XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGENEERING

CFD-based Study of boat tail Length Effects on 155 mm Projectile Drag

Lucas G. Amaral[1], Camila C. P. de Paula[1], João V. M. Gil[1], Victor S. Santiago[1], André L. T. Rezende[1]

[1]*Seção de Engenharia Mecânica, Instituto Militar de Engenharia*
Praça General Tibúrcio, 80 - Urca, Rio de Janeiro - RJ, 22290-270, Brazil
lucas.amaral@ime.eb.br, milapifano@ime.eb.br, jvmgil@ime.eb.br, santoro@ime.eb.br, arezende@ime.eb.br

Abstract. The study of projectile aerodynamics plays a crucial role in the development and optimization of ammunition, particularly in enhancing range and accuracy. In this context, the present work aims to investigate the effects of varying the boat tail length of a 155 mm artillery projectile on its aerodynamic drag coefficient. The methodology involved computational fluid dynamics (CFD) simulations in which several geometric configurations of the projectile's boat tail were analyzed. These simulations allowed for a detailed evaluation of the pressure and velocity fields around the projectile's surface, leading to the determination of aerodynamic coefficients for each configuration. Initially, the numerical results were validated against data from another reference on the subject, and subsequently, the mesh was employed to generate new numerical results. It was observed that specific geometric proportions result in a noticeable decrease in the drag coefficient, thus confirming the efficiency of the boat tail design technique in aerodynamic optimization. Moreover, the study underscores the relevance of rear-end geometry in the overall aerodynamic performance of artillery projectiles. The results provide valuable technical insights for the design and improvement of future ammunition, contributing to the broader field of external ballistics. The findings also highlight the importance of computational modeling as a powerful tool for evaluating structural modifications and guiding design decisions aimed at reducing aerodynamic resistance and improving projectile efficiency.

Keywords: Aerodynamic Forces, CFD, External ballistic, Ammunition.

1 Introduction

The study of projectile aerodynamics and its impact on external ballistics, particularly accuracy and range, is essential to advance ammunition performance and artillery development in the Brazilian army. The aerodynamic drag that acts on the projectiles can be divided into two main components: shear stress and surface pressure. The drag generated by shear stress is viscous in nature and results from air friction (Fox [1]). The drag associated with pressure differences includes two main types: wave drag, produced by high pressure at the projectile's front, and base drag, caused by low pressure at the rear. Base drag plays a major role in overall aerodynamic performance, as it arises from flow recirculation behind the projectile, leading to reduced pressure at the base and creating a force opposing its motion (Dali and Jaramaz [2]; Suliman et al., [3]).

To mitigate base drag, one of the most common design strategies is the application of a boat tail, which consists of reducing the projectile's rear section. Modifying the boat tail geometry (length and angle) significantly influence drag and range. Previous studies, such as The Engineering Toolbox [[4], analyzed different boat tail angles to optimize drag reduction for low-speed models, showing that optimal configurations at low velocities differ considerably from those at high Mach numbers. Even when the boundary layer separates across the base, the boat tail has been shown to significantly reduce total drag.

In addition, Suliman et al. ([3]) conducted computational simulations on the 155 mm M549 projectile, varying parameters such as Mach number and boat tail angle to assess efficiency. Their results indicated that, while optimal angles depend on specific flow characteristics, all configurations produced significant drag reduction.

In this context, the present study investigates the effect of boat-tail length variation on the drag and range of 155 mm artillery projectiles. To this end, Computational Fluid Dynamics (CFD) techniques are employed to analyze the airflow around modified projectile geometries and to determine the corresponding drag coefficients. Initially, the numerical model is validated against data available in the literature (Anderson [5]), and once validated, the computational mesh is used to generate additional numerical results. Figures 1 and 2 illustrate the flow at the projectile base for configurations with and without a boat tail.

