Elangovan et al. 2021. *Int. J. Vehicle Structures & Systems*, 13(1), 28-31
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.13.1.07

International Journal of
Vehicle Structures & Systems
Available online at www.maftree.org/eja

Computational Analysis of Base Drag Reduction for a Subsonic Missile Projectile at Different Flow Velocity Conditions

R.R. Elangovan[a], K.R. Vijayakumar[b] and G. Ramanan[b,c]

[a]*Dept. of Mech. Engg., Dr. MGR Educational and Research Inst., Chennai, Tamil Nadu, India*
Corresponding Author, Email: elangovanrajagopalan@yahoo.com
[b]*Dept. of Aeronautical Engg., ACS College of Engg., Bangalore, Karnataka, India*
[c]*Email: gramanan1987@gmail.com*

ABSTRACT:

Base drag is arising from flow separation at blunt base of a body. It can be a sizeable fraction of total drag in context of projectiles, missiles and after bodies of fighter aircrafts. The base drag is the major contribution of total drag for low speed regimes. Flight tests have shown that the base drag may account for up to 50% of total drag. In this paper an experimental work on a simple semi-circular flight vehicle body of length 500mm and diameter 50mm was conducted for the purpose of investigating base drag. The base drags for three configurations are calculated and the results are compared with CFD data. The three configurations used for testing are flat base configuration, closed nozzle configuration and boat tail configuration. The evaluations of base drag for the flow velocities of 20m/s, 35m/s and 50m/s at angles of attack such as -2°, 0° and 2° are experimented and compared.

KEY WORDS:

Base drag; Pressure scanner; Angle of attack; Hemispherical flight vehicle

CITATION:

R.R. Elangovan, K.R. Vijayakumar and G. Ramanan. 2021. Computational Analysis of Base Drag Reduction for a Subsonic Missile Projectile at Different Flow Velocity Conditions, *Int. J. Vehicle Structures & Systems*, 13(1), 28-31. doi:10.4273/ijvss.13.1.07.

1. Introduction

The cross section geometry of a flight vehicle body strongly affects flow separation, reattachment point and vortex structure especially in subsonic and transonic regimes. As the edges become rounded off, the transition from turbulent to laminar flow separation occurs with the angle of incidence. The present study describes a computational analysis method to predict the aerodynamic characteristics of three geometry having circular cross sections, a hemispherical body with flat base, a flight body with a nozzle at the base and with boat tail configuration. Major purpose of the series of tests reported in the project is to investigate the characteristics of the base flow associated with a missile body at very low subsonic speeds and the part played by vortex street in the determination of the base pressure. Earlier studies on base drag reduction have been done on missiles at high subsonic and transonic speed regimes [1]. This investigation of base drag on a simple hemispherical missile body with three base configurations: baseline profile, a missile with a nozzle at base and same missile with boat tailing is aimed at estimating base drag co-efficient for each of these configurations and finding best suitable profile with the minimum base drag.

The reduction in base drag of flight vehicles cause considerably increases in their range and also enhances their performance. These flight vehicles play an important role in defence applications. Various studies have been done on bodies with blunt-edged base to investigate the drag acting on bodies. The base drag due to recirculation at the back affects the performance characteristics of various bodies such as projectiles, missiles, rockets, trucks etc [2]. In such bodies with blunt base various case studies for the estimation and reduction of drag acting on them have been performed by many, both numerically and experimentally. Mair [2] studied the base pressure distribution of a slender body of square cross-section to determine the effect of boat tail geometry. The data were obtained over an angle-of-attack range of 0 to 8°.

Viswanath et al [3] estimated the head, base and friction drag coefficient of a missile. This procedure pertains to rockets and artillery projectiles, with or without fins and is a combination of theory and empirical data, gathered from numerous sources. Menezes et al [4] improved empirical model for base drag prediction on missile configurations based on new wind tunnel data at Mach numbers from 2.0 to 4, 5, AoA to 16°, fin control deflections up to 20°, fin thickness to chord (t/c) ratio of 0.05 to 0.15 and fin locations flush with base to two chord lengths upstream of the base. Bak et al [5] demonstrated a reduction of base drag by boat-tailed after body consisting of smooth fairing leading to conical tail piece of semi angle 22° in low speed flow.

