

AERODYNAMIC CHARACTERISTICS OF FLOW OVER BOAT-TAIL MODELS AT SUBSONIC AND SUPERSONIC CONDITIONS

Tran The Hung^{1*}, Nguyen Trang Minh², Dao Cong Truong²

Abstract: *In this study, the flow behavior and drag of the axisymmetric model at subsonic and supersonic speeds were investigated by a numerical approach. The numerical results were validated with previous experimental results to determine the model's accuracy. The numerical results showed that the optimal angles reduce from 14° at subsonic conditions to 6° ÷ 8° at supersonic conditions. At the supersonic speeds, shock waves occur at the head and boat-tail of the model, which leads to changes in the pressure distribution and drag of the model. The flow behavior and velocity distribution around the model were investigated and presented in detail in this study.*

Keywords: Aerodynamic drag; Boattail.

1. INTRODUCTION

Due to technical requirements, several flying objects are designed with a blunt base, such as warheads, missiles, and projectiles. The blunt base contributes to reducing the warhead length and facilitating the firing process and movement in the barrel. However, an additional drag, which is often called base drag, is added during flight. At supersonic conditions, studies conducted by Hoerner, Krasnow, and Cummings [1-3] indicated that base drag is an important component, accounting for up to 50% of the total drag. Therefore, reducing base drag plays an important role in the performance and range improvement of blunt-base objects during flight.

A boat-tail added to flying objects is a well-known drag reduction solution. Krasnow [3] proposed the rule of base-drag dependence on the base diameter. However, the accuracy of the formula for the subsonic and supersonic flow was limited. Cummings [2], who conducted numerical simulations for the boat-tail model, indicated that the pressure drag on the boat-tail surface increases while the base drag decreases as the boat-tail angle increases. The optimum boat-tail angle with minimum drag was found at around 7° ÷ 8° at supersonic flow. However, in that study, the $k-\varepsilon$ turbulence model was used for 2D pseudo-symmetric simulation. Therefore, boundary layer characteristics could not be obtained accurately. Other studies [4-6] were also carried out at supersonic conditions, in which some characteristics of flow and drag were presented.

Recently, more attention has been focused on energy saving, research on the boat-tail model at subsonic conditions has been developed. The studies of subsonic flow characteristics around the boat-tail were performed by Mair [7, 8], Mariotti et al. [9, 10], and Tran et al. [11-15]. The subsonic flow characteristics have been investigated. The results have shown significant differences compared to supersonic flow. Three states of flow around the boat-tail include: Fully attached flow at small boat-tail angles, flow with a separation bubble at the moderate boat-tail angles, and fully separated flow at high boat-tail angles. Characteristics of the small bubble occurring on the boat-tail are the main difference compared to supersonic flow. This leads to the appearance of optimal boat-tail angles, which are around 14° ÷ 16°. It can be seen that at supersonic flow, the optimal boat-tail angles are much smaller. However, the previous studies were relatively limited for certain models. In order to understand the aerodynamic flow characteristics, more studies on different models are needed.

However, in most of the previous research, experimental methods were often conducted. Since the experiment is usually limited to measuring equipment, only certain parameters such as flow and pressure distribution over the boat-tail surface are determined. Moreover, the relationship between low-speed and high-speed flows has not been presented. Nowadays, the development of computational technology provides powerful tools for fluid flow analysis. Most of the simulation methods are based on solving the Navier-Stokes equations by the finite element method. Turbulent models have been developed and used for flow analysis. Although numerical methods do not give a single root and depend on the chosen turbulent model, the results are still reliable by a suitable validation method. Numerical methods can be applied to extend the problem from existing experimental results.

In this study, the authors present some numerical results for the boat-tail model at subsonic and supersonic conditions by the numerical method. The numerical results were validated with previous experimental results conducted by the same author. Some results of the supersonic flow are presented.

