COMPUTATIONAL STUDY OF AERODYNAMIC CHARACTERISTICS OF A PROJECTILE BY VARYING BOAT TAIL CONFIGURATION

Aishwarya. E [1], Akhila. T. J [1], Amrutha. S. R [1], Sindhu. R. P [1],
Karthik Sundarraj [2], Prof. Prakash. S. Kulkarni [3], Dr. Manoj Veetil [4], Ganesh Pawar. R [5]

[1] Students, Aerospace Dept., IIAEM-JU, [2] Technical Manager – CFD Solutions, [3] Professor, Aerospace Dept., IISc, [4] Professor, Aerospace Dept. IIAEM-JU, [5] Research Associate, Aerospace Department, IISc

Abstract

In bluff bodies such as projectile and missile due to separated flow over its surface creating vacuum which results in increasing drag. Hence it is substantial to study and reduce drag since it involves the performance and other characteristics of projectile. To focus on this, many literature studies attempted in suggesting different designs or modifications on afterbody of a projectile which eventually decreases drag.

To get a better understanding on the effect of drag and other parameters on the performance of projectiles, the optimum M549 155mm projectile is modified and analysed for multiple Mach numbers in the present study. The main objective is to study how aerodynamic constraints change for different geometrical modifications and suggesting an optimized shape which yields better results compared to its counterparts. Among different modifications, rear cavity with thickness 15.5mm showed considerable reduction in drag compared to the standard M549 projectile.

Regarding the tail shape, there are different concepts that can be used to reduce the base drag such as the use of base bleed, splitter plates, boat tail afterbody was selected since it provides superior base drag reduction. While the boat tail afterbody is used in combination with other shape optimizations

Keywords: Rear cavity, Base drag, Vortex bursting, Bluff body, Vortex generator, coefficient of drag, Protuberances.

Nomenclature:

Cd – Coefficient of drag.
AOA – Angle of attack.
D – Diameter of the body.

Introduction:

The basic aerodynamic constraints which influence any object in flight are lift and drag. Hence there is a need to have a check on it to afford efficient flight. Bluff bodies such as launch vehicles, missiles, projectiles cause large amount of base drag due to the low-pressure wake zone at the rear end of the vehicles therefore increasing the total drag. An effort to reduce drag has always been a concern to ensure better performance. Many other parameters such as Mach number, profile of the body also has considerable effect on drag.

For predicting the drag coefficients many researches have been undergone, W. Jiajan [1] carried out experimental and computational work on drag prediction and the stability of the projectile.

P. R. Viswanath [2] suggested some techniques such as after body attachments, base bleed, ventilated cavities which highlights locking of vortices and thus reducing drag. From the related researches it motivated for prediction of aerodynamic characteristics over the M549 155mm projectile.

The idea of breaking down vortices which is called vortex bursting is implemented by making use of various protuberances along the body.

Vortex bursting prevents shedding of vortexes thereby providing an acceptable amount of drag reduction. This is obtained by interfering the vortex-shedding process, for instance by avoiding the occurrence of the separation of the boundary layer along a straight line. Thus, surface vortex generators are accompanied on the upper surface of the projectile.

Methodology:

In the field of computational fluid mechanics, the key elements that rule the flow physics are the Navier Stokes equations and the energy equations. So, in order to analyse the flow over a body the CFD Tool must fulfil two important requirements, first one being the mathematical model for the given flow problem and the numerical methods. For mathematical model, the CFD tool makes use of Navier Stokes equations which act as governing equations, which are used to solve the flow problem. The Navier stokes equations include the three Conservation laws including the conservation of mass, momentum and the energy equations. Numerical methods include the finite element method, finite difference and the finite volume method.

When a flow is turbulent, we make use of turbulent models which predicts the effects of turbulence in a given problem. These turbulent models act as mathematical models. In our present work, the turbulent model that we utilized is the K-ω SSTmodel. Basically, it is a two-equation model. Since our research mainly deals with flow separation and boundary layer problems, K-ω SST model is well known for its good behaviour in adverse pressure gradients and separating flow problems.