At higher Reynolds number, the drag reduction obtained with conventional tail might be greater and such a tail might then compare rather more favourably with boat tailed after bodies. Krieger et al [6] controlled drag reduction by base flow separation problems at high speeds and the drag associated with it are a significant source of observation in design of missiles, projectiles and other typical high-speed vehicles. A large wake at the base of vehicle would cause an increase in overall drag due to reduced base pressure. From literature study, an experimental technique is planned at very low subsonic speeds using base drag on three missile profiles which best suited for experimental modelling and fabrication. The pressure distribution on body and the base of each missile configuration are obtained and drag on the body due to base pressure is calculated for each of the three missile geometries.

2. Experimental details

One of the most important aerodynamic performance characteristics for flight vehicles is total drag. Base drag is a component of aerodynamic drag caused by a partial vacuum in flight vehicle's tail area. The vacuum is created by vehicle'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 motor fills this void. The drag takes a sharp jump at burnout when this gas disappears. The determination and minimization of base drag is essential in minimizing total drag of flight vehicles [7]. Base drag is influenced by a variety of flow and geometrical parameters. With turbulent boundary layer ahead of base and in absence of jet flow, the major factors include: Mach number in free stream just ahead of the base, boundary layer momentum thickness ahead of the base, base diameter and Angle of Attack (AoA). The conical section of a ballistic body, that progressively decreases in diameter towards the tail to reduce overall aerodynamic drag is referred to as boat tail body. The drag so produced by this body is known as boat tail drag.

Boat tail is the minimal drag when compared with that of base drag and nozzle shroud configuration drag. Vertical angles and ambient air density mostly affects the boat tail drag in missiles. A nozzle shroud includes a centre body that is at least partially surrounded by an outer shroud. The outer shroud is readily spaced from the centre body to define a pre-mixed flow passage between the outer shroud includes a main body that defines an inner side portion, an outer side portion and a forward end portion that is axially separated from and an aft end portion. At low-speed subsonic flow over a sphere or an infinite cylinder with its axis normal to the flow, if the flow was inviscid (frictionless), the streamlines would form a symmetric pattern. Hence, the pressure distribution over front and rear surfaces would also be symmetric. This symmetry creates a momentous phenomenon, namely, that there is no pressure drag on the sphere if the flow is frictionless [8]. The actual flow over a sphere or cylinder, in presence of friction leads to separated flows in regions of adverse pressure gradients.

There will be a separated flow on back face of cylinder, with a relatively flat wake and with the associated high pressure drag. The considered configurations are shown in Figs. 1-3. For the case of a blunt body, the drag is relatively large and most of the drag is due to pressure drag. The three missile geometries to be investigated for base drag are designed using CATIA V5 R20. All the three models considered are simple hemispherical bodies of 500mm length and cross-sectional diameter of 50mm.

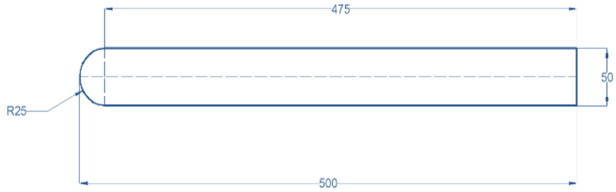

Fig. 1: Base line configuration

Fig. 2: Boat tail base configuration

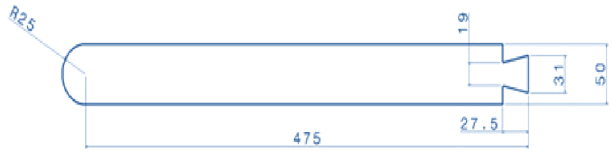

Fig. 3: Nozzle base configuration

The wind tunnel is calibrated by setting rpm and calculating corresponding velocities for each rpm. For calculating velocity, Pitot - static tube is used, which measures dynamic pressure. Then a graph is plotted for rpm vs. Velocity. Electronic pressure scanner is used for measuring pressure from multiple ports simultaneously. It can also be connected to computer through a RS232 online. Further secondary storage memory can be read on the computer and data can be downloaded in excel format or text. In order to distribute the ports over base, area weighted method has been used. In this method, the base of model which is in circular cross section is divided into equal area of concentric circles and the ports are distributed as shown in Fig. 4. Since we are interested in calculating only the base drag, the ports are distributed only at the base [9]. Based on limitations in fabrication of the model, area weighted method is applied. In boat tail base, as the area of base got reduced on comparing to the flat base, the number of ports also got reduced to 21 ports. For fabrication of model, aluminium metal has been used. Since it is non-corrosive, less weight it is preferred for our model. For drilling of the ports 2mm drill bit is used with pressure inserts of 1.8mm outer diameter.