2. NUMERICAL METHODS AND MODEL

2.1. Model and mesh generation

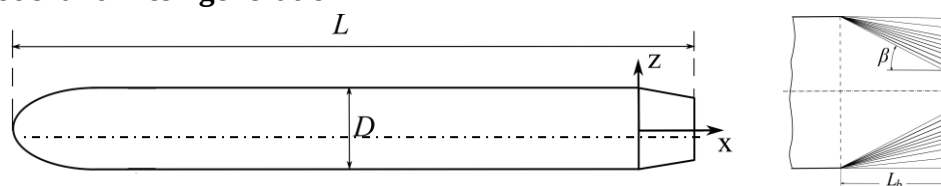


Figure 1. Model geometry.

The axisymmetric models used in this research have a diameter of $D = 30$ mm. The total length of the head and the body is 230 mm. The nose part of the model has an ellipsoid shape with the longest semi-axes of $2D$. The ellipsoid part was designed to prevent the separation on the surface, which affects flow on the boat-tail. The main part of the model has a cylinder part, which allows inserting measurement devices during experiments. The rear part has a conical shape. Consequently, the change of flow at the rear part can be captured easier. The fixed length of the boat-tail is $0.7D$, and the boat-tail angle is adjustable. Note that the above results were widely used in previous studies by Tran et al. [11-13]. Consequently, our results can be compared with the ones in previous studies. The results of the current study, then, can be validated and confirmed. The selection of the standard model also helps to prevent the effect of upstream flow to flow at the boat-tail. Consequently, the flow characteristics of the boat-tail can be highlighted.

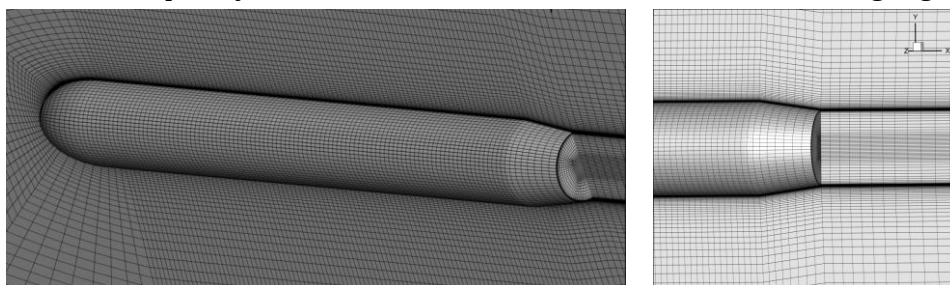


Figure 2. Mesh structure around the model and boat-tail.

The 3-D full domain was structured with dimensions length $\times$ width $\times$ height = $30D \times 5D \times 5D$ for subsonic simulations. However, in supersonic conditions, shock waves occur at the nose and base of the model, so the domain must be expanded in order to capture all shock waves. Thus, for supersonic simulations, the 3-D full domain was structured with higher dimensions length $\times$ width $\times$ height = $66D \times 33D \times 33D$. The effects of the grid size on the solution convergence were investigated. The results have shown that at subsonic conditions, the grid size differences lead the drag to change trivially. The same results have also been shown in the previous research [14]. The structured grid size of 2.8 million cells was selected in this research for time savings. The mesh structure around the model is shown in figure 2.

2.2. Numerical methods

In an aerodynamic simulation of flying objects, the Reynolds-averaged Navier-Stokes (RANS) equations, which solve the averaged equations, are often applied. Since the time derivative is not calculated and the number of variables is reduced, so the calculation speed increases. Obviously, although ignoring the time characteristic reduces the solution accuracy, appropriate validation methods via comparing with experimental results can provide reliable results. When averaging these equations according to Reynolds, there are stress tensors, which are often called Reynolds stresses. These components are often determined by one or two additional transport equations. The one equation, called Spalart-Allmaras turbulent model, allows to save time, but the accuracy is very low. The two-equation model with turbulent kinetic energy k and energy dissipation rate ε or ω can be used. The k - ε model can provide high accuracy results for the model, but the results of the boundary layer are not sufficiently accurate. In fact, Large-eddy simulation can be applied for highly accurate results. But the methods are very expensive and require a lot of numerical time. In this study, the k - ω SST turbulent model was selected to investigate the aerodynamic characteristics of the flow around the research model. This model allows to simulate the boundary layer accurately and reduce the computational time. Simulations were performed by the commercial ANSYS Fluent software. Post-calculation data was processed on specialized software such as Tecplot and Matlab so all results could be displayed clearly. Specifically, the RANS equations are written in the following equations. The first equation describes the law of conservation of mass, the next three equations represent the law of conservation of momentum, and the last equation describes the law of conservation of energy.

