Aerodynamic Analysis of a Ducted Re-Entry body

1Kanuganti Nived, 2Nalukala Omkar, 3K Sai Priyanka
1Student, 2Student, 3Assistant Professor
Institute of aeronautical engineering

Abstract - **Lower ballistic coefficient offers an optimal re-entry, wherein the vehicle decelerates higher up in the atmosphere thereby decreasing the imposed aerothermal loads. The current numerical study investigates the effects of an add-on duct to the re-entry vehicle. The add-on duct shall circumvent the vehicle from the shoulder to the base. The add-on duct is expected to improve the aerocapture ability of the re-entry vehicle. The drag and ballistic coefficient of the ducted re-entry vehicle will be evaluated and compared with those of the baseline model. Apart from this, the effect of the distance of the duct from the body will also be investigated. In addition to this the variation of drag and ballistic coefficient due to the change in speed regime will also be investigated. All the investigations will be performed using the CFD solver Fluent. The results reveal that the position-1 has the highest drag when the duct-position is at position-1. When the body is tested at Mach 3, the drag coefficient increased for all the positions. The position-2 showcased the highest drag coefficient for the model. The model tested at Mach 5 revealed that the position-2 of the duct has the highest drag coefficient.**

keywords **- Ducted re-entry, CFD, Supersonic.**

I. Introduction

Atmospheric re-entry is a significant aspect in a manned mission. The interplanetary manned mission will require a re-entry into the earths surface. An efficient re-entry requires an accurate accountability of the re-entry method and the trajectory. Care must be taken to determine the g-loads acting on the vehicle, the temperatures the surfaces of the body face, most significantly the loads and temperatures shall be with stand able by the crew.

Generally the re-entry in the earths surface occurs at extreme velocities, velocities that corresponds to the Mach numbers in supersonic and hypersonic regimes. The supersonic and hypersonic flight conditions are related with large amount of difficulties, such as aerodynamic problems.

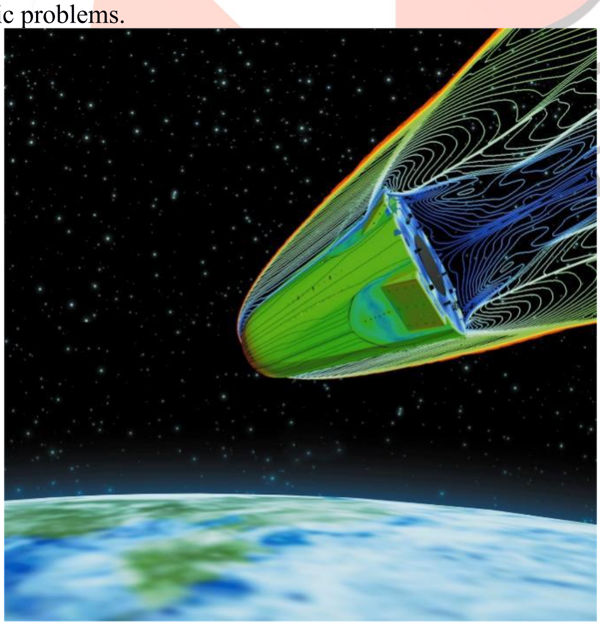


Fig 1. 1: DLR Expert Earth Re-entry vehicle

The re-entry vehicles serves as a tool for space exploration for all the human space missions. The re-entry vehicle must possess emergency abort capabilities, must sustain the crew in the space for a longer period of time, and provide a safe and comfortable re-entry from the deep space into the earths atmosphere. These versatile characteristics are widely regarded as the basis for justifying the betterness of the re-entry vehicle.

In order to incorporate the basic mentioned attributes in a re-entry vehicle, there is several R & D performed years before the mission is accomplished. The research and development involves the experimental and computational test conducted across various systems of the reentry vehicle. The below figure illustrates the experimental tests being carried out on the scaled models of the various space vehicles. The tests shall reveal the flow-configuration obtained for the design considered. In addition, the test shall also allow the investigators to quantify the flow-field parameters, such as the temperature, pressure and fluxes acting on the body of the space vehicle. The quantification will aid to understand the identify suitable material that can withstand such strong temperatures.