Validation:

To ensure correctness of the results obtained for optimized design,[1] is validated considering CFD simulations for four different Mach numbers.

- MACH 1.3
- MACH 1.5
- MACH 1.8
- MACH 2.0

Results are found identical to the reference with less than 1.28% error.

Y+ value is calculated for the corresponding Mach number.

Sl no.	Mach no.	Y+
1.	1.3	0.00000057
2.	2	0.00000044
3.	4	0.00000067

TABLE 1

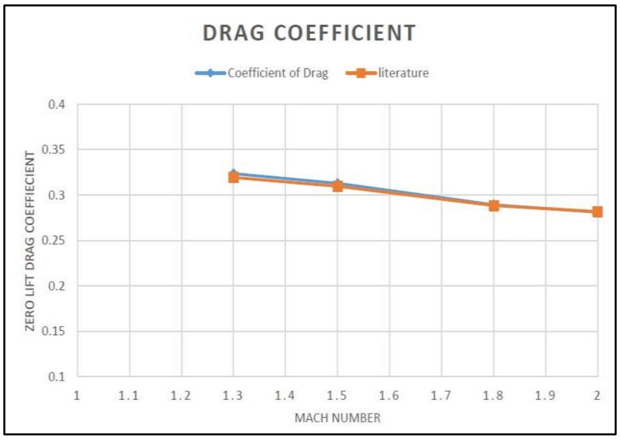

FIGURE 1: VALIDATION

Mach no.	Simulated Cd	Reference Cd
1.3	0.32355	0.319453
1.5	0.31258	0.309807
1.8	0.28921	0.288585
2.0	0.28108	0.281833

TABLE 2

Modelling:

As far as validation is concerned, initial design is considered [1] for simulation to obtain simulated Cd.

The model and all other augmented geometry are designed using ANSYS SPACECLAIM. Initial design is shown in the diagram below and the total length was maintained to be 155mm.

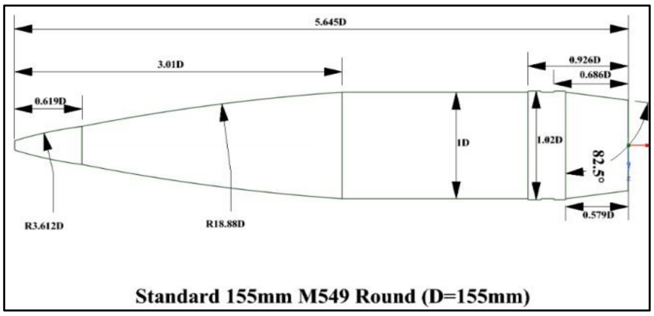

FIGURE 2: STANDARD M549 MODEL

The geometry considered was axisymmetric

- To reduce computation time
- And also, to reduce the node count and mesh size.

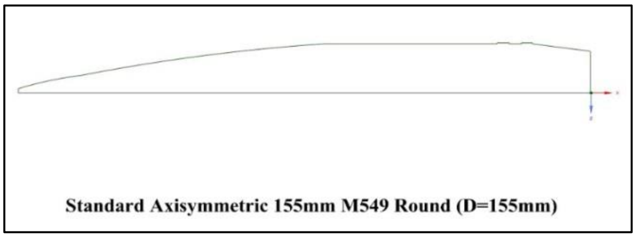

FIGURE 3: AXISYMMETRIC M549 MODEL

The domain that envelops the projectile ensures to encapsulate the formation of shocks at the forebody and shedding of the vortex at the rear. Initially the domain used in [1] was considered but reversed flow was observed. Hence a larger domain forming quarter circle maintaining 3D and 6D before and after the body was well considered. (D = 155mm base diameter of the projectile)