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho U) = 0 \quad (1)$$

$$\begin{aligned} & \frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = \\ & = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left(\lambda \nabla \cdot U + 2\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left[\mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[\mu \left(\frac{\partial v}{\partial z} + \frac{\partial u}{\partial x} \right) \right] \\ & \frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = \\ & = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[\mu \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left(\lambda \nabla \cdot U + 2\mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial z} \left[\mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \right] \end{aligned} \quad (2)$$

$$\begin{aligned}
& \frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = \\
& = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left[\mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[\mu \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \right] + \frac{\partial}{\partial z} \left(\lambda \nabla \cdot U + 2\mu \frac{\partial w}{\partial z} \right) \\
& \frac{\partial}{\partial t} \left[\rho \left(e + \frac{U^2}{2} \right) \right] + \nabla \cdot \left[\rho \left(e + \frac{U^2}{2} \right) U \right] = \rho \dot{q} - \frac{\partial}{\partial x} \left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k \frac{\partial T}{\partial y} \right) \\
& + \frac{\partial}{\partial z} \left(k \frac{\partial T}{\partial z} \right) - p \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} \\
& + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z}
\end{aligned} \tag{3}$$

The additional transport equations for k và ω is given:

$$\begin{aligned}
& \frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \\
& \frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \\
& F_2 = \tanh \left[\left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right] \\
& P_k = \min \left(\tau_{ij} \frac{\partial U_i}{\partial x_j}, 10\beta^* k \omega \right) \\
& F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\sigma_{\omega 2} k}{CD_{k\omega} y^2} \right] \right\}^4 \right\} \\
& CD_{k\omega} = \max \left(2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \\
& \phi = \phi_1 F_1 + \phi_2 (1 - F_1)
\end{aligned} \tag{4}$$

where the eddy viscosity ν_t is defined as:

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega; \Omega F_2)} \tag{5}$$

Table 1. The constant parameters of the additional transport equations.

Parameters	α	β	β^*	σ_k	σ_ω
1 st value	5/9	3/40	9/100	0.85	0.5
2 nd value	0.44	0.0828		1	0.856

The constant parameters in equations 4, 5 depending on the flow position near or far from the surface, according to Menter [16, 17]. The values of these parameters are presented in table 1. Boundary conditions of the computational domain are defined in table 2.

Table 2. Boundary conditions of the computational domain.

Conditions	Inlet	Outlet	Model
Subsonic	Velocity Inlet	Pressure outlet	Wall
Supersonic	Pressure far field	Pressure outlet	Wall

The ideal gas was selected for this study. The density of the gas is $\rho = 1.25 \text{ kg/m}^3$, and the input temperature is $T = 300 \text{ K}$. Input flow is considered as uniform gas with a turbulent intensity of 5%. Other parameters are selected as default parameters of the numerical program.

In this study, numerical simulation was conducted at two main Mach numbers of $M = 0.07$ and $M = 2$. The results of low Mach number can be used to compare to experimental results of previous studies. Additionally, it should be noted that the drag of the model does not change much at subsonic flow. Consequently, the low Mach number $M = 0.07$ in this study can be used to refer to the case of subsonic flow. Similarly, at Mach number higher than 2, the flow condition around the base does not change much. As a result, the selection of two cases in this study is significant for boat-tail flow at subsonic and supersonic conditions.

