density hot air ahead of the body. This has the advantage over a structural aerospike that the air density is lower than that behind a shock wave providing increased drag reduction.

2. Literature Survey

N. Sreekanth, J. Akhil and S. R. Nagaraja (2016), [1] investigates the heat flux and drag reduction techniques on a hypersonic re-entry vehicle. Many researchers have reported a reduction in these parameters by the introduction of an aero spike at the stagnation point of the blunt head of a re-entry vehicle. The effect of introducing a secondary surface (called the secondary spike), along with the aero spike, on the blunt head is discussed in this work. The study has been carried out by varying the location of the secondary spike with respect to the axis of the vehicle for a set of free stream conditions. Also, the effect of variation in free stream conditions on a re-entry vehicle with a primary and secondary spike has been investigated. It has been observed that some configurations work well for some free stream conditions but fail in comparison to a single spiked body for some other conditions. They have concluded, the effects of secondary spike on drag and heat flux reduction is studied and the results obtained show that secondary spike can be employed as means to reduce the drag and heat flux values. However, it is also observed that the location of the secondary spike and the free stream conditions are critical in determining the effectiveness of the secondary spike. Employing the secondary spike could result in detrimental effects, as observed in some simulations. A more extensive study investigating the effect of parameters like secondary spike length should also be done in order to get a clear picture of the impact of secondary spike on the flow field. The addition of secondary spike could result in structural stability issues, which should also be studied before it can be implemented. Also, employment of secondary spike along with other methods of heat flux and drag reduction techniques, such as opposed jet injection and energy deposition, can also be considered and studied.

Jiss J Sebastiana, Sandeep Eldho Jamesa, Abhilash Suryana (2016), [2] have studied the problem of drag and heating hypersonic vehicles when moving at very high speeds. One of the ways to reduce this drag and heating is by the use of an aerospike. In this study, the flow around a blunted body fitted with an aerospike is analyzed using a commercial software ANSYS Fluent and an open source Direct Simulation Monte Carlo (DSMC) code, called as dsmc Foam in Open FOAM, at a high Mach number (M=6) at different length to diameter ratios (L/D = 1.5, 2) at an angle of attack 0o. The aerospike placed in front of the body replaces the strong detached shock wave ahead of the body with a system of weaker oblique shock waves. A recirculation region is developed between the shock and the blunt body, which acts like a streamlined profile, thus reducing the drag and wall heat flux

Wan T and Liu CM (2017), [3] studied the Pressure drag and aero heating stirred by the shock wave is the main challenge of hypersonic flight, and blunt body is always the principle configuration at hypersonic flow regime for heat distribution, but it would induce substantial drag. Therefore, both aerospikes and aerodisks can be efficiently utilized as a mean for drag reduction. In this work, we investigate the effect of different geometric shapes of aerospikes with various disk gap widths on drag reduction. Accordingly, a series of numerical simulation work was implemented to find the behavior as to hypersonic flow over aerospiked blunt bodies. Moreover, the drag reduction efficiency of spiked blunt bodies would be optimized via the Kriging method. For the models we studied, we found that the drag on the spiked blunt bodies is much lower than the spike off one. The drag reduction efficiency especially would be predominated by the scale of recirculation zone, which increases as both the spike length and the gap size of aerodisk increase. Hence, the performance of drag diminution will depend on the design parameters such as main body configurations, aerospike length, and tip geometric shapes. From the analysis they have found the flow patterns including strong bow shock waves, expansion fans, shear layers, flow separation, recirculation regions, compression wave, etc. The bow shock (i.e., conical shock or foreshock) emits from the aerospike and covers the whole hemispherical body. The separation point along the aerospike will influence the pressure and temperature distributions along the main body surface, which counts on the turbulence model and related parameter settings. In terms of the drag components, the pressure drag is the predominant role in both the aerodisk and the blunt main body; however, the viscous drag would be the minor part in the total drag of all spiked blunt bodies. For models with same spike length, then the broader the gap width, the less the total drag. They found that the maximum drag reduction for same spike length would be found at widest gap. Thereby, the drag reduction performance attained by both the aerospike and aerodisk would rely on the trade-off between the spike length and the gap width.

Wei Huang (2015), [4] studied the drag reduction and thermal protection for hypersonic vehicles. A counter flowing jet and its combinations is one of the most promising drag and heat release reduction strategies. In the current survey, research progress on the drag and heat release reduction induced by a counter flowing jet and its combinations is summarized. Three combinatorial configurations are considered, namely the combination of the counter flowing jet and a forward-facing cavity, the combination of the counter flowing jet and an aerospike, and the combination of the counter flowing jet and energy deposition.

