Prediction of Drag Coefficient of a Base Bleed Artillery Projectile at Supersonic Mach number

D Siva Krishna Reddy1*, Prashant Kumar Sah1 and Ashish Sharma1

1Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur 603203, Tamil Nadu, India

*E-mail*: sivakriv@srmist.edu.in

Abstract An artillery projectile in flight produces aerodynamic drag in base region known as base drag. By injecting hot gases into the base region at low speed, base drag can be reduced. The reduction in base drag significantly increases the range of the projectile. In the present work. Numerical simulations over 155 mm artillery projectiles are performed with and without base bleed. The free stream Mach number is 2.26. Simulations are performed with commercial CFD package, ANSYS-FLUENT. The numerical method is based on density-based algorithm and K-ω SST turbulence model was chosen for calculation of turbulent stresses. Computations are performed for Angle of Attack (AOA) for 0° and 10°. The predicted drag coefficient for AOA=0° is validated with experimental data. The hot gases are injected at a Mach number and pressure of 2.26 and 101325 respectively. With the base bleed, 14.4% reduction in drag coefficient for AOA=0° is observed. The effect of base bleed on flow field and surface pressure is analyzed and presented in the paper.

1. INTRODUCTION

Artillery rounds are often used as weapons in wars and conflicts. There has been a constant push to increase their range and explosive potential. One of the most important factors in increasing range is reducing the drag encountered by artillery shells in flight. Aerodynamic engineering culminated in the creation of a shell geometry with the least amount of drag [1]. Weapon systems' extended firing ranges and impact precisions are required to be continuously improved, especially as new ammunition is produced or existing ammunition is updated. Drag reduces the flight performance of aerodynamic bodies such as warheads, bombs, and rockets.

Projectile drag can be classified into three components: pressure drag (excluding the base), viscous (skin friction) drag, and base drag [2, 3]. The relative magnitude of the aerodynamic drag components are 20% pressure drag, 30% viscous drag, and 50% base drag [4]. Notably, of the three drag components impacting a projectile, the base drag often accounts for one-half, if not much more, of the overall drag for large caliber ammunition. Reducing the base drag is an effective and realistic way to minimize the overall projectile drag and increase the projectile range by as much as 30% [5]. For long-distance flights, where the integrated effect of drag reduction is manifested, the base bleed technology is particularly useful. Such a capability is of significant importance today. Detailed understanding of the energy, mass addition and fluid-dynamic interactions around and particularly base region flow is necessary before solutions can be proposed to reduce the drag.
The base bleed device uses the propellant’s combustion to provide the mass flow rate injected in the base region. By burning this propellant near the base area, the base pressure of a projectile moving at supersonic speed can be controlled. Significant changes in base pressure were also found when a heated bleed gas was used. The peak base pressure is higher, and it occurs at no mass flow rate, than in the base bleed cases. As a result, the current thesis focuses on base drag optimization of 155 mm artillery projectiles using base bleed technology. Several CFD calculations were used to evaluate the drag reduction estimate.

2. PROBLEM FORMULATION:

The objective of the present work is to perform 3D simulations over the 155mm M107 artillery projectile with base bleed at 0° and 10° angle of attack. Flow field predictions must also be done over the M107 projectile shell for supersonic Mach number 2.26. To analyse the flow field to determine the impact of AOA on the drag coefficient. Lastly, to examine the flow fields for supersonic Mach numbers as the angle of attack is varied. The selected 155mm M107 artillery projectile and its geometry is shown below in Figure 1 [6].

![Figure 1: (a) Isometric view and, (b) Geometry of the 155mm M107 Artillery Projectile.](image)

The geometry and grids for the projectile were generated using ANSYS ICEM-CFD software. Quadrilateral cells were used in domain. The projectile contained hexahedral cells. The total number of mesh equals 3604315 cells. In the near wall of projectile fine mesh is employed to capture the boundary layer effect properly and coarse grids are used towards the outlet computational domain. The width of first grid normal to the wall is 10^-4 mm. The Y+ value is maintained on the projectile body. The mesh sizing lies between 10^-7 m and 10^-3 m. The computational domain was extended to be 9 times diameter far from projectile base in downstream direction, 3 times diameter around the projectile, and 0.1 times diameter far from projectile nose in the upstream direction as shown in Figure 2.

3. SIMULATION METHODOLOGY:

The flow field over the projectile was simulated using ANSYS-Fluent. The simulations were run on structured grids created with the ICEM CFD meshing tool. A density-based solver was used to solve this case. The SST k-omega turbulence model was chosen for all cases as it was best turbulence model for external flow cases. Since we observed the flow at Mach 2.26 where the flow would be compressible, we had used the pressure far field boundary condition at the inlet. Hence, the properties of air were set with respect to ideal gas equation. In the outlet region, it was set to the pressure outlet condition to control the static pressure of the environment.