2.3. Validation

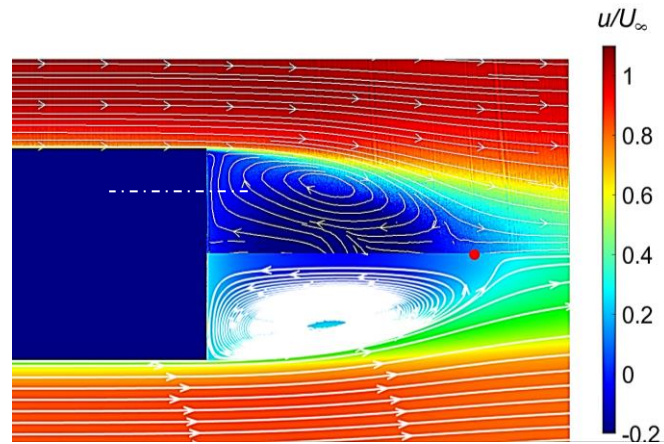


Figure 3. Structure of near-wake flow obtained by experiments (the upper part) and by numerical simulation (the lower part).

The image of the flow around the boat-tail at $\beta = 0^\circ$ and the flow velocity of 10 m/s is shown in figure 3. In which experimental results are presented in the upper half, and the lower half is the simulation result. The experimental results were calculated through the image processing of luminescent smoke particles around the model. Two thousand seven hundred pairs of images were used to simulate the flow around the model. The experimental results were presented in the previous study [11]. The experimental and numerical results are quite similar. It can be seen that the numerical method gives good results and can be used to simulate the afterbody flow.

Figure 4 shows the velocity distribution of the boundary layer at the model base by numerical and experimental methods. It can be seen that the number of grid cells is suitable to simulate the boundary layer. The results calculated by the numerical method

are quite similar to the experimental one. Therefore, the numerical method can be used to calculate and simulate the flow field and drag of the research model.

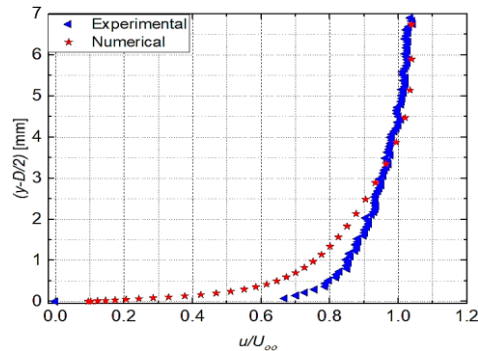


Figure 4. The boundary layer profile at the model base.

3. RESULTS AND DISCUSSION

3.1. Optimal boat-tail angle at subsonic and supersonic conditions

Figure 5 shows the values of the drag coefficient of the models with different boat-tail angles at subsonic ($M = 0.07$) and supersonic conditions ($M = 2.0$). Preliminary calculations show that the optimal drag coefficient does not change at $M < 0.5$. As the experimental results have been collected, this paper focuses on calculations at $M = 0.07$ in order to validate the computational model. Due to time-saving, several certain boat-tail angles have been selected. Experimental and numerical results at subsonic conditions are relatively similar with small errors.

Clearly, at the subsonic flow, the optimal boat-tail angle is between 14° and 16° . These results have been shown by Tran et al. in previous theoretical and experimental studies [11-15]. However, at the supersonic flow, the optimal boat-tail angle is around 6° to 8° . Compared to previous studies for subsonic flow, the optimal boat-tail angle in this study is quite similar.