Deng F, Jiao Z, Liang B et al (2017), [5] High lift-to-drag ratio is considered crucial for high altitude and long endurance hypersonic vehicles. One of the simplest and most useful methods is to install an aerospike in front of the vehicle's nose. In this paper, the flight aerodynamic characteristics are investigated by simulating and comparing the lifting-body with or without the aerospikes at Ma=8. The flow fields around aerospikes using different spike lengths and a hemispherical disk along with the lifting-body are analysed. The results of aerodynamic characteristics indicate that L/D=2 is the best ratio of spike length to nose diameter. By comparing with the baseline model, maximum drag reduction of the nose's part is 49.3% at α=8º using hemispherical disk. In addition, three shapes of aerospike disks are compared to search for a best disk for hypersonic drag reduction. The best drag reduction is found for the double flat faced disk aerospike, which gives a pressure drag reduction of 60.5% of nose's part at α=8º. Furthermore, when the flight angle of attack increases, the drag increases significantly. Employing certain installation angle is shown to effectively improve the drag reduction around the angle of attack and results in improving the lift-to-drag ratio.

They investigated the performance of disk spikes in the context of hypersonic lifting body aerodynamics at a range of incidences. Hypersonic flow over a spiked lifting-body at Mach number 8 with different L/D ratios and disk shapes at flying angles of attack are numerically simulated. The major findings are as follows: The area of low pressure region depends on the aerospike length to nose diameter ratio which is the main factor for drag reduction. A significant drag reduction with the application of hemispheric disk aerospike at the nose can be achieved. An aerospike with L/D = 2 gives best reduction when comparing with other L/D ratios at α=8º, which leads to a maximum drag reduction of 49.3% and 4.39% for nose and a whole vehicle, respectively.

Comparison of hemispheric, single flat faced and double flat faced disks were made. The low pressure region at nose with double flat faced aerospike is larger than that with single flat faced aerospike at different angles of attack. For the vehicle nose, drag reduction caused by double flat faced aerospike is the best, having 60.5% pressure drag reduction of nose's part at flight α=8º. By installing the aerospike with a proper angle, the drag can be effectively reduced at the cruise angle. When the angle of attack is changed around the cruise angle, the drag remains at the similar value. This phenomenon means the drag reduction is stable and efficient using an aerospike with an installation angle at the gliding phase. The drag reduction of the whole vehicle is 8.7% at α=8º, using double flat faced aerospike. The lift-to-drag ratio is 3.634, which is 9.1% better than the baseline model.

The present work considers the calculation of drag force generated with base cone model with blunt shape and later the drag force generated in presence of spikes in hemispherical and triangular form and its reduction in comparison with base model with a mach number of 2.75 at 0° angle of attack.

3. Modeling and Analysis

The base model for conducting the flow analysis is obtained from the brahmaos missile. The dimensions of the missile are considered for creating the base model, with hemispherical spikes and triangular spikes. The 3D model of the missile with spikes and without spikes are created using CATIAV5 software. The 2D view of the missile considered as base model and with hemispherical and triangular spikes are represented in Fig. 3.

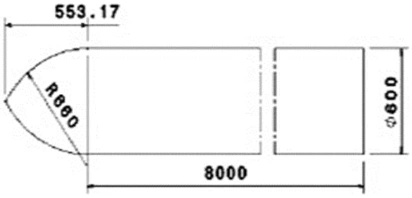

Fig. 3. Base model

The base model has a base diameter of 600mm The aerospike has a total length of 550mm with 30mm diameter with a projection of hemispherical shape with a radius of 30mm in case of hemispherical spike and in case of the length and diameter of the aero spike is similar to hemispherical spike and it consists a triangle at the front with 30mm x 60mm dimension. The Fig. 4 shows the base model with hemispherical spike. The 3D models of Base cone blunt shape, with hemispherical spike and triangular spike has been shown in Fig. 6, 7 and 8. The different models have been fabricated for representation is shown in Fig. 9.

