Flow Structure on Boattail Surface by Numerical Simulation

The Hung Tran

January 20, 2022

Flow structure on boattail surface by numerical simulation

Tran The Hung
[1]*Đại học kỹ thuật Lê Quý Đôn, Hà Nội*
Email: thehungmfti@gmail.com

Abstract: *In this study, a numerical method was applied to analyze flow on axisymmetric boattail surface at low-speed conditions. Different boattail angles and lengths were tested. Numerical results showed close to experimental observation for the similar configuration. Drag was shown minimum value at boattail angle around 14° at low-speed conditions. Effect of boattail angle, boattail length on drag of the model, pressure distribution and flow behavior on boattail surface was analyzed and discussed in details.*

Keywords: *boattail model, RANS, drag reduction, skin friction*

1. Introduction

An axisymmetric blunt-base model is characterized by a large separation behind the base. The separation is known as near-wake flow and is one of the most complicated parts of flow fields. It is the main source of drag and low stability [1], [2]. In fact, the wake flow was studies widely in fluid dynamics for both two dimensional and three dimensional cases. However, by comparison to the case of two dimensions, flow around the base of axisymmetric model is much more complicated [3].

Reducing drag to improve aerodynamic performance is designed for control strategy. The drag control device can be divided into active and passive techniques. The active technique controls the wake flow by adding some blowing or suction devices to the surface, which requires an external energy source. In the opposite side, passive control devices change the flow fields by modification of base geometry. Consequently, the boundary layer separation is delayed. The passive control devices include: base cavity, lock-vortex afterbody, step afterbody boattail and groove cavity. It should be noted that since flow at blunt base of the axisymmetric model is much different to the case of two dimensions, some passive control devices work well for two dimensional flow does not work for three dimensional one [4]. Among of those device for drag reduction, boattail should a high effectiveness. In fact, the boattail for drag reduction has been studied widely in previous studies by Tran et al. using both experiments as well as numerical methods [5]–[9].

In this study, a variety conical boattail models with different length and angle will be investigated to understand the effect of boattail length on drag reduction. The work was conducted by numerical method, which is based on Reynolds-averaged Navier-Stokes equations. We found that drag trend with different boattail angle is slightly changed for different boattail length in the range of 0.5 to 1.0 diameter of the model. The paper is organized as followed: Section 2 presents the numerical method; Section 3 three discusses about the results of the study; Finally, the study is concluded in section 4.

2. Numerical methods

The axisymmetric model using in this study has a diameter of D = 30 mm and the ratio length to diameter around L/D= 7.6 (Figure 1). Boattail length can be changed from 0.5 to 1.0D and four angles of 10, 14, 18 and 20° was tested. Parameter of the boattail is shown in Table 1.

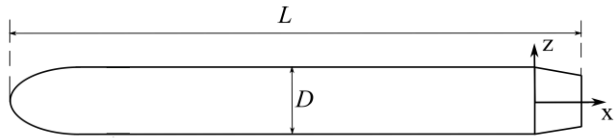

Figure 1. Model geometry

Table 1. Boattail geometry

Parameter	
Length (D)	0.5, 0.7, 1.0
Angle (°)	10, 14, 18, 20

In this study, RANS is applied for numerical process. Although RANS has some limitation, it can save time. In fact, RANS methods are still widely used in current studies [9], [10]. k-ω SST model was selected to obtain averaged flow fields around the model. We use commercial software ANSYS Fluent, which was copyrighted by Faculty of Aerospace, Le Quy Don Technical University, for numerical process. Mesh with 4.8 million cells is selected for this study to obtain high accurate results. The inlet velocity is fixed at 22 m/s, which was similar to previous experimental study by Tran et al [4].

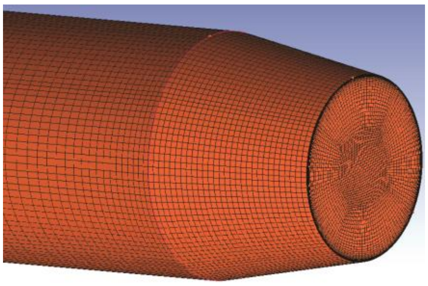

Figure 2. Mesh around the model

3. Results and discussions

Firstly, boundary layer near the separation flow is examined. Here, we used experimental results in previous publication by Tran et al. [6] to show the ability of the numerical method in simulation flow around the boattail surface. Figure 3 shows boundary layer profiles before the shoulder for both numerical and experimental methods. Clearly, the numerical results show closed results to experimental method, which indicates that the mesh generation and numerical scheme are sufficient for the study.