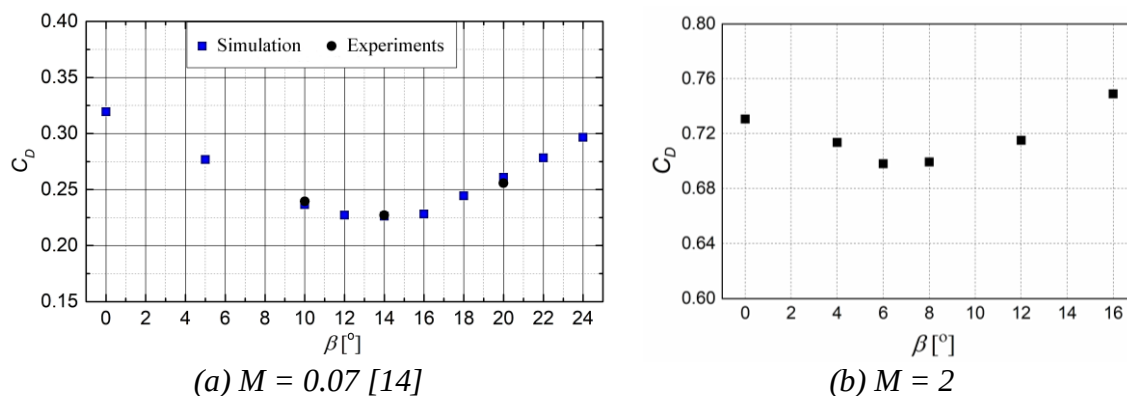


Figure 5. Model drag coefficients at subsonic and supersonic conditions.

3.2. Flow field around the model

Figure 6 shows the velocity distribution at the nose and the body of the model at the supersonic speed $M = 2.0$. At the supersonic condition, shock waves appear around the nose of the model. In addition, a low-pressure area appears at the tip of the model.

However, the separated flow phenomenon does not occur at the tip. The occurrence of a shock wave increases the model drag. In order to reduce drag at supersonic conditions, the nose cone shapes are commonly designed.

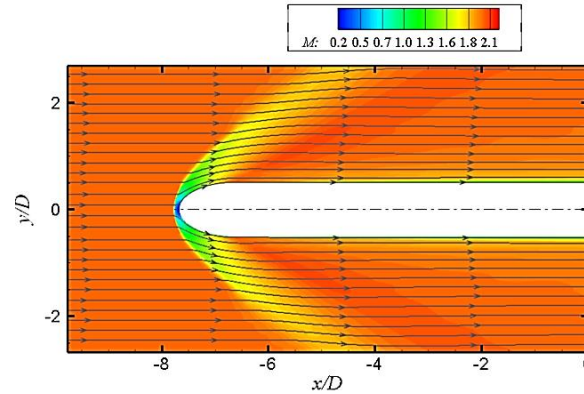


Figure 6. The supersonic flow around the model nose and body ($M = 2.0$).

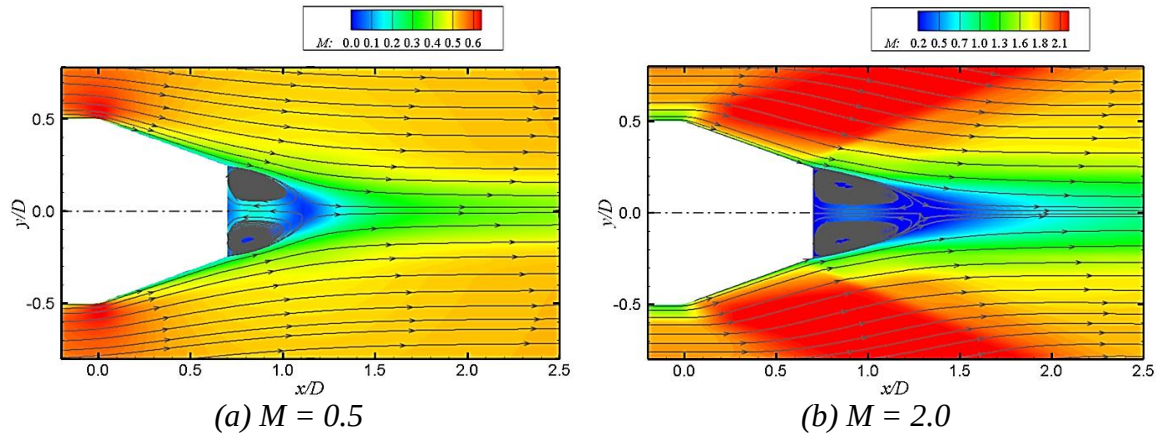


Figure 7. The velocity streamlines and the flow field around the boat-tail at $\beta = 20^\circ$.