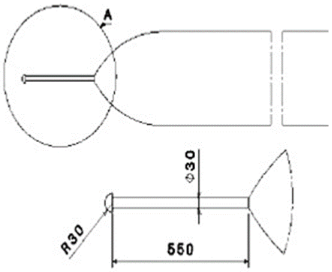

Fig. 4. Model with hemispherical spike

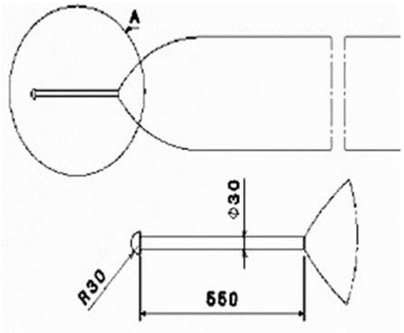

Fig. 5. Model with triangular spike

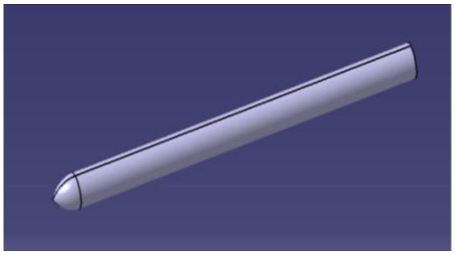

Fig. 6. 3D base model with blunt cone shape

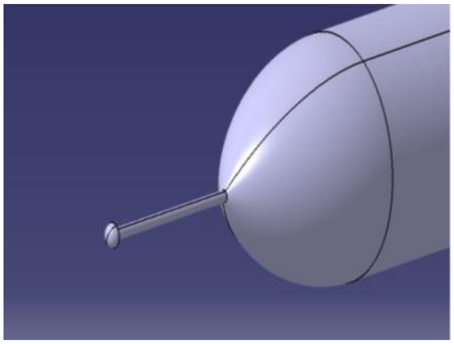

Fig. 7. 3D Base Model with hemispherical aerospike

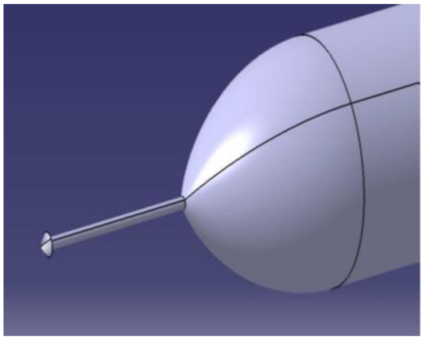

Fig. 8. 3D base model with triangular aerospike

Fig. 9. Fabricated models for representation

4. Numerical Simulation

The created models are further considered for analyzing the drag force created through flow over the base model, with hemispherical and triangular spike. The models are analyzed as axisymmetric models as they possess symmetry with respect to the axis. The flow analysis of the models is done by using ANSYS Workbench 15.0. As the accuracy of the any CFD solution depends on the quality of the grid used, we have checked the quality of the grid using two parameters via, Skewness cells, which have skewness in the region of 0.3 to 0.85. For structured and in the region of 0.3 to 0.9. For unstructured, are considered as good quality cells. For the we have generated the maximum equi-angular skewness we have found to be 0.91, which indicated that the grid is of a good quality. Aspect ratio for a rectangular face is the length to breadth ratio of it. For a good quality cell it ranges from 1to10. For the grid we generated the maximum aspect ratio is 9.5.

In the present study three different types of boundaries are applied: inflow, outflow and fixed walls. The flow fields under consideration here are supersonic. According to the theory of characteristics all variables are prescribed at in flow boundaries, i.e. Dirichlet boundary conditions, and Neumann boundary conditions are used for all variables at outflow boundaries. At fixed walls the no slip condition are applied. All computations are initialized with the state of the incoming air. Here some parameters and its magnitude for analyzing missile body is given below. There is an advantage of employing the complete Navier-Stokes equations as it extends not only to the investigations that can be carried out on a wide range of flight conditions and geometries, but also in the process the location of shock wave as well as the physical characteristics of the shock layer, can be precisely and near accurately determined. The flow analysis was performed with inlet mach number of 2.75 and the temperature and pressure were considered as standard atmospheric condition. The flow considered being steady state and the air considered to be compressible flow, obeying ideal gas law for density variation. The Fig. 10(a), (b) and (c) shows the Meshed models of base cone with blunt shape, hemispherical spike and triangular spike.