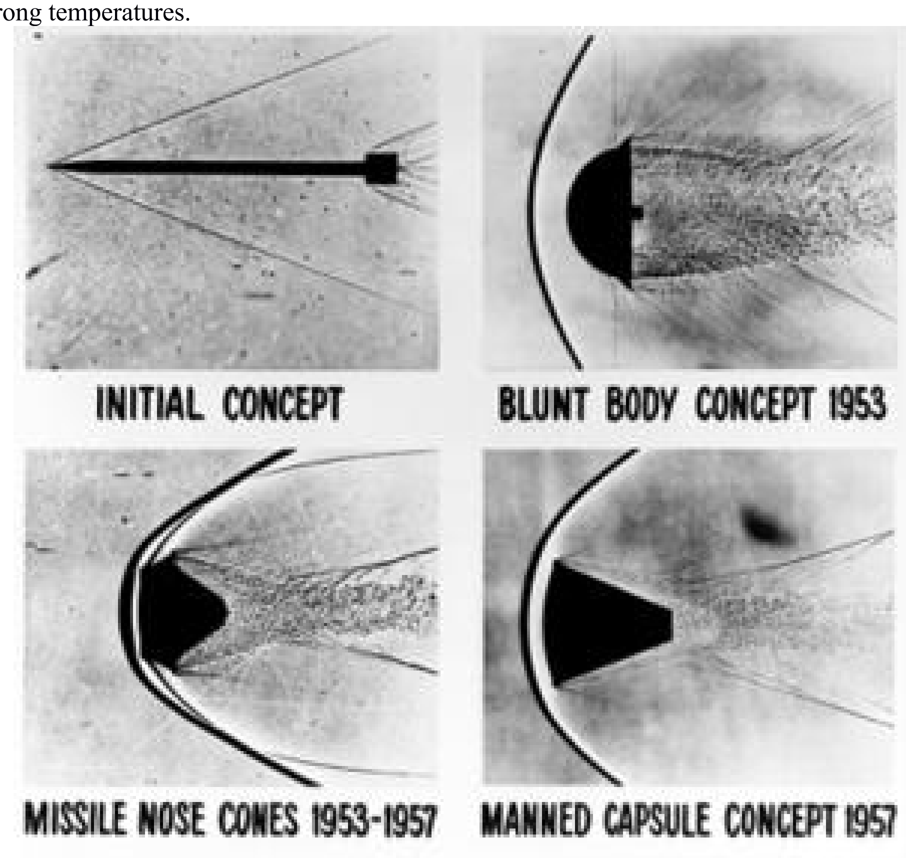


Fig 1. 2: Experimental test on the space vehicles

Apart from the experimental test being performed on the space vehicles of different shapes, there is an excellent computational tool that will allow us to investigate virtually the flow parameters of the various space vehicles. Computational Fluid dynamics[1] has played a significant role in the evaluation of the flow parameters, and research on efficient systems. The below figure (next page) shows the CFD visualization of the flow past a bluff body. The results of CFD can aid in assessing various and numerous aspects of flow. The advent of CFD has allowed various researchers across the world to perform numerical experiments and solver various engineering cases.

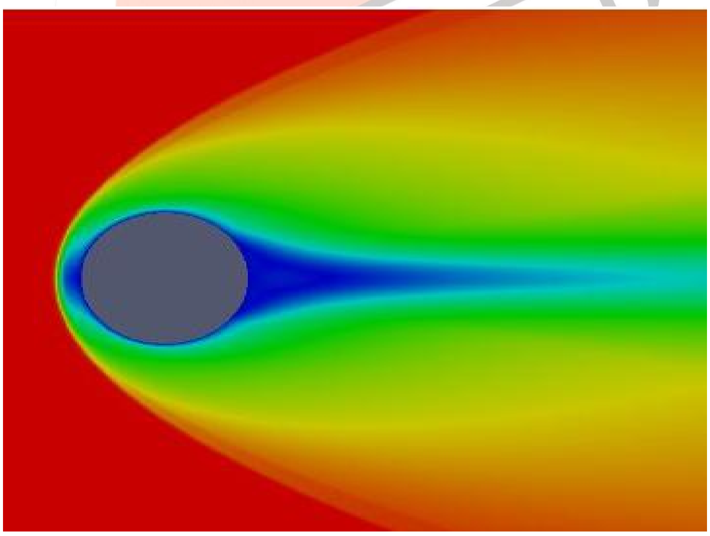


Fig 1. 3: CFD Analysis of a blunt body

For the past years, the aerodynamics of space vehicles (re-entry vehicles) has been an interest for the researchers. Especially the effects on the drag-coefficient dur to a physical modification in any vehicle has piqued the curiosity of several aerodynamicists. The reentry vehicles have always been a subject of vital interest by the researchers due to immense number of practical implications.