Figure 7a shows the velocity and the flow field around the boat-tail at $M = 0.5$, while figure 7b depicts the velocity distribution at $M = 2.0$. The boat-tail angle in these two cases is $\beta = 20^\circ$. The difference in these flows at different Mach numbers can be clearly seen. Specifically, at the supersonic flow, shock waves occur at the junction between the model body and boat-tail. At the subsonic flow, the high velocity appears before the junction. This leads the afterbody near-wake region to be changed. Figure 7b shows that the near-wake region is longer and narrower than that in the subsonic case. This difference can affect the pressure distribution around the boat-tail and the model drag.

3.3. Pressure distribution on the boat-tail and base surface

Figure 8 shows an image of the pressure distribution on the boat-tail surface. The value $x/D = 0$ is at the junction between the model body and the boat-tail. The difference in pressure distribution is presented obviously. For the subsonic flow, higher speed flow at the junction leads to the formation of minimum pressure in this region. However, in the case of the supersonic flow, the pressure is almost unchanged along the boat-tail surface. The area above the boat-tail surface, which is characterized by low-pressure values, leads to an increase in the model drag.

Figure 9 presents the pressure distributions on the model base at $\beta = 0^\circ$ (blunt base). The difference appears at the base edge. At the supersonic condition $M = 2.0$, the pressure changes largely at the model base. Inside the base, the pressure distributions show a similar trend for both cases.

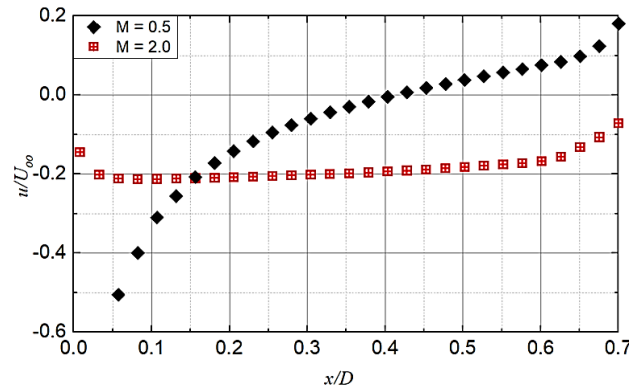


Figure 8. The pressure distributions on the boat-tail surface at $\beta = 20^\circ$.

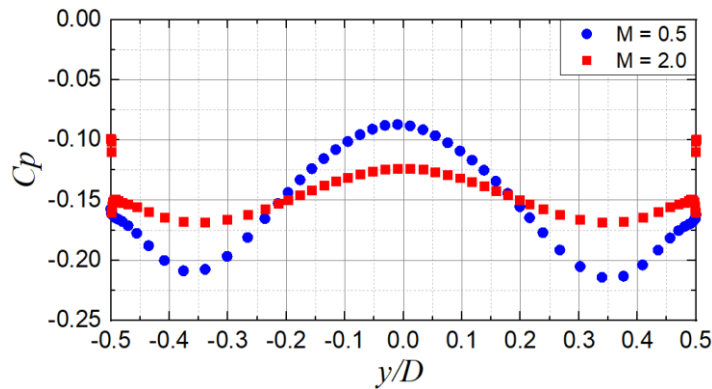


Figure 9. The pressure distributions on the model base at $\beta = 0^\circ$.