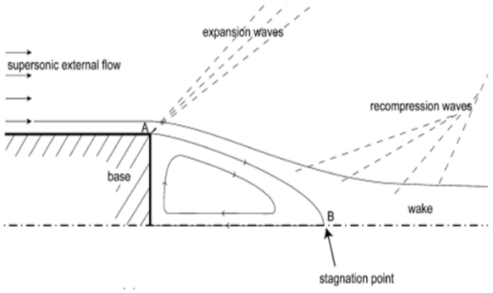


Figure 1. Projectile base flow without boat tail

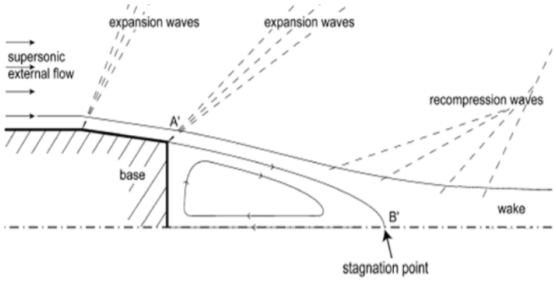


Figure 2. Projectile base flow with boat tail

2 Methodology and Methods

2.1 Mathematical Modeling

Governing Equations

The airflow around artillery projectiles involves relatively high velocities, so it is not possible to disregard the variation in density. Thus, the equations governing the flow are: conservation of mass, conservation of linear momentum, and conservation of energy. These equations are expressed, respectively, as follows:

$$\nabla \cdot (\rho \mathbf{u}) = 0 \ ; \ \nabla \cdot (\rho \mathbf{u}\mathbf{u}) - \nabla \cdot \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T) = -\nabla p \ ; \ \nabla \cdot (\rho \mathbf{u} C_v T) - \nabla \cdot \left(\frac{k}{C_v}\right) \nabla (C_v T) = p \nabla \cdot \mathbf{u} \quad (1)$$

Turbulence Model

To study turbulent flows, Reynolds decomposition is applied to the Navier-Stokes equations, analyzing the mean values of flow properties. To approach this problem computationally, it is necessary to incorporate models designed to estimate the Reynolds stress tensor. In this case, the SST $k - \omega$ model was chosen (Menter, [6]), which is suitable for simulating flows characterized by adverse pressure and velocity gradients and boundary layer separation.

Numerical Method

For the Computational Fluid Dynamics (CFD) analysis, the Finite Volume Method (FVM) was employed to discretize the flow domain. This method subdivides the space into discrete control volumes, forming a computational mesh over the studied region, where the Navier–Stokes equations are solved for each element according to the defined boundary conditions.

The commercial CFD software ANSYS Fluent was used to perform the numerical simulations. A second-order upwind scheme (Setzwein et al., [7]) was applied for the discretization of the convective terms, ensuring improved accuracy of the results. The density-based algorithm (Singh and Mukhopadhyay, [8]) was adopted to solve the system of differential equations iteratively at each computational step, while the multigrid method (Lien and Ryan, [9])was implemented to accelerate convergence.

2.2 Problem Conditions

Computational Mesh and Boundary Conditions

The flow around the projectile is axisymmetric, allowing the problem to be simplified to a two-dimensional model using a longitudinal plane with its origin at the center of the projectile base. The projectile profile used in the simulations is shown in Figure 3. Table 1 illustrates the boat tail lengths that were varied to optimize the projectile design to reduce drag and increase range.

Owing to symmetry about the axis, only the upper quadrant is analyzed. The downstream extent of the computational domain ensures that flow disturbances do not reach the boundaries (Silva et al., [10]). A structured mesh is employed, refined near the projectile surface to accurately capture local gradients. Based on a mesh refinement analysis (Paulo et al,[11]), a configuration with 385,200 control volumes was selected. The refinement

was carried out until the drag coefficient showed no variation in its third decimal place, defining the optimal discretization level. Boundary conditions include Mach numbers ranging from 0.6 to 3.0 at the inlet, atmospheric static pressure at the outlet, and an adiabatic no-slip wall condition on the projectile surface. Along the axis of symmetry, the radial velocity and all radial gradients are set to zero. Gauge pressures are used, with an operating pressure of 101,325 Pa and a constant ambient temperature of 288.15 K. Figure 4 illustrates the computational mesh and the boundary conditions applied in the simulations.

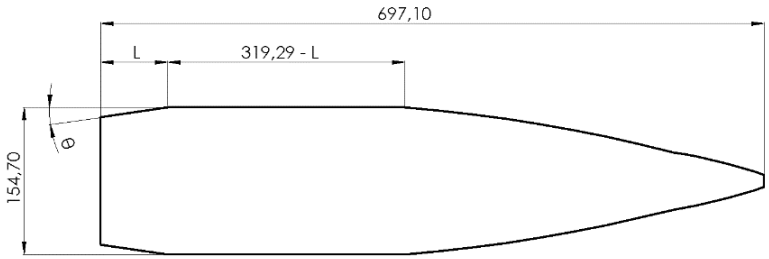


Figure 3. Schematic drawing of the projectile.