Axi-symmetric Shape: This is a simplest shape being employed or considered by various vehicles. [2] Most commonly the forebody of this type of vehicles is a spherical shaped blunt body. The bluntness is required to allocate the payload in the vehicle, and create enough drag during the landings. The below figure illustrates an axisymmetric body. It is a combination of forebody and aftbody. The forebody nose can have any diameter, depending on the requirement. And the aft body follows.

Fig 1. 4 Axi-symmetric body – Picture Courtesy NASA

The variation of forebody and aftbody depends on the application of the space vehicle, the number of crew, the payload, etc. The below figure shows an axi-symmetric body with lengthier aft body, in comparison with the forebody/nose.

Fig 1. 5 Axi-symmetric body -Computational model

II. Literature Survey

Various researchers have investigated the aerodynamics of re-entry bodies. Mungiguerra and Fedele [3] investigated the MINI-IRENE capsule. It is a Flight Demonstrator, which is a novel concept capsule. The demonstrated possessed a variable geometry, the purpose of the variable geometry is to enhance the range of available platforms for retrieving the payloads and data from LEO. The significant characteristics of this vehicle was the "umbrella-like" deployable front structure. The purpose of this structure was to reduce the capsule ballistic coefficient, which lead to acceptable wall heat fluxes, mechanical loads, stability and final descent velocity.

Zhou et al. [4] investigated the various aerodynamic configurations of the capsuled for the reentry manned vehicle. The authors stated that the capsule with high L/D ratio have statically stable trim points, and this is undesirable, because when the vehicle flies past the trim point during re-entry the consequence may be disastrous. The authors proposed an effective design methodology for enhancing mono-stability features of re-entry capsules without compromising the lift-to-drag ratio. The results were demonstrated based on CFD analysis to understand the flow characteristics.

Raj et al. [5] used numerical simulations to propose add-on to the existing re-entry vehicle. A circumventing duct was proposed in their work. Different designs were simulated to evaluate the drag-coefficient and ballistic coefficient of the vehicle with the duct. The results revealed that the drag from all the models was increased with the presence of duct circumventing the re-entry model.

Sharma et al. [6] performed numerical simulation to investigate a novel concept i.e., a flow-through duct in blunt cylinder replicating a re-entry vehicle. The effects of aerodynamic heating and drag were evaluated by using steady state simulations on the axi-symmetric body. The study revealed that the smaller ratio of duct dia have shown significant reduction of peak surface fluxes.

Savino et al. [7] focused on the design of deployable, thermal protection system. Two sphere cone configurations were investigated from an aerodynamics point of view. An evaluation of thermal and aerodynamics loads was performed to assess the longitudinal stability of the capsule.

Chang et al. [8] investigated the aerodynamic characteristics of the re-entry vehicle using CFD. The authors focused on the hypersonic flow regime over a set off capsule configurations. The results demonstrated a good agreement between experimental and numerical. Surface heat flux and aerodynamic force predictions of the capsule configurations are discussed in detail.

Mehta [2] used numerical simulation to analyse the flow past apollo and orex re-entry vehicle. The author demonstrated the flow-field characteristics of the such as bow shock, sonic line, expansion fan, and recirculating zone in the numerical computations. The effects of the module geometrical parameters, such as radius of the spherical cap radius, shoulder radius, cone angle and back shell inclination angle on the flow field, which will provide a useful input for the optimization of the reentry module.

Whitmore[9] et al. compared four different configurations to analyse the aerodynamics of the bionic and capsule configuration. The volumetric efficiency, peak deceleration during reentry, peak dynamic pressure during reentry, available downrange during reentry, available cross range during reentry, handling qualities and controllability, stagnation point heating rates and temperatures, and technology readiness level were considered as the parameters for evaluating the performance characteristics.

Bhaktian [10] explored the augmentation of decelerative forces experience during re-entry. The amount of drag was evaluated using the jet manipulation system.