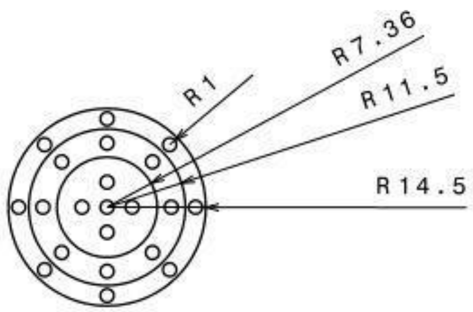

Fig. 4: Ports distribution in boat tail base

3. Results and discussions

Experimental process has a series of steps and those are same for all other models. At first the ports are numbered at the base and to corresponding ports tubes are connected. The connected tubes are taken out through strut and ends of the tube are to be connected to pressure scanner. The model is mounted inside the wind tunnel and held on tightly with the help of bolt along with angle braces [10]. Then the axis alignment is checked for 0° then the pointer braces are also held tightly. Then the run is started inside the tunnel at 600rpm for which the corresponding velocity is 20m/s. In similar way, the velocities are calculated for 1050rpm and 1500rpm. From our calibration, we found that these three rpm give corresponding velocities of 20m/s, 35m/s, 50m/s required for our experiment. The corresponding readings are stored in scanner for our calculation. It's followed for the remaining two models. Figs. 5 and 6 show the pressure coefficient Cp with r/R for boat tail base at different flow velocities for various AoA.

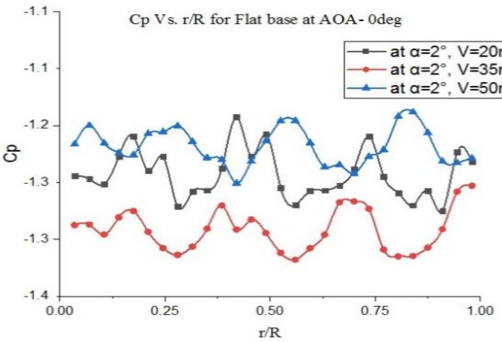

Fig. 5: Cp vs. r/R for boat tail base at α = 0°

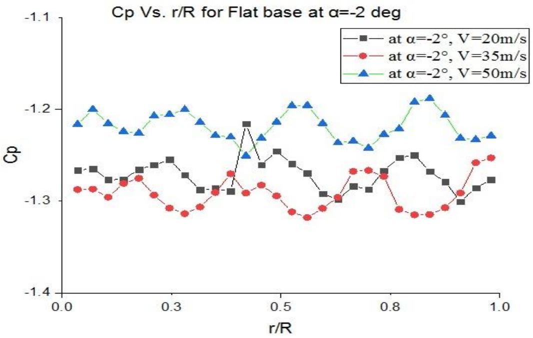

Fig. 6: Cp vs. r/R for boat tail base at α = -2°

Effect of drag coefficient on different AoA with boat tail is shown in Fig. 7. The computational analysis presents higher drag coefficient which is due to body combination which down streams the flow [11]. The reduction in drag coefficient by using a combination of boat tail, bottom hollow and bottom loss at all flight bodies are given. Fig. 8 shows the effect of boat tail angle on different flow velocities, where computational analysis always decreases the angle at higher velocities. As missile increase the boat tail angle, drag coefficient (Cd) will be decreased until a certain optimum value after which Cd will increase with boat tail angle [12]. Effect of drag coefficient on boat tail angle at different velocities, reveal that optimum value of θ can be between 7.5° and 10° for supersonic flow (9.5° for M = 1.6), which is a good result compared with results of Ref. [14], and between 14° and 16° for subsonic flow (15° for M = 0.7) as shown in Fig. 9. Figs. 9-11 show that using boat tail drag coefficient with different AoA at different velocities, demonstrate that Cd reduces by about 55% in subsonic and transonic flows, by about 12% in positive AoA [13, 14]. Reduction of drag coefficient in high-speed flows, especially in higher angles is very important in growing the missiles range.