Table 1. boat tail length and angle for simulations

L (mm)	θ (degrees)
-	0
70.100	8.5
119.844	5
85.393	7
66.200	9
53.941	11

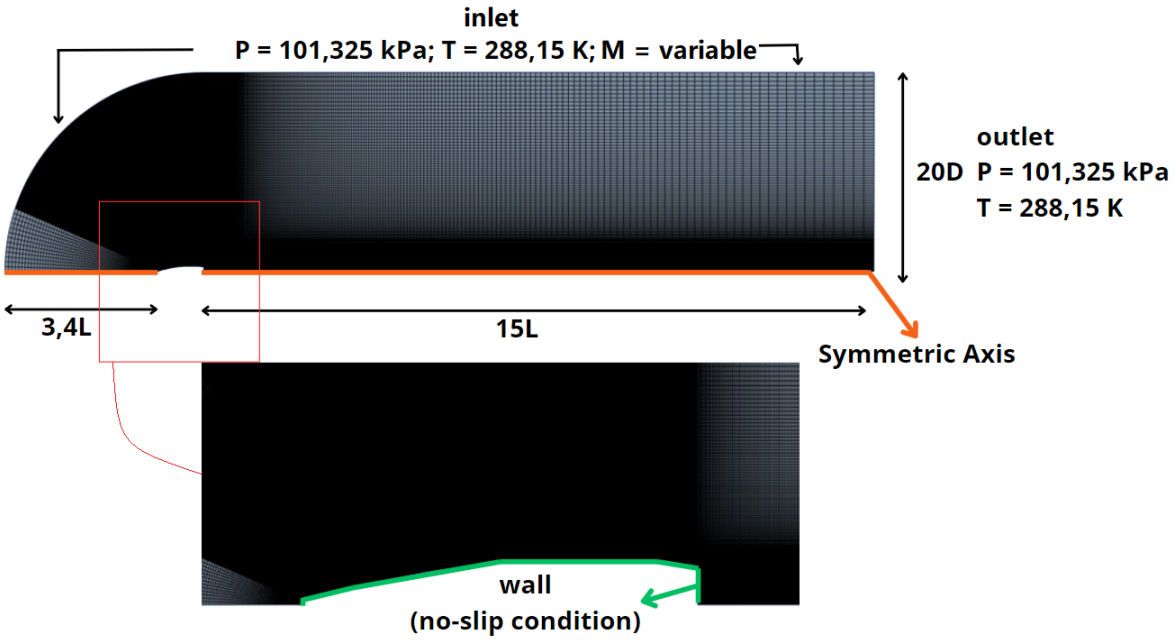


Figure 4. Mesh and boundary conditions used

3 Results and Discussion

Mesh and boundary conditions validation

Based on a mesh refinement analysis (Paulo et al.,[11]), a configuration with 385,200 control volumes was selected. Figure 5 shows the CFD simulations for the 155 mm projectile with an 8.5° boat tail, included for comparison. The drag coefficient is plotted against Mach number and compared with reference data (Suliman et al.,[3]) to validate the mesh. All simulations were considered converged when the residuals of the variables dropped below 10^{-6} .