Although numerous works focused on the re-entry vehicle, few works discussed the drag characteristics and ballistics of the bluff body. In this work a quantification with a systematic analysis of the flow over a bluff body is performed to evaluate the drag and ballistic co-efficient of the blunt body with circumventing duct. The blunt body considered for this study will be an axi-symmetric body.

III. Methodology

Ballistic coefficient determines the amount of deceleration a body possess. The below figure illustrates the two commonly used re-entry vehicle. The body on the left is a streamlined body, the stream line bodies have high ballistic coefficient. On the other side, the body is a blunt vehicle with low-ballistic coefficient. Generally, the body with low-ballistic coefficient has higher decelerative forces. Therefore, during re-entry low-ballistic coefficient is preferred. The difference between both the bodies is with respect to the aft body, on the left side the aftbody is increasing its diameter, on the right side, the diameter of the aft body is reducing.

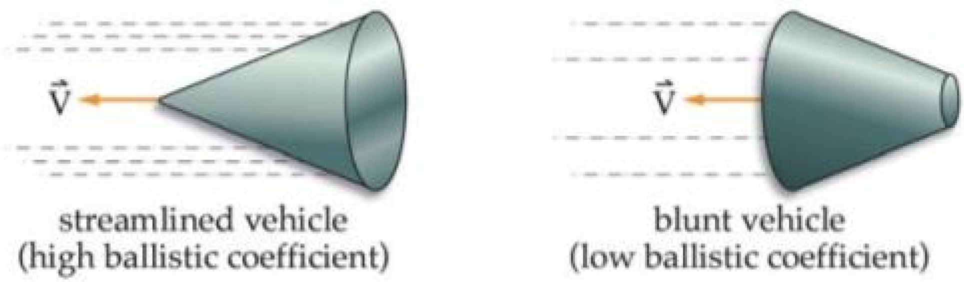


Fig 3. 1: Blunt versus Streamlined Vehicle

In this research, we are investigating the ballistic coefficient and drag coefficient for the body which has a constant aft body diameter, as shown in figure 3.2. The body has a diameter of 15mm. The length of aft body is 15mm.

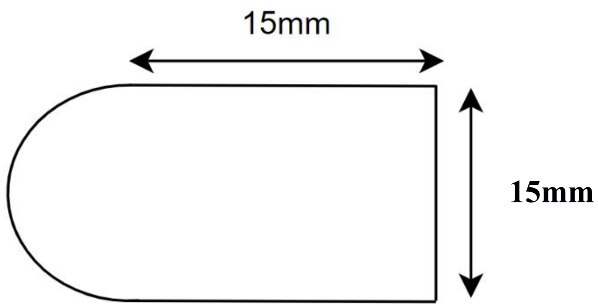


Fig 3. 2 Dimensions of the Main body

The next model, which is targeted to improve the decelerative forces is the model with a duct circumventing the aft body of the above model. The below figure (fig 3.3) shows the model with the duct.

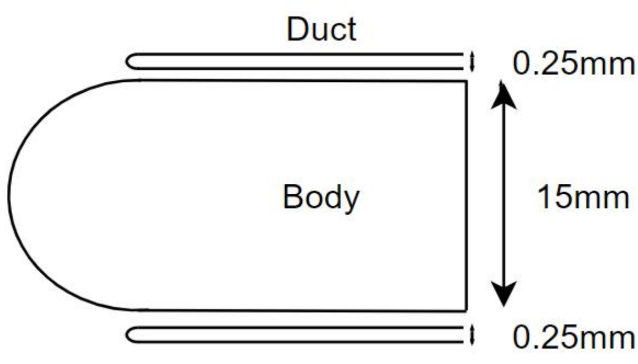


Fig 3. 3: Dimensions of the Main body with Duct

The duct is along the length of the model. The lip radius of the duct is 0.125mm. The research involves the investigation of the airflow characteristics i.e., the drag characteristics and ballistics coefficient of the model with duct placed at various positions. The distance between the duct and the aft body is varied as follows:

Table 3. 1: Positions of the Duct

Model	Distance of Model with the body (mm)
1	0.25
2	0.35
3	0.45

Three different models are created by considering the different positions of the duct. The geometry is created in the ANSYS Design modeler. Each position of the duct is investigated for three different Mach numbers 2, 3 and 5.