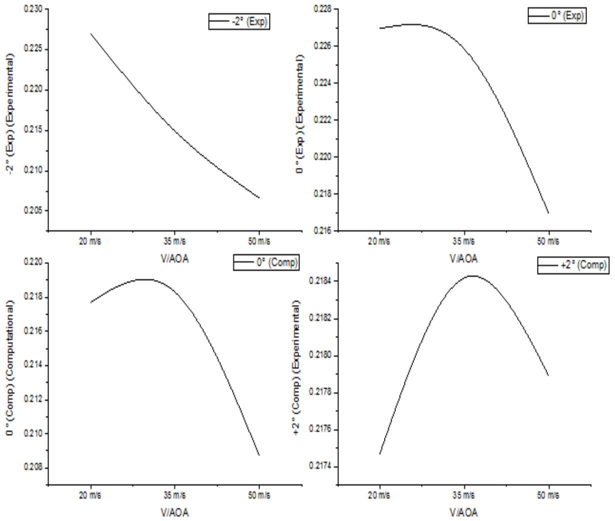

Fig. 7: Effect of boat tail angle on different flow velocities

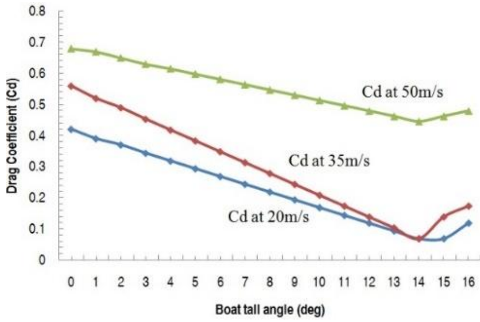

Fig. 8: Effect of Cd on boat tail angle at different velocities

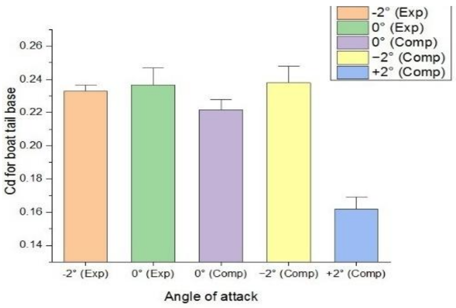

Fig. 9: Cd vs. AoA for boat tail base at 20m/s

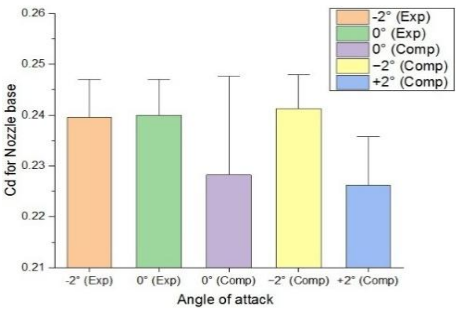

Fig. 10: Cd vs. AoA for boat tail base at 35m/s

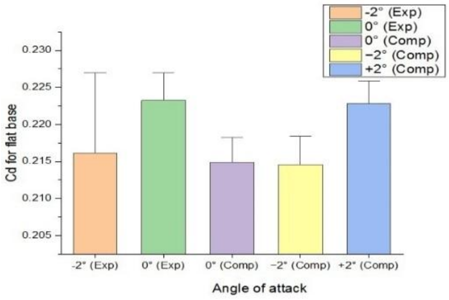

Fig. 11: Cd vs. AoA for boat tail base at 50m/s

4. Conclusion

In this paper a comprehensive study has been done at very low subsonic speeds using experimental techniques to study the base drag on three missile profiles by choosing best suited experimental modelling and fabrication. The pressure distribution on body and base of each missile configuration was obtained and drag on the body due to base pressure was calculated for each of the three missile geometries. The values of coefficient of base drag were found to be higher for flat base configuration and nozzle configuration when compared to boat tail configuration wherein we are taking only drag at shroud of the boat tail and nozzle flare section. But when we consider only the base, we get base drag was higher for boat tail and flat base configuration by small margin to that of the nozzle base. However, coefficient of base drag for nozzle configuration was found to be lesser than that of baseline configuration for which the drag considerably increases with increase in velocity and AoA. From the results obtained for each of three cases, on comparison, it was inferred that the boat tailing of base with estimated drag at shroud of boat tail could be an efficient way of designing the body in order to minimize the drag of projectile. This reduction in drag can be accounted for an increase in the range of missiles.