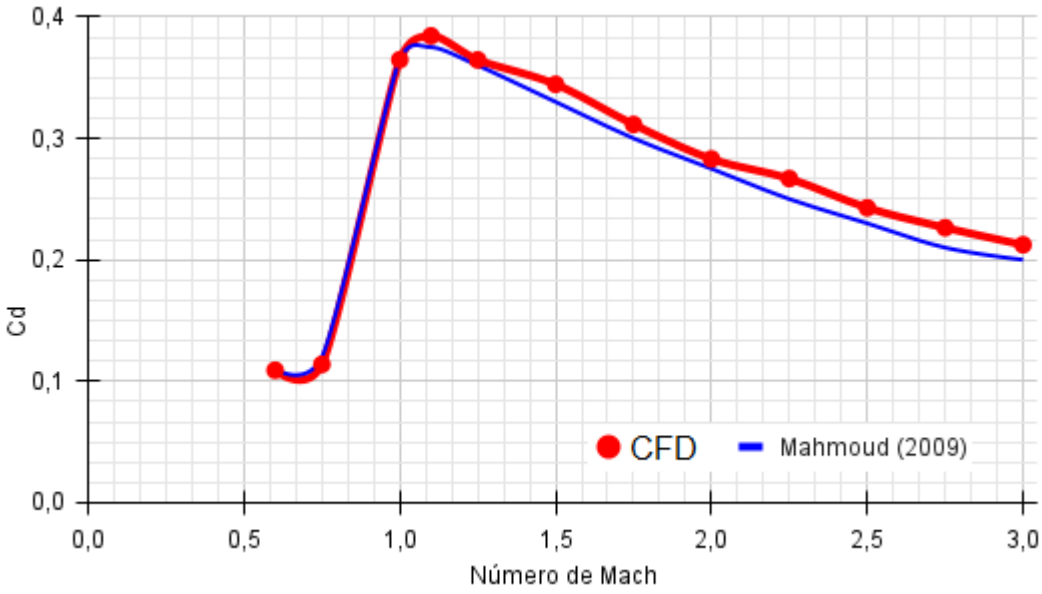


Figure 5. Drag coefficient comparison for the simulated 155 mm projectile and the Suliman et al. [3] results.

Based on this comparison, the maximum percentage error observed was 7.8% at Mach 2.75, while the minimum was 0.08% at Mach 1. These results indicate that the approach closely matches the reference values, showing good agreement with the simulations by (SULIMAN M. A; MAHMOUD, 2009). With the obtained results, it was possible to validate the simulation using the cited reference, thereby enabling the proposed variations in boat tail length to be carried out as intended in this study.

CFD Results

To analyze the impact of varying the boat tail length on the drag coefficient, Computational Fluid Dynamics (CFD) was employed using the FLUENT software. This tool enables detailed modeling and aerodynamic simulation of projectiles, making it suitable for high-speed flow studies. The CFD simulations allowed for the determination of aerodynamic coefficients (Cd) for different boat tail configurations across a range of Mach numbers, covering subsonic to hypersonic regimes.

Angles of 0°, 5°, 7°, 8.5°, 9°, and 11° were tested to evaluate the influence of each configuration on aerodynamic drag. The results showed that the 7° angle was the most efficient, presenting the lowest drag coefficients in the Mach range from 0.6 to 3.0. This configuration demonstrated a significant reduction in drag compared to the other angles tested. The analysis confirmed the effectiveness of the boat tail technique in reducing drag, validating the choice of the 7° angle as the optimized configuration to enhance the range and accuracy of the 155 mm projectile.

Figure 6 illustrates the behavior of the drag coefficient as a function of Mach number based on the CFD results.

Figure 7 illustrates the variation of the drag coefficient as a function of boat tail angle, based on data obtained from FLUENT. The results indicate a trend toward optimal performance at a 7° angle, where the lowest drag coefficient is observed.