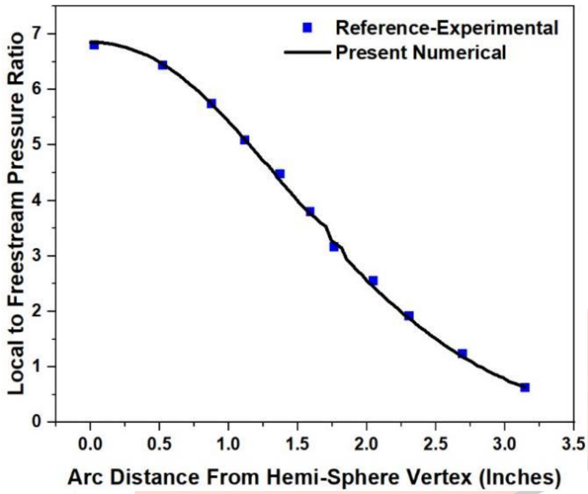
In order to validate the numerical scheme, an experimental study by white[11] is considered. The investigated experimentally, the flow past a bluff body having 4 inches diameter. The experimental study involved the flow of Mach 2.23. The below graph illustrated the comparison of local to freestream pressure ratio on the blunt body.

The experiment boundary conditions are tabulated below

Table 3. 2: Experimental Boundary Condition

Mach	Angle of Attack	Pressure altitude
2.23	0	32,000 Feet

The dimensions of the experimental model is illustrated in the below figure.



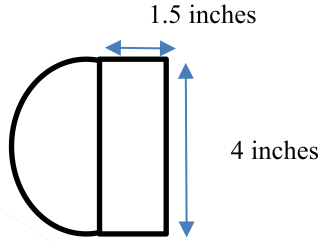


Fig 3.4: Experimental Vs. Numerical Pressure Distribution

The graph in figure 3.4 clearly illustrates that with the considered numerical scheme, the solver fluent has resulted in identical pressure distribution. Therefore, this demonstrates a successful code verification to perform consequent CFD simulation

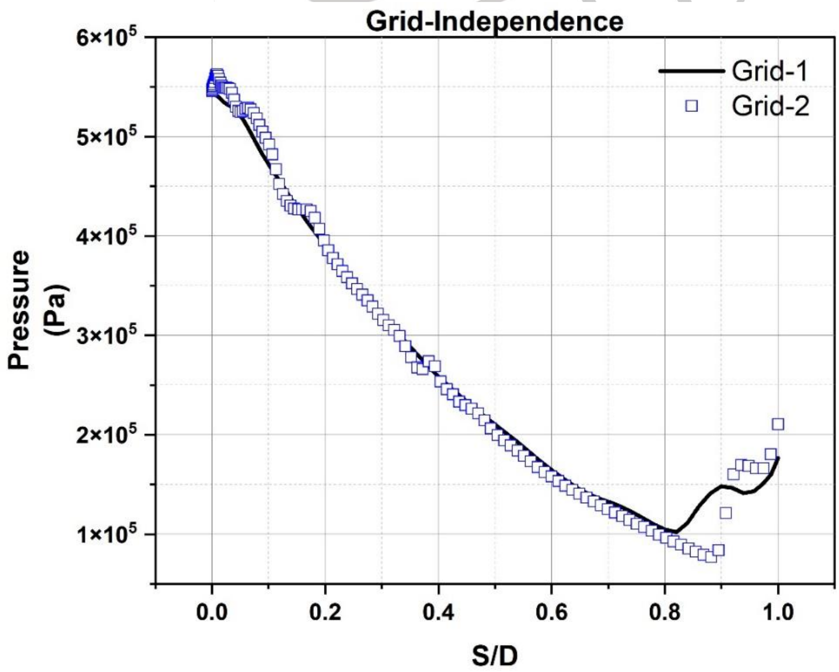


Fig 3.5: Grid Independence Study

The numerical solver Fluent is used to perform a flow analysis, for this the duct is considered that will be simulated numerically, followed by modification of the geometry.

In this research work we considered 3 positions at 3 Mach numbers that is at Mach 02,at Mach 03 and at Mach 05

IV. RESULTS

The results evaluated the aerodynamic characteristics of the model with different duct positions. The figure 6.1 represents the Cd obtained at different positions of the duct for the different Mach number.

The results reveal that the position-1 has the highest drag when the duct-position is at position-1. When the body is tested at Mach 3, the drag coefficient increased for all the positions. The position-2 showcased the highest drag coefficient for the model. The model tested at Mach 5 revealed that the position-2 of the duct has the highest drag coefficient.