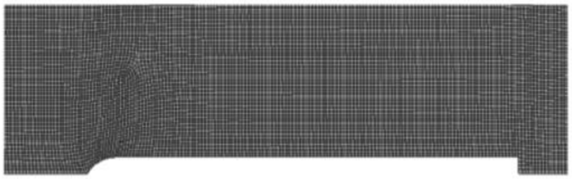

Fig. 10. (a) Meshed view of full flow domain mesh

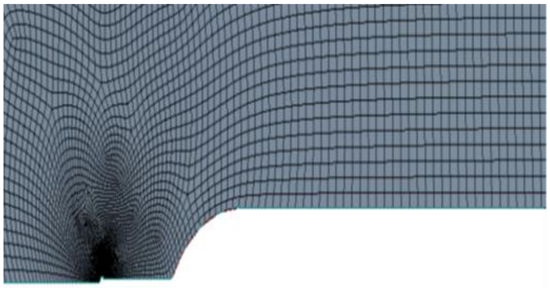

Fig. 10. (b) Meshed view of hemispheric flow domain mesh

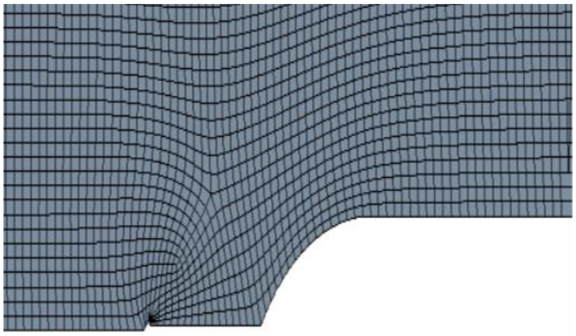

Fig. 10. (c) Meshed view of triangular spike flow domain mesh

5. Results and Discussion

The flow analysis of supersonic flow over missile body with blunt shape, with hemispherical spike and with triangular spike have been done and the results for the Pressure, Temperature and Mach number have been obtained through analysis. The flow conditions considered are tabulated in Table 1.

1) *Pressure Contour*

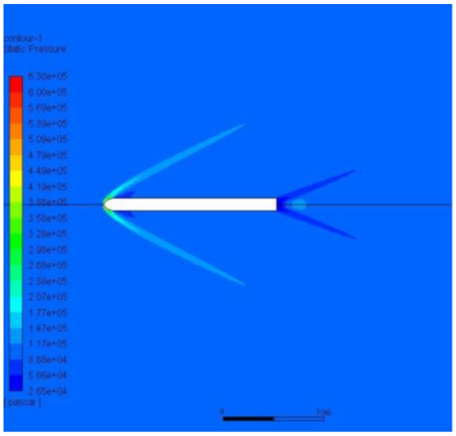

Fig. 11. (a) Pressure contour over base model

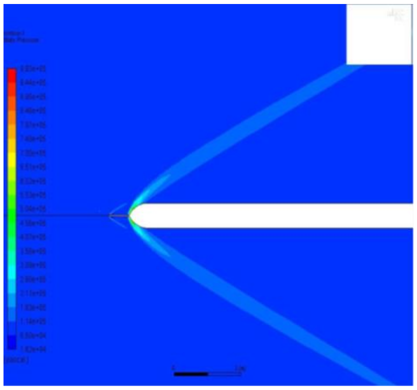

Fig. 11. (b) Pressure contour over hemispherical spike model

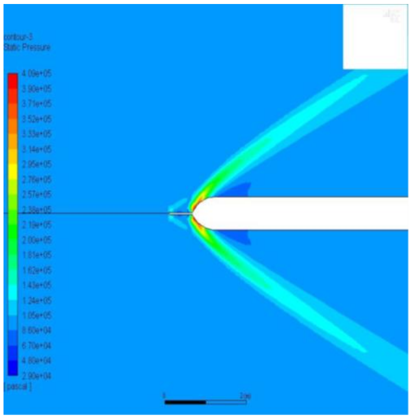

Fig. 11. (c) Pressure contour over triangular spike model

Table 1
Parameters considered for flow analysis

Parameter	Value / Characteristic	SI Unit
Type of Stream	Free	-
Mach Number	2.75	-
Angle of Attack	0.0	Deg