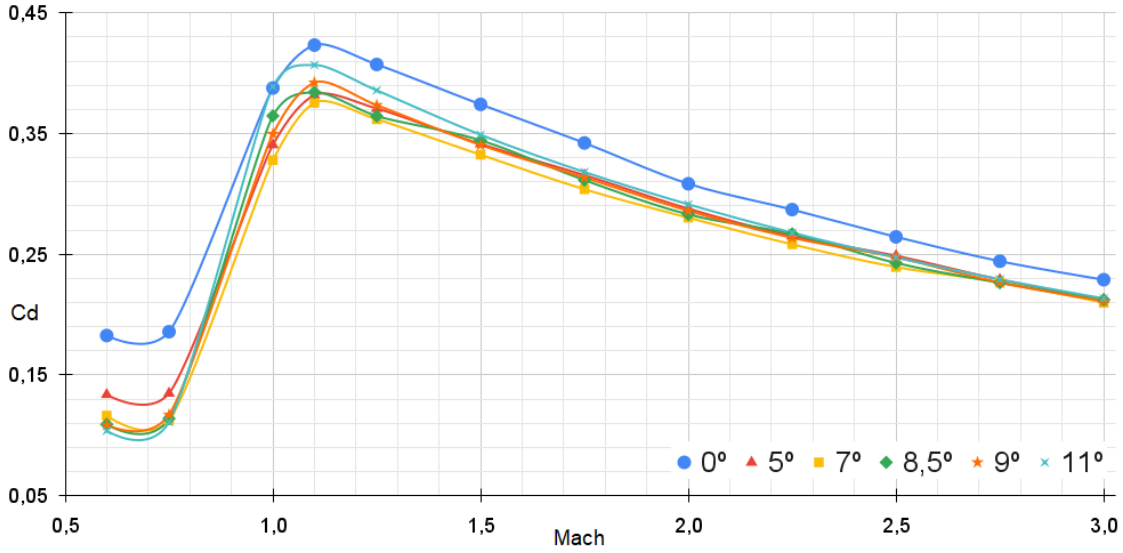


Figure 6. CFD-based drag coefficient as a function of Mach number

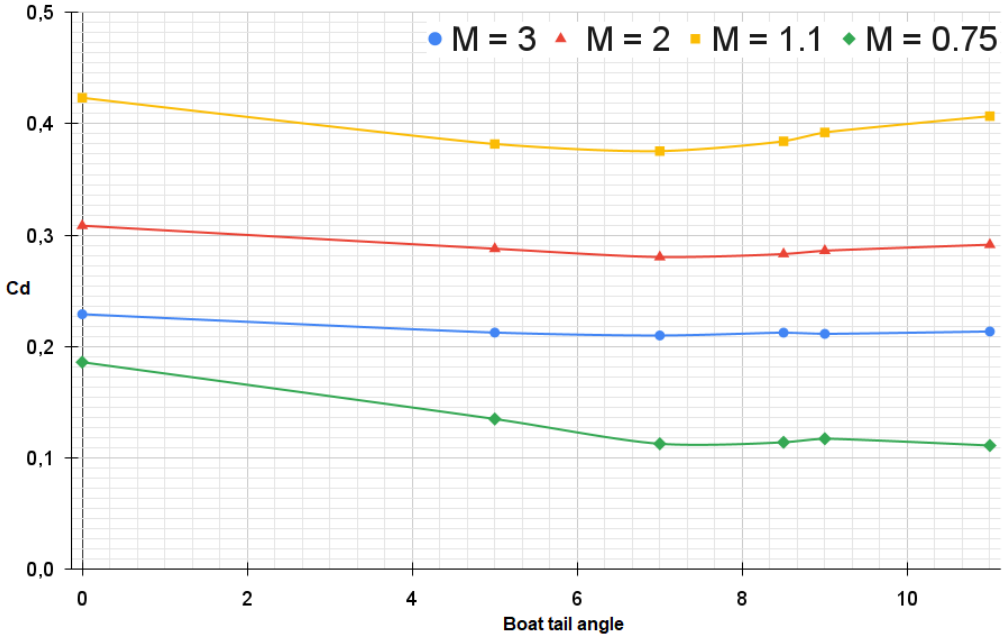


Figure 7. CFD-based drag coefficient vs boat tail angle

Figure 8 presents the pressure contours obtained at Mach 2 for boat-tail angles of 0°, 5°, 7°, 9°, and 11°. As expected, all configurations exhibit a distinct low-pressure region at the projectile base, which is particularly pronounced in the geometry without a boat tail. This phenomenon is consistent with the formation of a separated wake flow downstream of blunt bodies. Among the analyzed cases, the configuration with a 7° boat-tail angle stands out by producing the highest base pressure. This result is in agreement with the numerical predictions discussed earlier and provides further evidence of the stabilizing effect of this geometry on the flow field. The increase in base pressure directly implies a reduction in pressure drag, which is a major component of the overall aerodynamic resistance acting on the projectile. Therefore, the 7° configuration can be considered the most efficient among those evaluated, since it minimizes drag while maintaining stable flow behavior, reinforcing its suitability for applications involving transonic and supersonic regimes.

