

COB-2023-1895

PARAMETRIC STUDY OF AN EXPERIMENTAL ROCKET'S DRAG IN TRANSONIC FLOW

Pedro Lacerda
Vinicius Dionisio
Guilherme Takahashi

Universidade de Brasília

pedrolacerdamontes@gmail.com, nicius_alves@hotmail.com, gui.luis.carvalho@gmail.com

Abstract. *This paper presents a study aimed at reducing the drag generated by the nose cone of an experimental rocket that operates in the transonic regime. The motivation behind the research was the lack of references on the influence of the fineness ratio of the nose cone on the drag of rockets in the transonic regime. The research is relevant for future projects as experimental rocketry teams often design rockets that operate in this flow regime. The authors used a parametric study, employing the commercial CFD software, ANSYS Fluent, to calculate the drag generated by a Von Karman shaped nose cone. The parameter varied was the fineness ratio of the nose cone, which ranged from 1 to 3 with intervals of 0.5, and the observed parameter was the total drag generated by the nose cone. The simulations showed that the ideal fineness ratio to minimize the drag generated by the nose cone in these conditions is around 2.5 and this conclusion is reinforced by analyzing the trends of the viscous and pressure drags. However, when compared to the total drag generated by the rocket, the improvement achieved by adjusting the fineness ratio of the nose cone is not significant. The best case scenario results in less than 5% reduction in drag, while the improvement compared to other cases is less than 1%. Overall, this study contributes to the understanding of the influence of the fineness ratio of the nose cone on the drag of rockets in the transonic regime. While the results showed some improvement, the study suggests that there may be other factors that play a more significant role in drag reduction for experimental rockets. Therefore, the findings of this study can serve as a reference for future designs. The authors suggest that further experiments in a wind tunnel are necessary to validate the results obtained numerically in this work*

Keywords: *Transonic, CFD, Nose cone, Fineness*

1. INTRODUCTION

Efficient nose cone design plays a crucial role in minimizing drag in rocketry. When designing a nose cone, it is important to consider two primary sources of drag: viscous drag and pressure drag. Flow regime is categorized based on the Mach number in subsonic, transonic, supersonic, and hypersonic flows. In subsonic flow, the entire flow has a Mach number below 1, supersonic flow occurs when the Mach number exceeds 1 and hypersonic flow refers to Mach numbers greater than 5. Transonic flow is characterized by a Mach number range between 0.8 and 1.2, where a mixture of subsonic and supersonic flow can coexist.

Viscous drag, also referred to as skin drag, emerges due to the fluid-surface interaction and prevails in subsonic flows. Conversely, pressure drag arises from the aerodynamic forces induced by the airflow around an object and becomes more prominent in supersonic flow regimes. To mitigate viscous drag, a shorter and blunter shape is preferred as it minimizes surface area, while to reduce pressure drag, longer and sharper nose cone designs are recommended (Iyer and Pant, 2020).

However, in transonic flow, where both subsonic and supersonic flow regions coexist, it becomes challenging to determine the optimal nose cone design for minimizing drag. The transition between subsonic and supersonic flow introduces complex aerodynamic phenomena, making it difficult to establish a definitive strategy for drag reduction.

Various factors influence drag generation in the nose cone, and researchers have made significant efforts to better understand them. For instance, Carvalho and Claudino (2019) conducted a computational fluid dynamics (CFD) analysis to investigate the effects of different nose cone shapes on drag generation in subsonic flow. In their study, they compared tangent, elliptical, conical, and parabolic nose cone shapes across a range of Mach numbers from 0.05 to 0.62. Their findings revealed that, under those specific conditions, the elliptical shape outperformed the other nose cone shapes in terms of drag reduction.

Another relevant study is conducted by Sabanovic (2021), who investigated the effects of leading edge bluntness in projectiles at supersonic speeds ranging from Mach 2 to 4. The study found that incorporating a blunt tip in the nose cone design improved the overall performance, despite the resulting increase in surface area.

In a study by Sahai *et al.* (2014), minimum drag revolution bodies were derived for hypersonic flow conditions considering varying fineness ratios. The fineness ratio represents the ratio of the length to the diameter of the body, whereby

shorter bodies with larger diameters have low fineness ratios, and longer bodies with smaller diameters have high fineness ratios. The research findings indicated that increasing the fineness ratio led to a decrease in wave drag coefficient.