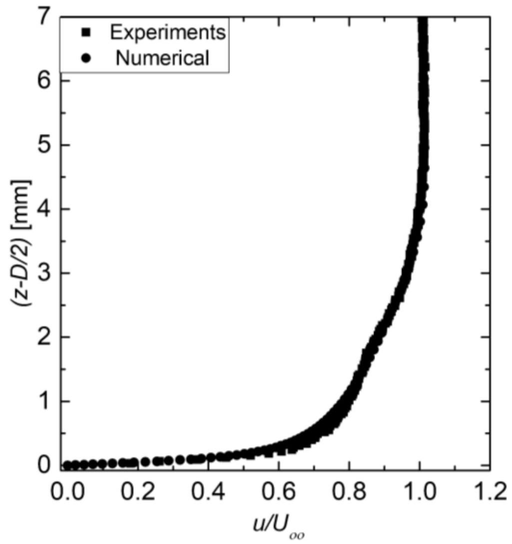

Figure 3. Boundary layer profiles for case of β = 16°

Table 1 shows boundary layer parameters by numerical approach, which include displacement thickness δ^*, momentum thickness θ and H-factor. The H-factor is around 1.6, which indicates that the boundary layer closed to the conjunction is fully turbulent.

Table 1 Characteristics of boundary layer

Parameters	Value
δ_{99}/D	0.227
δ^*/D	0.0490
θ/D	0.0357
H-factor	1.60

Figure 4 shows drag of the model with different boattail configurations. There is existence of minimum drag at boattail angle of around 14° for fixed boattail length. It is explained by the increasing of boattail pressure drag, which occurs from low pressure region near the shoulder [4]. Additionally, separation flow may occur at high boattail angle resulting in redistribution of pressure on the surface. Clearly, increasing boattail does not always have positive effect on drag reduction of the model.

For the fixed boattail angle, increasing length leads to reduction of aerodynamic drag. It can be explained by the quick recovery of pressure on the base surface. Additionally, the wake structure becomes smaller with increasing boattail length.

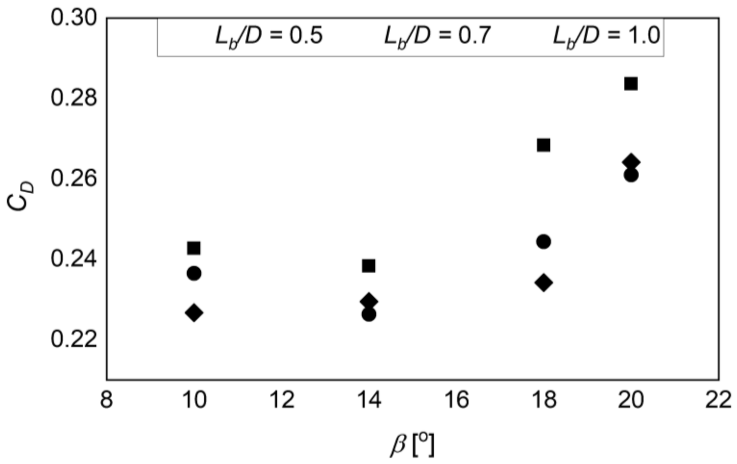

Figure 4. Drag of the model at different boattail configuration

Figure 5 shows pressure distribution on the boattail surface for different length at fixed angle of β = 20°. Clearly, for short boattail length, the near-wake structure affects flow behvior on the boattail surface. Consequently, pressure at L_b/D = 0.5 is slightly difference from two other cases. At L_b/D > 0.5, pressure around the shoulder is slightly changed for different length. Consequently, we can use pressure trend with the maximum length to predict the trend at smaller length. The results are similar to observation by Mair [2], who investigated pressure distribution on similar boattail model.

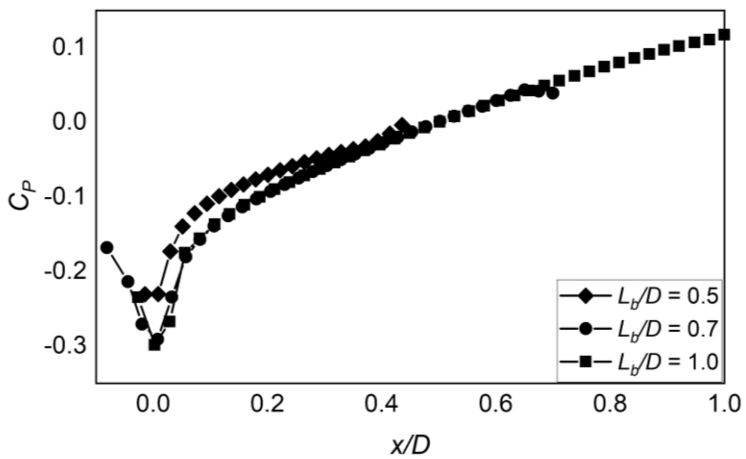

Figure 5. Pressure distribution on boattail surface at different length

Figure 6 presents streamwise skin friction distribution on boattail suface for different angle at L_b/D = 1.0. The distribution of skin friction allow us to analyzing flow behavior on the surface. Here, separation is determined by skin-friction crossing the horizontal axis to become negative while reattachmend is determine by the value changing to possitive.