The most relevant practical implication of these findings is that the correct application of the boat-tail geometry leads to an increase in the projectile's range. By reducing the base drag through a more favorable pressure

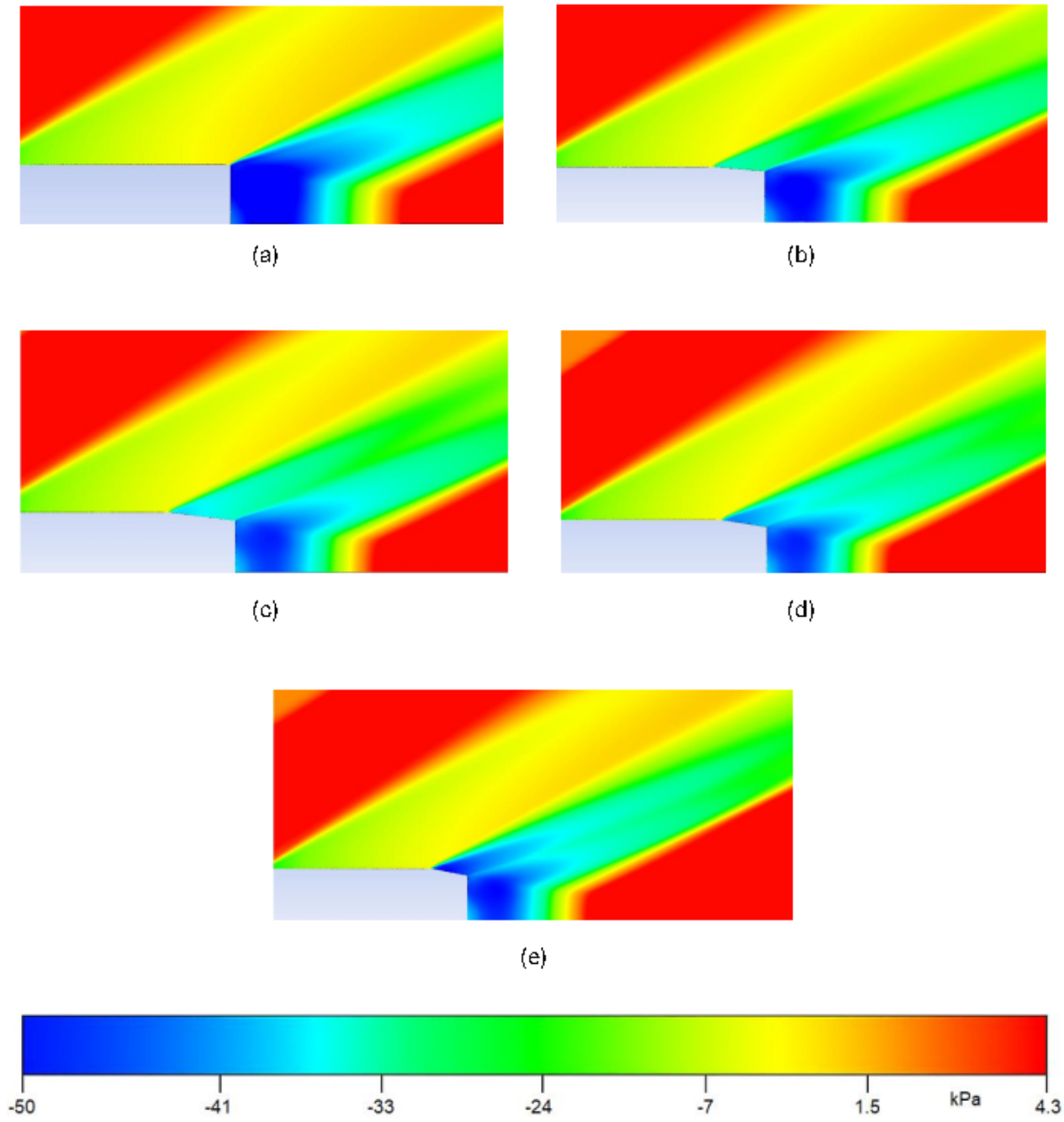


Figure 8. Pressure contours for the configurations: (a) 0°, (b) 5°, (c) 7°, (d) 9°e (e) 11°.

distribution, the aerodynamic efficiency of the projectile is significantly improved. This reduction in drag not only decreases energy losses during flight but also allows the projectile to maintain higher velocities over longer distances, thereby extending its effective range. Consequently, the optimization of the boat-tail angle is not merely an academic exercise, but a crucial design feature with direct impact on the overall performance of long-range ballistic systems (Paulo et al.,[12]).