The reference area was the area of the artillery shell which was used to calculate aerodynamic coefficients. Explicit method was used as the solution method for all cases. All spatial discretization were set as 2nd order upwind methods. The boundary condition taken for the base bleed is mass flow inlet with 830K temperature. Injection parameter is defined as ratio of the mass flow rate of the bleed
jet to the product of the free stream density, free stream velocity and area of the base region of the projectile. The injection parameter is taken as 0.0113 which results in mass flow rate of 1.509 Kg/sec. The area of the base bleed region is 0.00358 m$^2$. The injection pressure is calculated as 34721 Pa as explained below.

Temperature ($T$) = 830 K, Mass flow rate ($m$) = 0.1509 Kg/m$^3$, Area ($A$) = 0.00358 m$^2$

Mach number ($M$) = 0.5 (assumed)

Figure 2: Representation of (a) Full Computational Domain, (b) Mesh of M107 155mm Projectile, and (c) Sectional plane at Z-axis of the Mesh domain.
\[ V = \sqrt{\gamma \cdot R \cdot T} \cdot \sqrt{M} = \sqrt{1.4 \cdot 287 \cdot 830} \cdot 0.5 = 288.74 \text{ m/s} \]

\[ m = \rho \cdot A \cdot V, \Rightarrow \rho = m / A \cdot V = 0.1509 / (0.00358 \cdot 288.74) = 0.1459 \text{ Kg/m}^3 \]

\[ P = \rho \cdot R \cdot T = 0.1459 \cdot 287 \cdot 830 = 34575 \text{ Pa} \]

The base bleed fluid is taken as air.

4. RESULTS AND DISCUSSION:

4.1 SIMULATION WITHOUT BASE BLEED

4.1.1 Contours of Mach number

The computed flow field in term of Mach number contour for free stream Mach number 2.26 is shown in Figure 3. The flow feature over the projectile shell is identified. Since the configuration of the projectile shell's nose region is blunt and the flow is supersonic, a bow shock wave forms, decelerating the flow. Due to the stagnation conditions in the nose stagnation field, the Mach number decreases to a subsonic value through the shock wave, and the flow undergoes isentropic compression. The shock wave layer that prevails away from the nose region has a supersonic Mach number. The flow accelerates as it passes over the expansion fan at the end of the ogive field, causing the flow's Mach number to rise above 2.26.

The boundary layer is applied to the artillery shell's surface and separates at the boat tail's tip. The higher velocity flow away from the projectile surface appears to collapse due to the sudden slippage of the boundary layer, resulting in the creation of a re-circulating bubble and the stagnation stage. The formation of a recirculating bubble causes a pressure loss, which increases the projectile's base drag. The re-circulation bubble has a subsonic flow. A shear surface separates the inner re-circulation bubble from the outside inviscid flow. A re-compression shock wave that starts in the near-wake turns the flow to free stream direction.

\[ \text{Figure 3: Mach number contours over the projectile shell for M 2.26 at AOA 0°} \]
Figure 4: Representation of base flow field for $M = 2.26$ without base bleed, (a) Mach contour, and (b) Vector flow field.

Figures 4. (a) and (b) show Mach number contours and velocity vectors in the base region. The vector plot shows the flow of air at base region of the projectile. The air follows the general path of flow along the end of the projectile body and changes course inwards as the pressure in the center of the flow of base region is low and the air fills up that area of low pressure region. As air moves from the high pressure region to the low pressure region the re-circulation bubbles forms which increase the drag and slows down the speed of the projectile.
4.1.2 Surface Pressure Distribution without base bleed

The pressure distribution over the projectile shell surface is shown in Figure 5. At start there is sudden decrease in pressure that can be observed in the nose region. The flow then starts to accelerate over the two secant ogive regions, where the pressure drops slightly over the length. The pressure increases slightly over the cylindrical region and then drops once again over the boat tail region. At base region the pressure is very low because of that drag coefficient is high which is 0.2783. Experimental drag coefficient is 0.2744. The error in the drag coefficient prediction is 1.42%.

4.2 SIMULATION WITH BASE BLEED

4.2.1 Contour of Mach number with base bleed and without base bleed

Mach contour of base region for M 2.26 with base bleed at zero angle of attack is shown. and with the base bleed the wake region pressure increases and drag coefficient decreases to 0.2381 which is 14.4% less than the projectile without base bleed. The Mach contour without and with base bleed is shown in Figure 4 and 6.

4.2.2 Comparison of Surface Pressure Distribution over the base region

The surface pressure distribution at zero angle of attack over the base region for the condition having without base bleed and with base bleed is compared in the below Figure 7. When fuel is injected into the base region, the pressure in the base region rises. Before injecting fuel, air travels over the projectile walls, and at the base, a recirculation bubble forms and the pressure drops, increasing drag. However, when fuel is injected at the base, the recirculation bubble dissolves and the pressure rises, reducing drag.
Figure 6: Representation of base flow field for M = 2.26 with base bleed, (a) Mach contour, and (b) Vector flow field.