2) *Temperature Contour*

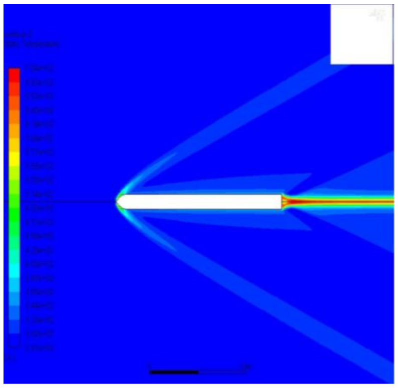

Fig. 12. (a) Temperature contour over base model

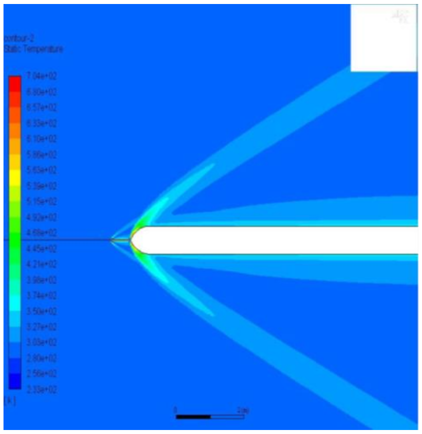

Fig. 12. (b) Temperature contour over hemispherical spike model

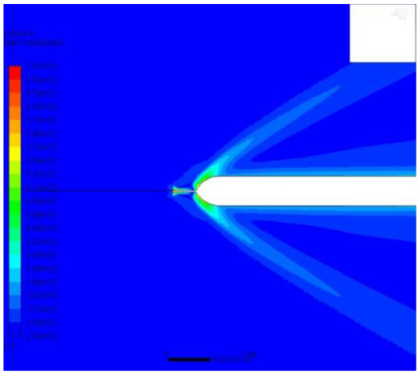

Fig. 12. (c) Temperature Contour over Triangular Spike Model

3) *Mach Number Contour*

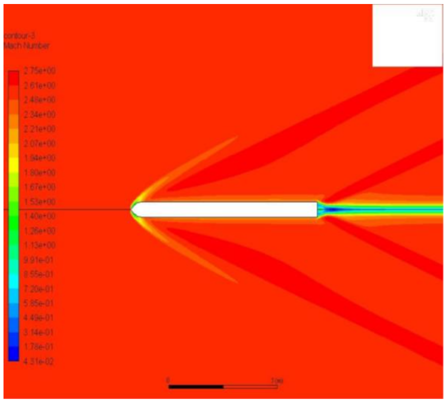

Fig. 13. (a) Mach number contour over base model

Table 2
Results of Static Pressure, Static Temperature and Mach Number for different models

Model	STATC PRESSURE		STATIC TEMPERATURE		MACH NUMBER	
	Pascals		K		Nil	
	Min	Max	Min	Max	Min	Max
Base Model	2.65E+04	6.30E+05	2.61E+02	7.04E+02	4.31E-02	2.75E+00
Hemispherical Spike	1.62E+04	9.93E+05	2.33E+02	7.04E+02	3.75E-02	3.62E+00
Triangluar Spike	2.90E+04	4.09E+05	2.79E+02	7.01E+02	4.22E-02	2.77E+00

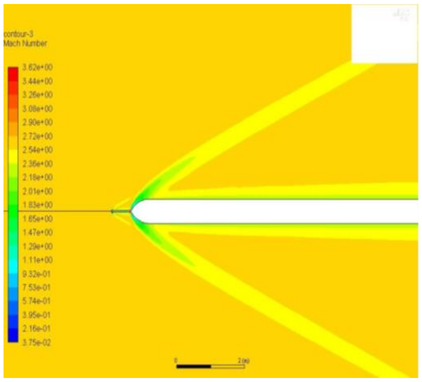

Fig. 13. (b) Mach number contour over hemispherical spike model

The drag force created over various models have been calculated and it is found to be that the drag force generated over the blunt cone base model is found to be 94427.26 N and for model with hemispherical spike the drag force generated is 90463.41 N and for triangular spike model 91134.81 N force is generated. With reference to the base model hemispherical spike has shown 4.1% of drag force reduction and triangular spike has shown 3.6% of drag force reduction. The Table 3.0 shows the drag force generated for various models and its corresponding reduction with respect to base model with blunt cone shape.

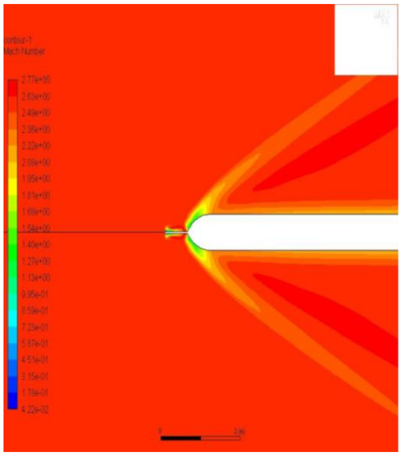

Fig. 13. (c) Mach number contour over triangular spike model

Table 3
Drag force values of different models

Model	Drag force	Reduction
	N	%
Base Model	94427.26	
Hemispherical Spike	90463.41	4.1
Triangular Spike	91134.81	3.6

6. Conclusion

The following points may be considered for arriving the conclusion of the present work.