Clearly, boattail model of 10° is featured by attached flow, where skin-friction value is posstive on the whole surface. For boattail models of 14° and 18°, a separation bubble occurs on the surface featured by negative region of skin friction. However, at boattail model higher than 18°, the whole boattail model is inside the reversed flow region. It is other factor, which can explain for high drag at boattail model above 18°.

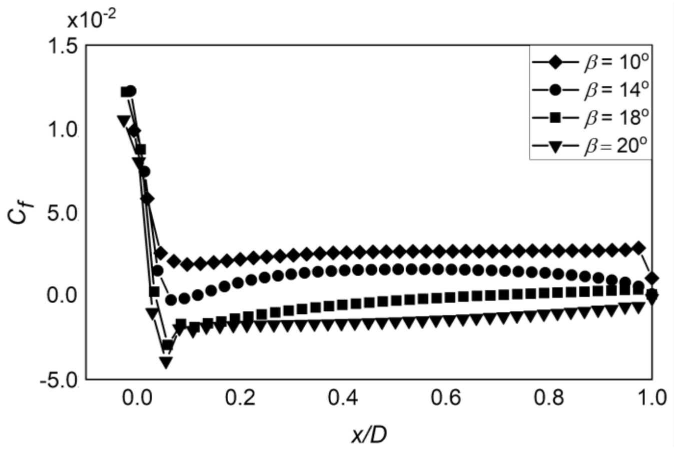

Figure 6. Skin-friction distribution on the surface

4. Conclusion

In this study, a numerical method was presented for analyzing the flow behavior around axisymmetric boattail model. Different boattail length and angle were tested. The numerical results were close to experimental data by previous study. The results showed that trend of drag at fixed boattail angle is similar for different boattail angles. The boattail model with minimum drag is around 14° at low-speed condition. Additionally, at $L_b/D > 0.5$, pressure around the shoulder is slightly changed for different length. Finally, the at boattail angle below 14°, flow on boattail surface is fully attached, while at angle above 18°, flow on the boattail is fully separated.

References

[1] G. Rigas, A. R. Oxlade, A. S. Morgans, and J. F. Morrison, "Low-dimensional dynamics of a turbulent axisymmetric wake," *J. Fluid Mech.*, vol. 755, pp. R51–R511, 2014, doi: 10.1017/jfm.2014.449.

[2] T. H. Tran and L. Chen, "Optical-Flow Algorithm for Near-Wake Analysis of Axisymmetric Blunt-Based Body at Low-Speed Conditions," *J. Fluids Eng.*, vol. 142, no. 11, pp. 1–10, 2020, doi: 10.1115/1.4048145.

[3] W. A. Mair, "The effect of a rear-mounted disc on the drag of a blunt-based body of revolution," *Aeronaut Quart*, vol. 16, 1965.

[4] P. R. Viswanath, "Flow management techniques for base and afterbody drag reduction," *Prog. Aerosp. Sci.*, vol. 32, no. 2–3, pp. 79–129, 1996, doi: 10.1016/0376-0421(95)00003-8.

[5] T. H. Tran, T. Ambo, T. Lee, L. Chen, T. Nonomura, and K. Asai, "Effect of boattail angles on the flow pattern on an axisymmetric afterbody surface at low speed," *Exp. Therm. Fluid Sci.*, vol. 99, no. May, pp. 324–335, 2018, doi: 10.1016/j.expthermflusci.2018.07.034.

[6] T. H. Tran *et al.*, "Effect of Reynolds number on flow behavior and pressure drag of axisymmetric conical boattails at low speeds," *Exp. Fluids*, vol. 60, no. 3, pp. 1–19, 2019, doi: 10.1007/s00348-019-2680-y.

[7] T. H. Tran, T. Ambo, L. Chen, T. Nonomura, and K. Asai, "Effect of boattail angle on pressure distribution and drag of axisymmetric afterbodies under low-speed conditions," *Trans. Jpn. Soc. Aeronaut. Space Sci.*, vol. 62, no. 4, pp. 219–226, 2019, doi: 10.2322/tjsass.62.219.

[8] T. H. Tran, "The Effect of Boattail Angles on the Near-Wake Structure of Axisymmetric Afterbody Models at Low-Speed Condition," *Int. J. Aerosp. Eng.*, vol. 2020, 2020, doi: 10.1155/2020/7580174.

[9] T. H. Tran, H. Q. Dinh, H. Q. Chu, V. Q. Duong, C. Pham, and V. M. Do, "Effect of boattail angle on near-wake flow and drag of axisymmetric models: a numerical approach," *J. Mech. Sci. Technol.*, vol. 35, no. 2, pp. 563–573, Feb. 2021, doi: 10.1007/s12206-021-0115-1.

[10] A. D. Le, H. Phan Thanh, and H. Tran The, "Assessment of a Homogeneous Model for Simulating a Cavitating Flow in Water Under a Wide Range of Temperatures," *J. Fluids Eng.*, vol. 143, no. 10, p. 101204, 2021, doi: 10.1115/1.4051078.