Figure 7: Surface Pressure Distribution for M 2.26 for base bleed and without base bleed
4.3 SIMULATION WITH BASE BLEED AT AOA 10°

4.3.1 Mach number contour at AOA 10°

The Mach number contour with base bleed at AOA 10° over the projectile surface and at base region is shown in Figure 8. Asymmetry is induced in the flow as AOA is introduced. For the high angle of attack, the re-circulation bubble collapses towards one side. This is because of one side experiencing more flow and other side of shell experiencing less flow. From the flow it is clearly indicated that how the recirculation bubble shifts with increasing angle of attack. At AOA=10° the recirculation bubble is completely shifted in the downward direction. At AOA 10° with base bleed the drag experienced is 0.2788.

4.3.2 Surface pressure distribution at AOA 10°

When projectile gets fired AOA 10° less air gets in contact with the projectile wall because of which there is reduction in the surface pressure compared with projectile fired at AOA 0°. And at angle of attack 10°, again when it approaches the base region the pressure increases and again at the base region pressure gets reduces which is depicted from the Figure 9.
Figure 8: Representation of base flow field for M=2.26 at AOA=10° with base bleed, (a) Mach Contour over the projectile, (b) Mach contour at base and (c) Vector flow field

Figure 9: Surface pressure distribution over the projectile shell for α=0° and α=10°

5. CONCLUSION:

Supersonic flow field over M107 155 mm shell at Mach number of 2.27 is computed. The compressible NS equations are solved with a finite volume algorithm. Simulations are performed with ANSYS Fluent for AOA 0° and 10°. The flow field is analysed with the aid of Mach number and surface pressure distribution. The flow features such as bow shock, expansion around the nose, separation region and re-compression shock wave are captured very well. The relevant flow phenomena have been observed on the surface pressure distribution also. The predicted drag coefficient is deviated by 1.42% from the experimental value.

Next, flow field with base bleed is computed. The hot gases are injected at a temperature of 830K into the wake region. The injected gases alter the base flow field. The effect was analysed with the help of velocity vectors. The single recirculation bubble splits into multiple separation bubbles. The injected mass enhances local pressure in the recirculation region, and hence over all pressure on the base
increases. This reduces base drag and hence drag coefficient. At Mach number of 2.27, and AOA =0°, the reduction in the drag coefficient found to be 14.4%. Strong asymmetry is noticed in the base bleed solution at AOA 10. The predicted drag coefficient for AOA 10 with base bleed found to be 17.09% higher than that of AOA 0 with base bleed.

ACKNOWLEDGEMENT:
We would like to thank CAD Laboratory, Department of Mechanical Engineering, SRMIST, Kattankulathur for providing computational resources. We would also like to thank the SRM High Performance Computing Centre (HPCC) for providing the access to the computational facility.

REFERENCES:

1. Petri Kaurinkoski, Antti Hellsten, “Numerical Simulation of a supersonic base bleed projectile with improved turbulence modelling”, *Journal of Spacecraft and Rockets*, Vol. 35, No. 5, 1998.

2. Sahu, J. Supersonic flow over cylindrical after bodies with base bleed. *Computational Mechanics* Vol. 2, p 176–184, 1987.

3. Ashoke De, Piyush Chettri, “Drag Reduction with Optimum Designing of a Base Bleed Projectile Using Computational Analysis.” *Innovations in Sustainable Energy and Cleaner Environment, Green Energy and Technology* p 23-45, 2020.

4. Jan-Kaung Fu, Shen-Min Liang, “Drag Reduction for Turbulent Flow over a Projectile.” *Journal of Spacecraft and Rockets*, 31:1, pp.1-4, 1994.

5. Mohammed Amin DALI, Slobodan JARAMAZ, “Optimization of Artillery Projectiles Base Drag Reduction using hot base flow”, *THERMAL SCIENCE* Vol. 23, No. 1, pp. 353-364, 2019.

6. Robert L. McCoy, "Modern Exterior Ballistics", 4th edition, ISBN-0-7643-0720-7, Schiffir Publishing Ltd., 1998.

7. Lieske, R., “Determination of Aerodynamic Drag and Exterior Ballistic Trajectory Simulation for the 155mm, DPICM, M864 Base-Burn Projectile.” *Ballistic Research Laboratory Aberdeen Proving Ground, Maryland BRL-MR-3768*, 1989.

8. Habib Belaidouni, Saša Živković, “Numerical Simulations in Obtaining Drag Reduction for Projectile with Base Bleed” *Scientific Technical Review*, Vol.66, No.2, pp.36-42, 2016.

9. Kayser, L.D., Kuzan, J.D., & Vázquez, D.N. Flight Testing for a 155mm Base Burn Projectile. *Ballistic Research Laboratory Aberdeen Proving Ground, Maryland BRL-IMR-915*, 1988.

10. M.E. Wessam, Z.H. Chen, “Flow Field Investigations and Aerodynamic Characteristics of Artillery Projectile.” *International Conference on Electrical, Automation and Mechanical Engineering*, 2015.