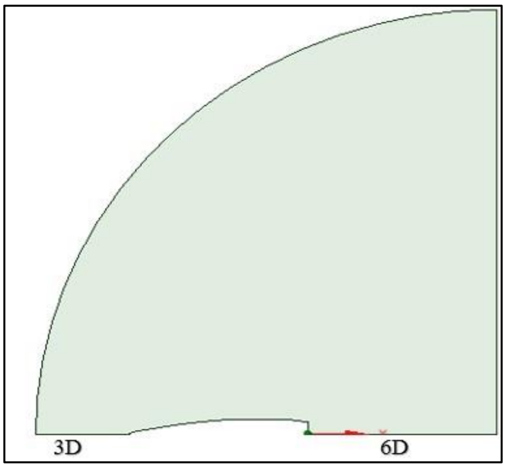

FIGURE 4: DOMAIN

Optimized Design:

- SHARP SERRATED VORTEX GENERATORS

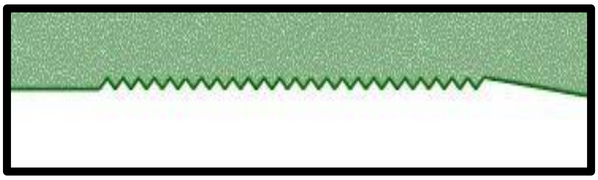

FIGURE 5: SHARP SERRATED VORTEX GENERATORS

The total length of the serrated region is 53.785mm, and the length of each edge is 1.55mm. the angle between the horizontal and the inclined line was maintained to be 135 degrees whereas the angle between the two inclined lines is 90 degrees. The detailed view of the sharp serrated vortex generators is shown below:

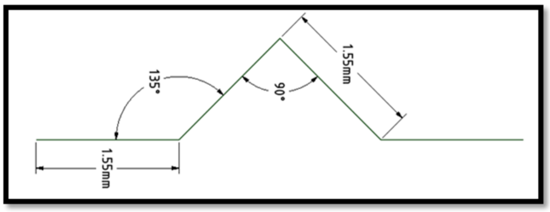

FIGURE 6: MEASUREMENTS OF SHARP TEETH VORTEX GENERATORS

- TRAPEZIUM SERRATED VORTEX GENERATORS

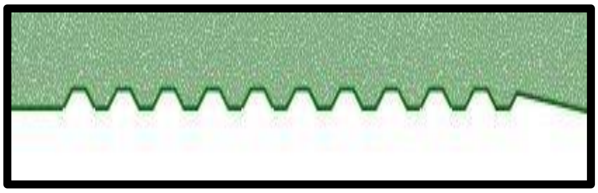

FIGURE 7: TRAPEZIUM SERRATED VORTEX GENERATORS

The total length of the serrated region is 53.785mm, and the length of each edge is 1.55mm and the angle between the edges is135 degrees. The detail view of the trapezium shaped vortex generators is shown below:

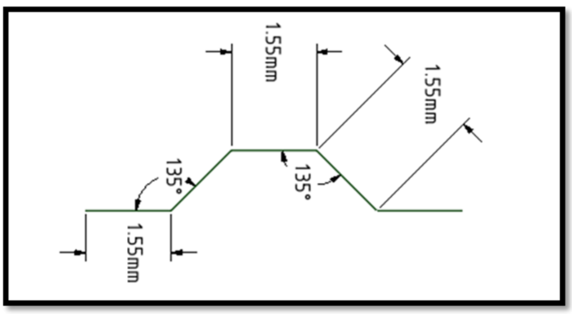

FIGURE 8: MEASUREMENTS OF TRAPEZIUM SHAPED VORTEX GENERATORS

- SHARP SERRATED BOAT TAIL EDGE

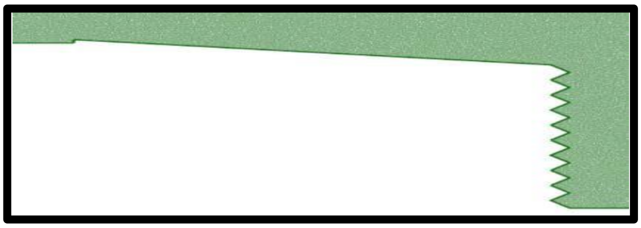

FIGURE 9: SHARP SERRATED BOAT TAIL EDGE

The total height of the serrated region was maintained to be 77.5mm and the length of each serration was taken to be 5mm. The angle between each serration is 90 degrees. The detailed view of the serrated boat tail edge is shown below:

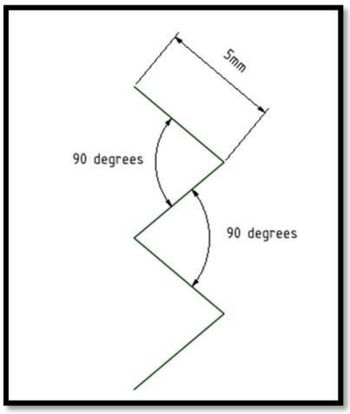

FIGURE 10: MEASUREMENTS OF SHARP SERRATED BOAT TAIL EDGE

- BLUNT SERRATED BOAT TAIL EDGE

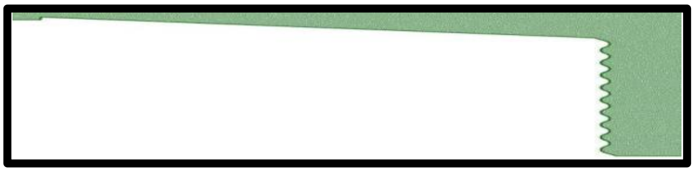

FIGURE 11: BLUNT SERRATED BOAT TAIL EDGE

The total height of the blunt serration is 77.5mm and the radius of each blunt serration was maintained to be 2.25mm. The detailed view is shown below:

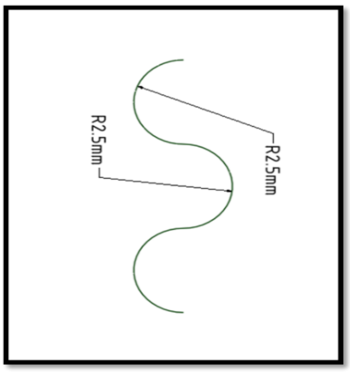

FIGURE 12: MEASUREMENTS OF BLUNT SERRATED BOAT TAIL EDGE

- REAR CAVITY

Two different configurations were selected for the study.

- REAR CAVITY OF THICKNESS 15.5mm

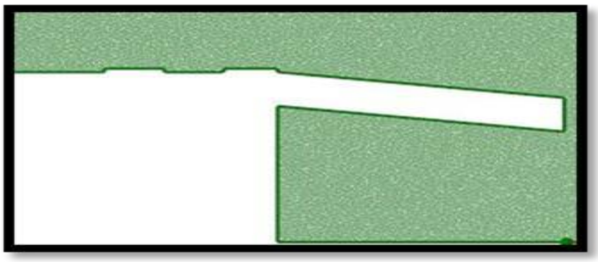

FIGURE 13: REAR CAVITY 15.5mm THICKNESS

- REAR CAVITY OF THICKNESS 23.25mm

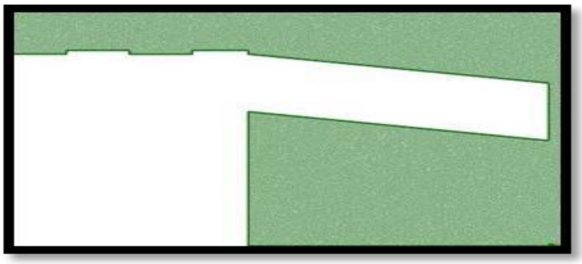

FIGURE 14: REAR CAVITY 23.25mm THICKNESS

The total length of the projectile is 5.645D (where D is 155mm).

The diameter (H) of the projectile is known i.e. 155mm.