4 Conclusions

In conclusion, this study provided a detailed numerical analysis of the influence of boat-tail geometry on the aerodynamic behavior of projectiles. The findings confirmed the decisive role of base pressure in determining total drag, demonstrating that relatively small variations in the boat-tail angle can produce significant differences

in aerodynamic performance. Among the configurations evaluated, the 7° geometry yielded the most favorable results, increasing base pressure, reducing drag, and consequently extending the effective range of the projectile. This highlights the importance of correctly applying boat-tail design principles, as improvements in aerodynamic efficiency directly translate into operational advantages in terms of accuracy, stability, and range.

Beyond the technical outcomes, it is important to recognize the broader context in which this work is situated. Research in defense engineering is often strategically sensitive, and consequently, the open scientific literature in this field remains relatively scarce and fragmented. This limited availability of accessible references can hinder the advancement of knowledge and the dissemination of innovative methodologies. By adopting a rigorous computational fluid dynamics approach, validated against established studies, this work contributes not only to a better understanding of boat-tail effects but also to the methodological framework available for future investigations.

Ultimately, the present study seeks to encourage further research in the area of defense-related aerodynamics, promoting dialogue between academic institutions and applied engineering sectors. Expanding investigations of this kind is essential to foster innovation, strengthen technological independence, and support the continuous development of more efficient and reliable systems. In this sense, the results presented here should be regarded not as an endpoint, but as a foundation upon which more comprehensive studies—incorporating experimental validation, multidisciplinary integration, and broader operational scenarios—can be built in the future.

Acknowledgements. The authors would like to thank the Brazilian Army for its institutional support and the Coordination for the Improvement of Higher Education Personnel (CAPES) for the funding and resources provided, which were essential for the development of this research.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] R. Fox, A. McDonald, P. Pritchard, and J. Mitchell. *Introduction to Fluid Mechanics*. Livros Técnicos e Científicos Editora Ltda, Rio de Janeiro, 9th edition, 2018.

[2] M. Dali and S. Jaramaz. Various methods of artillery projectiles base drag reduction. In *International Scientific Conference on Defensive Technology*, 2018.

[3] M. Suliman, O.-S. Mahmoud, M. A.-H., and O.E. Computational investigation of base drag reduction for a projectile at different flight regimes. In *International Conference on Aerospace Sciences and Aviation Technology*, 2009.

[4] The Engineering ToolBox. Engineering toolbox – resources, tools, and basic information for engineering and design of technical applications. `https://www.engineeringtoolbox.com/elevation-speed-sound-air-d_1534.html`. Accessed: 2023-06-20, 2009.

[5] J. Anderson. *Computational Fluid Dynamics: The Basics with Applications*. McGraw-Hill, Inc., 1995.

[6] F. R. Menter. Review of the shear-stress transport turbulence model experience from an industrial perspective. *International Journal of Computational Fluid Dynamics*, vol. 23, n. 4, pp. 305–316, 2009.

[7] E. P. F.. Setzwein and P. Gerlinger. A coupled multigrid solver with wall functions for three-dimensional flows. *AIAA Journal*, vol. 62, n. 12, pp. 4798–4811, 2024.

[8] A. Singh and S. Mukhopadhyay. Comparison of pressure-based and density-based solvers for scramjet modeling. *AIP Advances*, vol. 2584, n. 1, 2023.

[9] F. S. Lien and F. Kuhn. A coupled multigrid solver with wall functions for three-dimensional flows. *International Journal for Numerical Methods in Fluids*, vol. 103, n. 7, pp. 1349–1371, 2023.

[10] W. R. R. Silva, R. A. R. Paulo, V. S. Santiago, and A. L. T. Rezende. Investigation of the aerodynamic benefits of base bleed technology on artillery shells. *Proceedings of Congresso Nacional de Matemática Aplicada e Computaciona - CNMAC 2024*, 2024.

[11] R. A. R. Paulo, V. S. Santiago, and A. L. T. Rezende. Modelagem rans aplicada ao escoamento sobre um projetil 155 mm. *Proceedings of XII Congresso Nacional de Engenharia Mecânica*, 2024a.

[12] R. A. R. Paulo, V. S. Santiago, and A. L. T. Rezende. Numerical analysis of a 155 mm artillery shell range. *Proceedings of the joint XLV Ibero-Latin-American Congress on Computational Methods in Engineering*, 2024b.