Figure 10 shows the pressure distributions on the model base at $\beta = 20^\circ$. D' is the base diameter. Similar to the pressure distribution on the boat-tail surface, the distribution of base pressure remains nearly constant at the supersonic condition. It can be seen that the base drag at the supersonic condition is larger than the subsonic case. The numerical results show that the subsonic and supersonic flows are significantly different.

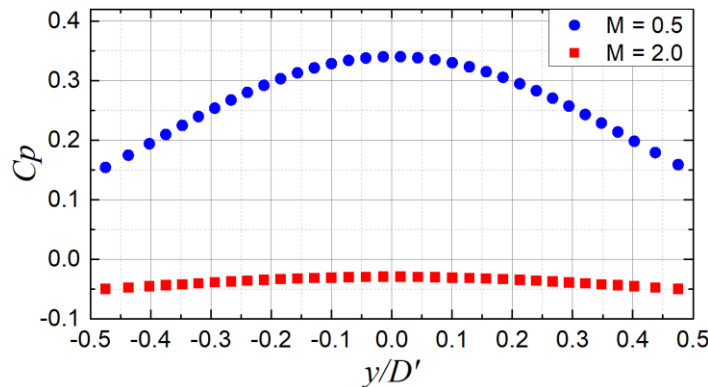


Figure 10. The pressure distributions on the model base at $\beta = 20^\circ$.

4. CONCLUSIONS

In this study, several results of flow behaviors and drag of the axisymmetric model with different boat-tail angles at subsonic and supersonic speeds were investigated by numerical approach. By the selected turbulent model, the research results indicate that the drag increases significantly from subsonic to supersonic speeds. The numerical results show the relative differences in flow behaviors between subsonic and supersonic conditions. At the supersonic flow, shock waves occur at the nose and boat-tail of the model, which leads to the pressure distribution and drag change. Especially, the pressure distributions at the body surface and the base are almost flat. At the subsonic flow, extreme points are at the junction between the model body and the boat-tail. The study also confirms that the optimal angles reduce from 14° at subsonic conditions to $6^\circ \div 8^\circ$ at supersonic conditions for the standard model with a boat-tail length of 0.7 diameters. These results are similar to previous studies of the supersonic flow, but the flow characteristics are analyzed more thoroughly than previous studies. However, due to the limitation of computer resources and time, the number of calculation cases is limited. In the following studies, the computational model selection and the number of computations need to be enhanced. Additionally, numerical simulations for other models with different lengths and angles should be conducted.

REFERENCES

- [1]. S. F. Hoerner, "*Fluid Dynamic Drag*," Bakersfield, USA: Hoerner Fluid Dynamics, (1965).
- [2]. R. M. Cummings, H. T. Yang, Y. H. Oh, "*Supersonic, turbulent flow computation and drag optimization for axisymmetric afterbodies*," *Computers and Fluids*, **Vol. 24**, No 4 (1994), pp. 487-507.
- [3]. N. F. Krasnov, D. N. Morris, "*Aerodynamics of bodies of revolution*," RAND CORP SANTA MONICA CALIF (1970).
- [4]. P. R. Viswanath, "*Flow management techniques for base and afterbody drag reduction*," *Prog. Aerosp. Sci.*, **Vol. 32**, No. 2-3 (1996), pp. 79-129.
- [5]. R. Kumar, P. R. Viswanath, and A. Prabhu, "*Mean and fluctuating pressure field in boat-tail separated flows at transonic speeds*," 39th Aerosp. Sci. Meet. Exhib., No. January (2001).
- [6]. R. Kumar, P. R. Viswanath, and A. Prabhu, "*Mean and fluctuating pressure in boat-tail separated flows at transonic speeds*," *J. Spacecr. Rockets*, **Vol. 39**, No. 3 (2002), pp. 430-438.
- [7]. W. A. Mair, "*Reduction of base drag by boat-tailed afterbodies in low speed flow*," *Aeronaut. Q.*, **Vol. 20** (1969), pp. 307-320.
- [8]. W. A. Mair, "*Drag-reducing techniques for axi-symmetric bluff bodies*," *Aerodynamic drag mechanisms of bluff bodies and road vehicles*, Springer (1978), pp. 161-187.
- [9]. A. Mariotti, G. Buresti, G. Gaggini, and M. V. Salvetti, "*Separation control and drag reduction for boat-tailed axisymmetric bodies through contoured transverse grooves*," *J. Fluid Mech.*, **Vol. 832** (2017), pp. 514-549.
- [10]. A. Mariotti, G. Buresti, and M. V. Salvetti, "*Drag reduction of boat-tailed bluff bodies through transverse grooves*," *ERCFTAC*, **Vol. 25** (2019), pp 489-495.
- [11]. T. H. Tran, T. Ambo, T. Lee., Y. Ozawa, L. Chen, T. Nonomura, K. Asai, "*Effect of Reynolds number on flow behavior and pressure drag of axisymmetric conical boat-tails at low speeds*," *Experiments in Fluids*, **Vol. 60**, No. 3 (2019), pp. 1-19.
- [12]. T. H. Tran, T. Ambo, T. Lee, L. Chen, T. Nonomura, K. Asai, "*Effect of boat-tail angles on the flow pattern on an axisymmetric afterbody surface at low speed*," *Experimental Thermal and Fluid Science*, Vol.99 (2018), pp.324-335.