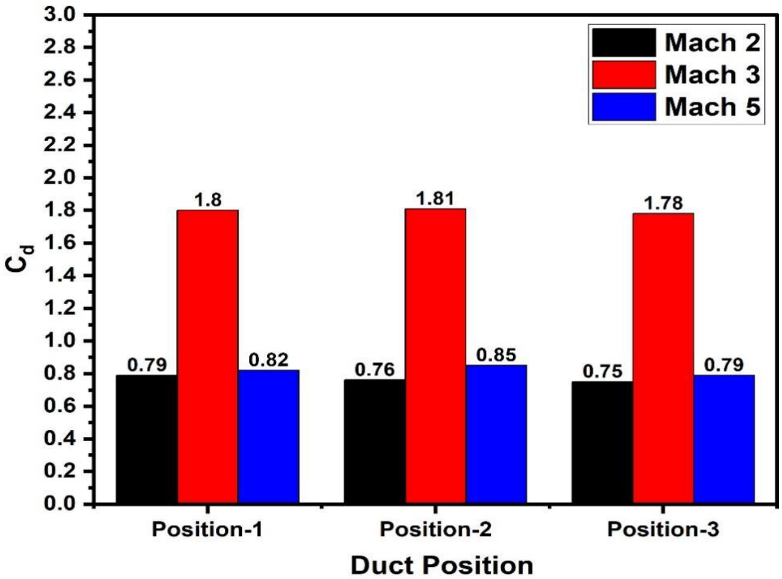


Fig 4. 1: Coefficient of Drag Vs. Duct Positions

The drag characteristics were different for all the cases. In order to quantify the ballistic coefficient, the drag values along with the reference area is used. The following formula is used to calculate the ballistic coefficient [12]:

$$\beta = \frac{m}{C_D S}$$

M is the mass of the body which is assumed as 75 grams based on the size of the model [5], Cd is the drag coefficient and S is the reference area.

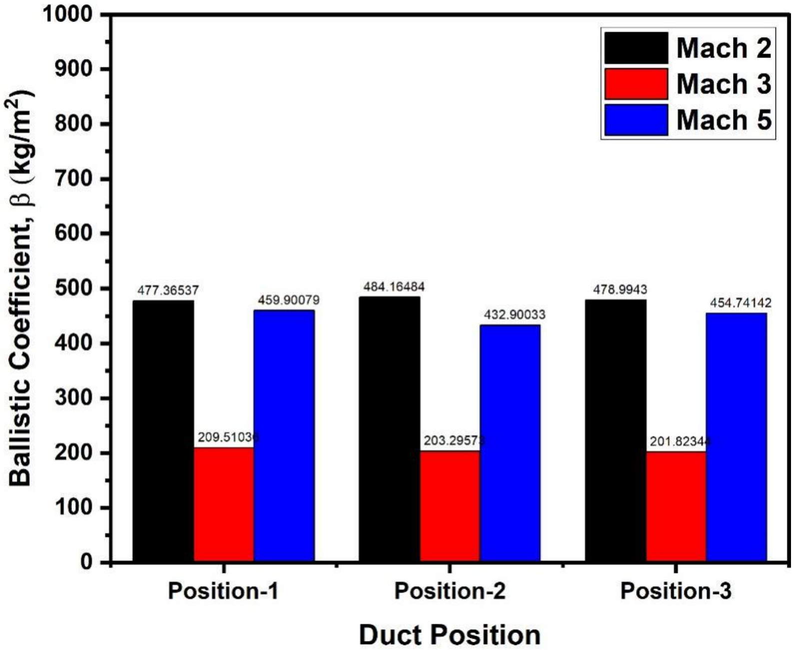


Fig 4. 2: Ballistic Coefficient Vs. Duct Position

The above figure illustrates the ballistic coefficient vs, the duct position. The lesser the ballistic coefficient the higher the decelerative forces that can withstand the model. The model at Mach 2 revealed that the position-2 has the highest ballistic coefficient. For the Mach-3, the position-1 and for Mach-5 the position-1.

V. Conclusion

Numerical experiments were performed on the blunt model to investigate and account the coefficient of drag acting on the model at different Mach numbers. In addition to the drag coefficient, ballistic coefficient is also evaluated. The study targeted the evaluation of the duct circumventing the aft body of the re-entry vehicle. The circumventing duct was varied with respect to the distance from the aftbody and the circumventing duct.