- The drag force created for the blunt nose base model is higher than the models with spikes.
- The presence of spikes has reduced the drag force created in comparison to the base model considered.
- The hemispherical spike provided in the base model has reduced the drag force by 4% and Triangular spike has shown a reduction of 3.6% of drag force in comparison to base model.
- The dimensions adopted for both the hemispherical and triangular model may be varied to study the effect over the drag force creation and its reduction with respect to the blunt cone shape model.
- The contours of Temperature, Pressure and Mach number have been generated to have a visual understanding about the flow over the missile considered for flow analysis.
- The angle of attack considered in the present study is 0 degree and it may be varied to different levels.
- The presence of spike, its length and its shape has a remarkable influence over the drag force created and different geometrical configurations may be considered for further study.

References

[1] N. Sreekanth, J. Akhil and S. R. Nagaraja, Design and Analysis of Secondary Spike on Blunt Head, Indian Journal of Science and Technology, 2016, Vol 9(45).
[2] Jiss J Sebastian, Sandeep Eldho James, Abhilash Suryan, Computational study of hypersonic flow past spiked blunt body using RANS and DSMC method, Procedia Technology 25 (2016) 892 – 899.
[3] Wan T, Liu CM, Drag Reduction Optimization for Hypersonic Blunt Body with Aerospikes, Journal of Aeronautics & Aerospace Engineering, 2017.
[4] Wei Huang, “A survey of drag and heat reduction in supersonic flows by a counter flowing jet and its combinations,” Journal of Zhejiang University-SCIENCE A (Applied Physics & Engineering), 2015, 551-561
[5] Fan Deng, Zihan Jiao, Bingbing Liang, Spike Effects on Drag Reduction for Hypersonic Lifting-body, Journal of Spacecraft and Rockets, 54 (6). pp. 1185-1195.
[6] Mohamad Kazemi Esfeh, Seyed Mohamad Tajalli, Pengfei Liu, Evaluation of aerospike for drag reduction on a blunt nose using experimental and numerical modeling, Acta Astronautica 160 (2019) 656–671.
[7] S. Manigandan, K. N. Sruthisree, Kaushali Aich, P. Gunasekar, S. Nithya, Devipriya, Venkatesh, Numerical analysis of slotted aerospike for drag reduction, International Conference on Recent Advances in Fluid and Thermal Sciences, (2019).
[8] Yadu Krishnan S, Hariprasad Thimmegowda, “A numerical investigation to determine and compare drag Characteristics of a blunt body with various configurations of Aerospikes at mach 6, 21st Annual CFD Symposium, 2019, Bangalore.
[9] Guilai Han, Zonglin Jiang, Hypersonic Flow Field Reconfiguration and Drag Reduction of Blunt Body with Spikes and Sideward Jets, International Journal of Aerospace Engineering, Volume 2018,
[10] Parag P. Mangave, Vivek P. Warade, A. P. Shah, Numerical Analysis of Drag On Blunt Bodies with The Use of Different Conical Spikes at Supersonic Speed, 2018, Proceedings of 4th RIT Post Graduates Conference (RIT PG Con-18).
[11] Naveen Kumar K, Gopalsamy M, Daniel Antony, Krishnaraj R and Chaparala B. V. Viswanadh, Design and Optimization of Aerospike nozzle using CFD, IOP Conf. Series: Materials Science and Engineering 247, 2017.
[12] T. Karthikeyan, S. K. Aravindhkumar, J. Arun Kumar, Design and Analysis of Aerospike Nozzle to Improve Thrust in Hybrid Rocket Engine, International Journal of Engineering Trends and Technology, Volume 36, Number 7, June 2016.

[13] Mohammed Imran, Introduction to Aerospike and its Aerodynamic Features, International Journal of Scientific and Research Publications, Volume 6, Issue 5, May 2016.

[14] M. Elsamanoudy, A. Ghorab, M. Hendy Drag Reduction Using Spiked-Aerodisk & Reattachment Ring for Hypersonic Hemispherical Bodies, 15th International Conference on Aerospace Sciences & Aviation Technology, 2013.

[15] R. Kalimuthu, R. C. Mehta, E. Rathakrishnan, Experimental investigation on spiked body in hypersonic flow, Aeronautical Journal, 2008.