In the context of transonic flow, Kumar (2015) conducted a comparative study utilizing CFD to evaluate the performance of ogival, conical, and elliptical nose cone shapes at Mach 1. The findings revealed that the ogive shape outperformed the other nose cone shapes, exhibiting lower drag force and consequently reduced aerodynamic heating.

Furthermore, the influence of the fineness ratio on drag generation in transonic flow has received limited attention in previous studies. Therefore, in order to make a contribution to this area of research, this study aims to investigate the underlying phenomena that contribute to drag generation in an experimental rocket operating in transonic flow.

2. ROCKET DESCRIPTION

The focus of this study is the experimental rocket developed by the rocketry team from Universidade de Brasília Capital Rocket Team in project Boscov. The rocket is composed of three main components: nose cone, body tube and boat tail. The body tube has a length equal to $L_b = 2.39$ m, the boat tail is 0.09 m long and has a diameter of 0.10 m at the end.

The nose cone has a Haak series shape with parameter $C = 0$, also known as the Von Kármán nose cone, which is defined by Eq. 1 and Eq. 2, where R and L are respectively the radius and the length of nose cone (Crowell Sr, 1996). The diameter of the nose cone is the same as the diameter of the airframe, which is 0.155 m, but its length is the subject of this paper's investigation. Figure 1 provides model of the rocket without fins, highlighting its main dimensions.

$$\theta(x) = \arccos\left(1 - \frac{2x}{L}\right) \quad (1)$$

$$y(\theta, C) = \frac{R}{\sqrt{\pi}} \sqrt{\theta - \frac{\sin(2\theta)}{2} + C \sin^3(\theta)} \quad (2)$$

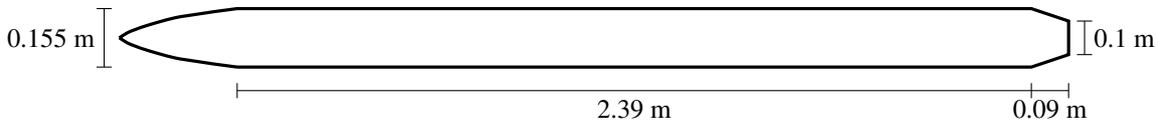


Figure 1. Finless model of the rocket. Source: Made by the authors.

Predicted trajectory obtained through a 6 degrees of freedom flight simulator indicates that the rocket is expected to reach a maximum velocity of approximately Mach 0.8, which may be considered the lower limit of the transonic regime. To optimize the nose cone design for improved performance, a series of CFD simulations were performed aiming to determine the ideal nose cone length that minimizes drag and enhances the rocket's overall performance under these specific conditions.

3. METHODOLOGY

Simulations were conducted using the commercial CFD software Ansys Fluent. To simplify the model, 2D axisymmetric simulations were performed. The implicit density based solver, alongside with the ideal gas model, was used to capture compressibility effects that may arise in transonic flow.

Turbulence was addressed using the SST $k-\omega$ turbulence model. This model has demonstrated its effectiveness in accurately predicting drag forces in both near-wall and far-wall regions, as long as a sufficiently refined mesh is employed near the walls (Vaughn and Chen, 2007). Following standard guidelines, mesh elements close to the walls were refined to be within the laminar sublayer of the boundary layer, that is, to have $y^+ \leq 5$. The y^+ represents the non-dimensional normal coordinate in respect to the walls and plays a crucial role in the functionality of the SST model.

Figure 2 depicts the computational domain and Tab. 1 summarizes the numerical setup used in the simulations.

A parametric study was conducted to determine the optimal nose cone length that minimizes drag. The study varied the fineness ratio (FR) of the nose cone. The FR values were varied from 1 to 3 in increments of 0.5 and the total drag generated by the nose cone was the main parameter observed and compared in each simulation.

Grid independence analysis was performed in order to guarantee reliable results. Cell density was varied both in the region close to and far from the walls as well as inside the boundary layer. Results from the analysis can be seen in Fig. 3 and were used in the creation of the meshes used in the calculations. Individual grids were generated for each of the five cases in order to account for the different nose cone lengths. Table 2 provides statistical information for each grid, while Fig. 4 illustrates an example of the grids utilized in the simulations.