Three positions were evaluated, at three different mach numbers i.e., 2 , 3 and 5. The contours illustrated the shock waves and the various configurations of the flow filed. The drag coefficient revealed different outcome for each position and mach number. The ballistic coefficient revealed that the for Mach-2, the position-1 showed lower values of ballistic coefficient this indicates that this position has the highest decelerative forces. And for Mach-3, the position-3 revealed a lower ballistic coefficient, and at Mach-5 the position-2 has the highest decelerative forces.

References

[1] van Leer, B., and Powell, K. G. “Introduction to Computational Fluid Dynamics.” Encyclopedia of Aerospace Engineering, No. October, 2010. https://doi.org/10.1002/9780470686652.eae048.

[2] Mehta, R. C. “Computations of Flow Field over Apollo and OREX Reentry Modules at High Speed.” Indian Journal of Engineering and Materials Sciences, Vol. 15, No. 6, 2008, pp. 459–466.

[3] Fedele, A., and Mungiguerra, S. “Aerodynamics and Flight Mechanics Activities for a Suborbital Flight Test of a Deployable Heat Shield Capsule.” Acta Astronautica, Vol. 151, No. May, 2018, pp. 324–333. https://doi.org/10.1016/j.actaastro.2018.05.044.

[4] Chen, B., Zhan, H., and Zhou, W. “Aerodynamic Design of a Re-Entry Capsule for High-Speed Manned Re-Entry.” Acta Astronautica, Vol. 106, 2015, pp. 160–169. https://doi.org/10.1016/j.actaastro.2014.10.036.

[5] ANBU SERENE RAJ, C., NARASIMHAVARADHAN, M., VAISHNAVI, N., ARUNVINTHAN, S., AL ARJANI, A., and NADARAJA PILLAI, S. “Aerodynamics of Ducted Re-Entry Vehicles.” Chinese Journal of Aeronautics, Vol. 33, No. 7, 2020, pp. 1837–1849. https://doi.org/10.1016/j.cja.2020.02.019.

[6] Sharma, G., Yadav, R., Studies, E., and Kulkarni, P. “Th 17 Annual CFD Symposium , August 11-12 , 2015 , Bangalore 17 Th Annual CFD Symposium , August 11-12 , 2015 , Bangalore Numerical Simulation of the Effect of a Flow-Through Duct on the Aerodynamic Heating and Drag of a Blunt Body.” No. August, 2015.

[7] Carandente, V., Zuppardi, G., and Savino, R. “Aerothermodynamic and Stability Analyses of a Deployable Re-Entry Capsule.” Acta Astronautica, Vol. 93, 2014, pp. 291–303. https://doi.org/10.1016/j.actaastro.2013.07.030.

[8] Chang, C. L., Venkatachari, B. S., and Cheng, G. C. “Effect of Counterflow Jet on a Supersonic Reentry Capsule.” Collection of Technical Papers - AIAA/ASME/SAE/ASEE 42nd Joint Propulsion Conference, Vol. 6, 2006, pp. 4498–4519. https://doi.org/10.2514/6.2006-4776.

[9] Whitmore, S. A., Banks, D. W., Andersen, B. M., and Jolley, P. R. “Direct-Entry, Aerobraking, and Lifting Aerocapture for Human-Rated Lunar Return Vehicles.” Collection of Technical Papers - 44th AIAA Aerospace Sciences Meeting, Vol. 17, No. January, 2006, pp. 12382–12436. https://doi.org/10.2514/6.2006-1033.

[10] Bhagwandin, V., and Sahu, J. Numerical Prediction of Pitch Damping Stability Derivatives for Finned Projectiles. In 29th AIAA Applied Aerodynamics Conference, American Institute of Aeronautics and Astronautics, 2011.

[11] White, J. T. “Application of Navier-Stokes Flowfield Analysis to the Aerothermodynamic Design of an Aerospike-Configured Missile Hughes Missile Systems Company Pomona , California AIAA / AHS / ASEE Aerospace Design Conference For Permission to Copy or Republish , Conta.” 1993.

[12] Sellers, J. J. “Returning from Space: Re-Entry.” Understanding Space: An Introduction to Astronautics, 2000, pp. 308–339.