- [13]. T. H. Tran, T. Ambo, L. Chen, T. Nonomura, and K. Asai, "Effect of boat-tail angle on pressure distribution and drag of axisymmetric afterbodies under low-speed conditions," *Transactions of the Japan Society for Aeronautical and Space Sciences*, **Vol. 62**, No. 4 (2019), pp. 219-226.
- [14]. T. H. Tran, H. Q. Dinh, H. Q. Chu, V. Q. Duong, C. Pham and V.M. Do, "Effect of boat-tail angle on near-wake flow and drag of axisymmetric models: A numerical approach," *Journal of Mechanical Science and Technology*, Vol.35, No.2 (2021), pp. 563-573.
- [15]. T. T. Hung, N. T. Minh, D. C. Truong, "Effect of boat-tail geometry on flow structure and drag of axisymmetric body," *Journal of Military Science and Technology*, **No 72** (2021), pp. 136-142.
- [16]. F. R. Menter, "Zonal two equation $k-\omega$ turbulence models for aerodynamic flows," AIAA-93-2906 (1993).
- [17]. F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, **Vol. 32**, No. 8 (1994), pp. 1598-1605.

TÓM TẮT

ĐẶC TRƯNG KHÍ ĐỘNG CỦA DÒNG CHẢY XUNG QUANH ĐUÔI VÁT TẠI VẬN TỐC DƯỚI VÀ TRÊN ÂM

Trong nghiên cứu này, dòng chảy và lực cản của mô hình vật đối xứng được phân tích cho trường hợp dưới và trên âm. Phương pháp mô phỏng số được thực hiện. Kết quả mô phỏng được kiểm chứng với thực nghiệm nhằm khẳng định tính chính xác của mô hình. Kết quả nghiên cứu chỉ ra rằng, góc tối ưu thay đổi từ 14° về $6^\circ \div 8^\circ$ khi vận tốc dòng thay đổi từ dưới tới trên âm. Tại dòng trên âm, sóng xung kích xuất hiện tại mũi và đuôi của vật. Điều này dẫn tới sự thay đổi phân bố áp suất và lực cản của vật. Các đặc trưng về dòng chảy và phân bố vận tốc quanh mô hình được trình bày và thảo luận trong nghiên cứu này.

Từ khóa: Lực cản; Đuôi hình côn.

Received 25th August 2021

Revised 29th September 2021

Published 11th November 2021

Author affiliations:

¹Faculty of Aerospace Engineering, Le Quy Don Technical University;

²Academy of Military Science and Technology.

*Corresponding author: thehungmfti@gmail.com.