REFERENCES:

[1] R. Mitchell, M. Webb, J Roetzel, F. Lu and J. Dutton. 2008. A study of the base pressure distribution of a slender body of square cross-section, *Proc. 46th AIAA Aerospace Sci.*, 428, Reno, Nevada. https://doi.org/10.2514/6.2008-428.

[2] W.A. Mair. 1969. Reduction of base drags by boat-tailed after bodies in low-speed flow, *The Aeronautical Quarterly*, 20(4), 307-320. https://doi.org/10.1017/S0001925900005126.

[3] P.R. Viswanath. 1996. Flow management techniques for base and after body drag reduction, *Progress in Aerospace Sci.*, 32(2-3), 79-129. https://doi.org/10.1016/0376-0421(95)00003-8.

[4] V.I. Menezes, K. Takayama, M Sun, J Gopalan and K.P.J. Reddy. 2006. Drag reduction by controlled base flow separation, *J. Aircraft*, 43(5), 1558-1561. https://doi.org/10.2514/1.19903.

[5] K.M. Bak. 2010. Experimental investigation and computational fluid dynamics analysis of missile with grid fin in subsonic flow, *Int. J. Engg. Sci. and Tech.*, 2(11), 6214-6220.

[6] R.J. Krieger and S.R. Vukelich. 1986. Tactical missile drag, tactical missile aerodynamics, *AIAA Proc. Astronautics Aeronautics*, 104, 383-420.

[7] H. Bournot, E. Daniel and R. Cayzac. 2006. Improvements of the base bleed effect using reactive particles, *Int. J. Thermal Sci.*, 45(11), 1052-1065. https://doi.org/10.1016/j.ijthermalsci.2006.01.018.

[8] W.P. Wolfe and W.L. Oberkampf. 1987. Drag predictions for projectiles and finned bodies in incompressible flow, *J. Spacecraft and Rockets*, 24(1), 14-22. https://doi.org/10.2514/3.25867.

[9] M.A. Suliman, O.K. Mahmoud, M.A. Al-Sanabawy and O.E. Abdel-Hamid. 2008. Computational investigation of base drag reduction for a projectile at different flight regimes, *Proc. 13th Int. Conf. Aero. Sci. & Aviation Tech.*, Cairo, Egypt. https://doi.org/10.21608/asat.2009.23743.

[10] M. Ramakrishna, M. Syedhaleem, C. Divya and A. Gurunathan. 2018. Flow characteristics over a missile at higher angle of attacks using experiments and simulation, *Int. J. Vehicle Structures & Systems*, 10(1), 10-13. https://doi.org/10.4273/ijvss.10.1.03.

[11] X. Sun, W. Huang, M. Ou, R. Zhang and S. Li. 2019. A survey on numerical simulations of drags and heat reduction mechanism in supersonic/hypersonic flows, *Chinese J. Aeronautic*, 32(4), 771-784. https://doi.org/10.1016/j.cja.2018.12.024.

[12] B.S. Maqsood, S. Sherbaz and L. Dala. 2017. Drag reduction of supersonic blunt bodies using opposing jet and nozzle geometric variations, *Aerospace Sci. and Tech.*, 69, 244-256. https://doi.org/10.1016/j.ast.2017.06.007.

[13] Y. Gu, G. Zhao, J. Zheng, Z. Li, W. Liu & F.K. Muhammad. 2014. Experimental and numerical investigation on drag reduction of non-smooth bionic jet surface, *Ocean Engg.*, 81, 50-57. https://doi.org/10.1016/j.oceaneng.2014.02.015.

[14] Z.G. Wang, X.W Sun, W. Huang, S.B. Li and L. Yan. 2016. Experimental investigation on drag and heat flux reduction in supersonic/hypersonic flows: A survey, *Acta Astronautica*, 129, 95-110. https://doi.org/10.1016/j.actaastro.2016.09.004.