Three ratios (h) were chosen 0.1, 0.2, and 0.3. The thickness of each rear cavity is calculated as follows:

Ratio (H/h)	H (in mm)	h (in mm)	h/2 (thickness of the cavity)
0.2	155	31	15.5
0.3	155	46.5	23.25

TABLE 3

Meshing:

Since there was a requirement for accurate result, it was better to proceed with structured mesh using ICEM CFD maintaining node count of 1M for all the configurations. Reason to consider 1M nodes for all the configurations is because the coefficient of drag from the experiment obtained from [1] is closer to the coefficient of drag attained using CFD. Thus, yielding better precision.

The quality of the mesh is 94.8%. Since ANSYS FLUENT supports unstructured mesh, conversion was performed without errors. The aspect ratio was between 25-54 and the growth ratio was 1.2.

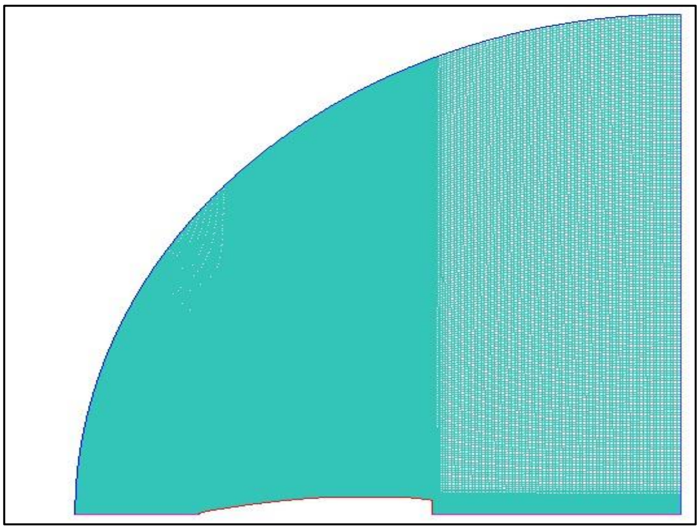

FIGURE 15: MESH

For any wall bounded turbulent flows, the boundary layer region must be captured very accurately. In order to do so, we use a nondimensional distance from the wall of the body to its first node and this distance is defined as y+. For our research work, for respective Mach number the y+ value was calculated and the inflation layer was accordingly modified.

Solver:

Analysis of the modified geometries and numerical simulations are carried out by using a commercial solver, ANSYS FLUENT 16. To reduce the computational time and to get good results, 2D Axisymmetric body is used. Then the results are compared with other modified geometry as well as the standard one by plotting graphs.

Since the analysis and simulations are carried out at supersonic speeds, density-based axisymmetric and steady flow solvers is used.

Implicit type of solution method and AUSM flux type is set and second order accuracy is achieved by using green gauss node-based solver.

The body is stationary having no -slip boundary condition at its walls.

Convergence:

Solution convergence is monitored using residuals and ensuring Cd, Cl values to remain stagnant.

Solution steering is utilized to modify the courant number. Additionally, to accelerate convergence rate FMG initialization is applied.

Results:

Design	Cd for Mach 2	Cd for Mach 4
Sharp serrated vortex generators.	0.36136	0.19755
Trapezium serrated vortex generators.	0.32587	0.19497
Sharp serrated boat tail edge	0.28563	0.16359
Blunt serrated boat tail edge	0.29123	0.19499
Rear cavity 1 (15.50 mm thickness)	0.21290	0.16466
Rear cavity 2 (23.50 mm thickness)	0.29469	Simulation in progress

TABLE 4

The optimized geometries were run for Mach 2 and 4.

Coefficient of drag for the rear cavity simulated for Mach 2 is found to be the best optimized configuration as the value is 0.21290.