Convergence was assessed using multiple criteria, including the following:

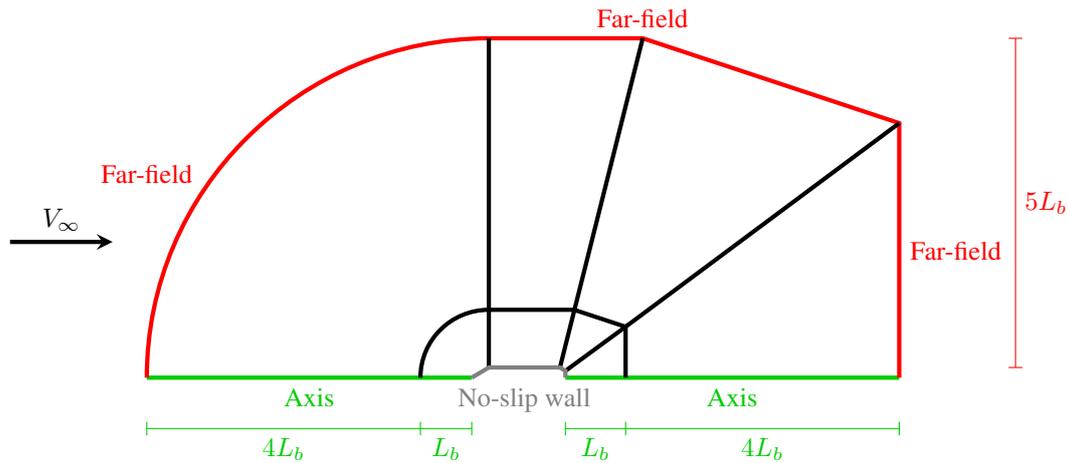


Figure 2. Computational domain used in the simulations. Source: Made by the authors.

Table 1. Computational domain and numerical setup.

Quantity	Value
Solver type	Implicit density based
Time	Steady
2D space	Axisymmetric
Turbulence model	SST k- ω
Density formulation	Ideal gas
Spatial discretization	First order upwind
Boundary condition (upstream, upper and downstream)	Pressure far-field
Boundary Mach number	0.8 axial
Boundary location with respect to walls	$5L_b$

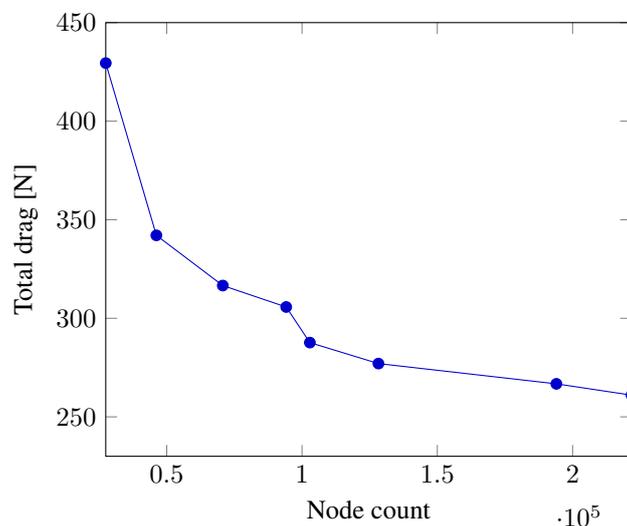


Figure 3. Grid independence analysis results. Source: Made by the authors.

- Scaled residuals: The residuals of the governing equations were monitored, and convergence was considered achieved when the scaled residuals reached a value of $10e-5$ or below.
- Total mass flux: The total mass flux through the computational domain was monitored. Convergence was indicated by the mass flux approaching zero, indicating a balanced flow with no significant inflow or outflow.
- Total drag: The total drag acting on the rocket was monitored throughout the simulations. Convergence was indicated by the drag value stabilizing and remaining constant.
- Local Mach number: The Mach number at scattered points in the flow field was monitored. Convergence was indicated by the Mach number remaining constant in each of these points.

Table 2. Mesh statistics for each case.

Quantity	Value				
	1	1.5	2	2.5	3
FR	1	1.5	2	2.5	3
Number of nodes	221786	221786	221786	221786	221786
First layer height [m]	3.18e-6	3.18e-6	3.18e-6	3.18e-6	3.18e-6
Maximum $y+$	1.26	1.26	1.29	1.30	1.30
Maximum skewness	0.41	0.59	0.65	0.69	0.92
Minimum orthogonal quality	0.31	0.26	0.18	0.14	0.11

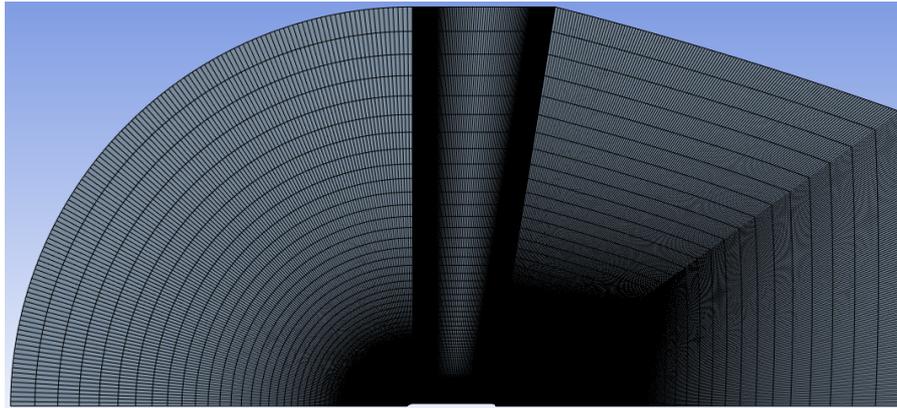


Figure 4. Typical mesh used in the simulations. Source: Made by the authors.

Observing the previous mentioned parameters, simulation convergence was reached in each case.

4. RESULTS AND DISCUSSION

The results of the nose cone analysis can be seen in Tab. 3. It can be seen from the data obtained that, as expected, as FR increases two things happen:

- Viscous drag increases due to increased surface area.
- Pressure drag decreases due to a more streamlined body.

However, as the fineness ratio increases beyond a certain point, the improvement in pressure drag becomes insignificant, and viscous drag starts to dominate, resulting in an overall increase in total drag. This trend can be observed by examining the curves for both viscous and pressure drag presented in Fig. 5. In the specific conditions analyzed in this study, it was found that, even though the minimum total drag was obtained at a fineness ratio of 2.5, there is almost no improvement in total drag when increasing the fineness ratio to values higher than 2.

This conclusion is further supported by examining the Mach number contours shown in Figures 6 and 7. In both figures, it is evident that a normal shock wave forms at the base of the nose cone, leading to a significant increase in pressure drag. On the other hand, for all the other cases, the flow remains purely subsonic.

The findings of this study suggest that elongating the nose cone may have the potential to prevent the formation of normal shock waves entirely. While the focus of this research was on the specific case of flow at Mach = 0.8, it is plausible that the most effective approach to minimizing drag in a nose cone in transonic flow with Mach less than 1 is to identify the minimum nose cone length that prevents the occurrence of shock waves. By doing so, viscous drag can be minimized and pressure drag significantly reduced. However, further investigation is required to validate this hypothesis and gain a comprehensive understanding of its implications.

Table 3. Results of drag in the nose cone.

FR	Pressure [N]	Viscous [N]	Total [N]
1	47.10	9.07	56.17
1.5	18.08	12.16	30.24
2	13.08	14.87	27.96
2.5	9.84	17.57	27.42
3	7.85	20.22	28.07

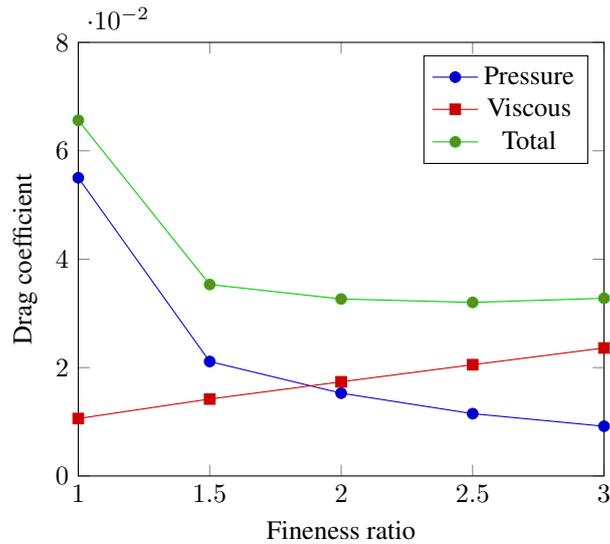


Figure 5. Drag coefficient on the nose cone as a function of fineness ratio. Source: Made by the authors.

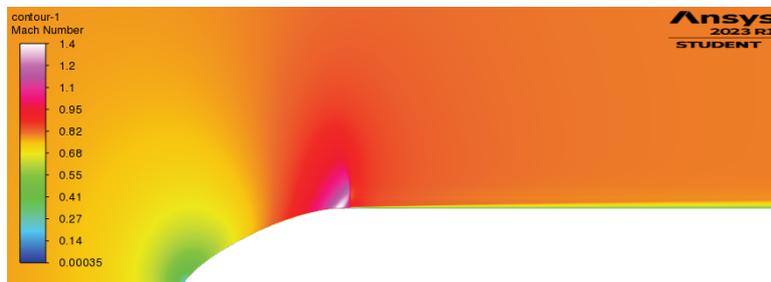


Figure 6. Mach number contours for the FR = 1 case. Source: Made by the authors.