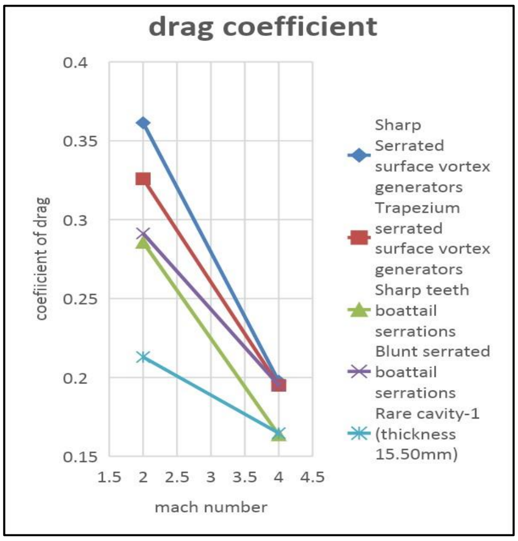

FIGURE 16: GRAPH OF DRAG COEFFICIENT V/S MACH NUMBER

Conclusion and Future Work:

Validation [1] of the model was done and the results were obtained were close to the experimental values. Amongst the optimised designs rear cavity with 15.5mm thickness gave lowest drag.

Simulations of standard M549 155mm projectile with a multi-step base configuration and rear cavity configuration with different shape are to be carried out. Also, activities with different modelled configurations at different angle of attack are under progress. Analysis for the swirl motion of the projectile will also be considered in the future. Combinations of different modifications and splitter plates are also on the track.

References:

1. W. Jiajan, R.S.M. Chue, T. Nguyen, S. Yu, Optimization of round bodies for aerodynamic performance and stability at supersonic speeds, Aeronaut. J. 177 (1193) (2013) 661–685 W. Jiajan.

2. ViswAnAth,P.R Flow management techniques for base and afterbody drag reduction, Prog Aerospace Sci, 1996, 32, pp 79-129.

3. Sor Wei Lun, "Aerodynamic Validation of Emerging Projectile Configurations," Naval Postgraduate School Monterey, CA 93943-5000, December 2011.

4. W. Jiajan, Aerodynamic characteristics of high performance rounds at Mach 1.8 to 4.

5.G.K Suryanarayana, Bluff-body drag reduction by passive ventilation.

6. Byregowda. G, Base Drag Considerations and Projectile Optimization on 0.5-Caliber Projectile.

7. M.Lorite-Díez, Drag reduction of slender blunt-based bodies using optimized rear cavities.

8. Kline, r., herrmAnn, w. r. and osKAy, V. A determination of the aerodynamic coefficients of the 155mm, M549 Projectile, Technical

9. Report No. 4764, Picatinny Arsenal, Dover, New Jersey, USA, BD 002730L, November 1974

10. Viswanath~ P. R. and Narasimha, R. (1974) Two-dimensional boat-tailed bases in supersonic flow, Aero. Quarterly XXV, 210-224. [10] Mair, W.
A. (1969) Reduction of base drag by boat-tailed afterbodies in low speed flow, Aero. Quarterly 99, 307-320.

11. P.R Viswanath, Drag reduction of afterbodies by controlled separated flows, AIAA J. 39 (1) (2001) 73–78,

12. Sidra I. Silton, Navier-Stokes Computations for a Spinning Projectile From Subsonic to Supersonic Speeds

13. Karthik Sundararaj, Computational study on base body drag reduction using Locked Vortex Flow Management Technique by attaching Double splitter plates at the base

14. Gary O Wheeler, Low Drag Vortex Generators, 2715 185th Ave. E., Sumner, Wash. 98390

15. Nose cone and tail structures for an air vehicle, Clifford T. Calfee and Virgil T. Calfee, both of 3294 S. Polk St, Dallas, Tex, Filed Jan. 30, 1962, Ser. No. 173,856 8 Claims

16. Boattail plates with non-rectangular geometries for reducing aerodynamic base drag of a bluff body in ground effect, Jason M. Ortega, Kambizsabari

17. Bullet, Gregory J. Giannoni, 1816 Stoney Crest Dr. Elberton, GA (US) 30635

18. Projectile with reduced base drag, Robert J. Paterson, Simsbury, Michael J. Werle, West Hartford, both of Conn: Walter M.Prez, Jr., Wibraham

19. Surface modification apparatus and method for decreasing the drag or retarding forces created by fluids flowing across a moving surface, Hilbert F. P. Drews, 5640 S. 76th St., Greendale, Wis.53129