Figure 7. Mach number contours for the FR = 1.5 case. Source: Made by the authors.

Additionally, the study also collected drag data from the rest of the components of the rocket, namely the body tube and the boat tail, to assess the impact of the nose cone on their drag characteristics. The drag values obtained for each of these components, along with the corresponding percentual improvement in total drag reduction, which can be calculated by Eq. 3, are presented in Table 4. The results reveal that, similar to the findings for the nose cone alone, there is not a substantial improvement in drag reduction after the fineness ratio reaches 1.5. At this point, the maximum improvement in total drag reduction is observed to be 7.14%.

$$\text{Improvement}_i = \frac{\text{Worst case} - \text{Total drag}_i}{\text{Worst case}} \times 100 \quad (3)$$

5. CONCLUSIONS

A parametric study was conducted using the CFD software Ansys Fluent to determine the nose cone length that minimizes drag in transonic flow for the rocket developed by Capital Rocket Team in project Boscov. The fineness ratio of the nose cone was systematically varied from 1 to 3 in increments of 0.5. The total drag of the nose cone and the entire rocket was compared for each fineness ratio to identify the optimal length. The results revealed that, for the nose cone alone, there was no improvement in drag reduction beyond a fineness ratio of 2.5. However, for the full rocket, the optimal fineness ratio was found to be 1.5.

Table 4. Results of total drag in the rocket.

FR	Nose Cone [N]	Body Tube [N]	Boat Tail [N]	Total [N]	Improvement [%]
1	56.17	107.27	97.31	260.75	0.00
1.5	30.24	111.63	100.27	242.14	7.14
2	27.96	111.39	103.51	242.86	6.86
2.5	27.42	111.48	105.80	244.70	6.15
3	28.07	110.80	106.31	245.18	5.97

Moving forward, the next steps of this study would involve experimental validation to assess the impact of real-life variables such as surface roughness, manufacturing imperfections, protrusions, and holes on the overall drag of the rocket. By conducting experiments, the researchers can further investigate and confirm the findings obtained through numerical simulations, ensuring the accuracy and reliability of the results in practical scenarios.

6. ACKNOWLEDGEMENTS

We would like to sincerely acknowledge the invaluable contributions and support provided by PhD. Olexiy Shynkarenko and PhD. Jungpyo Lee throughout the course of this project. Their expertise in hybrid propulsion, experimental rocketry, and overall guidance greatly enriched our understanding and implementation of the research.

Furthermore, we would like to extend our heartfelt thanks to the Capital Rocket Team for their exceptional contributions to the field of rocketry in Brazil. Their dedication, collaboration, and shared knowledge have been instrumental in advancing our research and fostering a vibrant rocketry community in the country.

7. REFERENCES

- Carvalho, L. and Claudino, G., 2019. "Cfd analysis of drag force for different nose cone design".
- Crowell Sr, G.A., 1996. "The descriptive geometry of nose cones". URL: <http://www.myweb.cableone.net/cjcrowell/NCEQN2.doc>.
- Iyer, A.R. and Pant, A., 2020. "A review on nose cone designs for different flight regimes". *International Research Journal of Engineering and Technology (IRJET)*, Vol. 7, No. 08, pp. 3546–3554.
- Kumar, G., 2015. "Cfd analysis of transonic flow over the nose cone of aerial vehicle".
- Sabanovic, T., 2021. "Analysis of the influence of 5.56 mm projectile shape on drag coefficient using cfd". *Defense and Security Studies*, Vol. 2, p. 79–85. doi:10.37868/dss.v2.id172. URL <https://journals.ardascience.com/index.php/dss/article/view/172>.
- Sahai, A., John, B. and Natarajan, G., 2014. "Effect of fineness ratio on minimum-drag shapes in hypersonic flows". *Journal of Spacecraft and Rockets*, Vol. 51, No. 3, pp. 900–907.
- Vaughn, M. and Chen, C., 2007. "Error versus y^+ for three turbulence models: Incompressible flow over a unit flat plate". In *18th AIAA Computational Fluid Dynamics Conference*. p. 3968.

8. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.