Numerical Investigation of Asymmetric Separation Vortices over Slender Body by RANS/LES Hybrid Simulation

By Ryoji INABA¹⁾, Hiroyuki NISHIDA¹⁾, Taku NONOMURA²⁾, Kengo ASADA³⁾ and Kozo FUJII²⁾

¹⁾Department of Mechanical Systems Engineering, Tokyo University of Agriculture and Technology, Tokyo, Japan

²⁾The Institute of Space and Astronautical Science, JAXA, Sagami-hara, Japan

³⁾Department of Aeronautics and Astronautics, The University of Tokyo, Tokyo, Japan

(Received June 27th, 2011)

We analyze the asymmetric vortices in the flow over a slender body at high angle of attack by numerical simulations aiming a proportional control of the side forces generated by vortices with a device such as dielectric barrier discharge (DBD) plasma actuator. With regard to the computational method, Reynolds averaged Navier Stokes/large-eddy simulation hybrid method is adopted with high-order compact spatial difference scheme. The grid convergence analysis is firstly conducted and the results show that the computational grid adopted in this study is fine enough for qualitative discussion. The total number of the grid point is 411 million points. Then, the effects of bump height on flow fields and aerodynamic characteristics are discussed. Note that bump is added near the body apex to simulate the symmetry-breaking imperfection. As a higher bump is adopted, stronger asymmetry is observed in the flow fields. On the other hand, side-force has nonlinearity with the bump height.

Key Words: Flow Control, Plasma Actuator, Slender Body

Nomenclature

L	: Length of body
R_N	: Curvature radius of cone apex
θ_{cone}	: Half apex angle of cone
D	: Base diameter of body
α	: Angle of attack
θ_c	: Body apex angle
u	: Velocity vector
q	: Heat flux vector
ρ	: Density
p	: Static pressure
e	: Total energy per unit value
τ	: Stress tensor
δ	: Kronecker's delta
Re	: Reynolds number based on base diameter
Pr	: Prandtl number
M	: Mach number
μ	: Viscosity
C_y	: Side force coefficient
C_{y_s}	: Sectional side force coefficient
x	: Cartesian coordinate

Subscripts

∞	: Free stream quantity
ref	: Reference quantity

1. Introduction

A slender body is a typical body shape of a rocket vehicle and a fore-body of an aircraft. It is generally known that the asymmetric vortices occurred over a slender body when the

angle of attack becomes high.¹⁾ This asymmetrically separated vortices produce side force and yawing moment acting on the body, leading to unstable attitude of a vehicle. Therefore, to control the side force and yawing moment can be one of the important research themes. Many studies^{2,3)} have been conducted to clarify the generation mechanism of the asymmetric separation vortices and understanding the structure and transport characteristics of the vortices. These previous researches aim at suppression and control of the asymmetric vortices, i.e. the side force using flow control devices. This is because conventional attitude control devices such as a thruster and a rudder are not efficient and not effective to overcome the side force; high-power thruster is needed due to the large side force and the rudder in the wake flow does not work well. Active flow control is one of the effective solutions, and several active flow control devices such as a micro jet and a small moving flap have been proposed and studied^{4,5)}; however, no active flow control device has been put into practical use due to lack of comprehension of its mechanism or difficulty in providing high pressure working fluids.

In recent years, a control method using the dielectric barrier discharge (DBD) plasma actuator gets much attention as the more effective method.^{6,7)} The DBD plasma actuator has a quite simple configuration as shown in Fig. 1. The DBD plasma actuator consists of two electrodes separated by a dielectric and displaced in the stream wise direction. The momentum is added to the boundary layer by setting the actuator on the body surface, and the flow separation can be suppressed. This device has a several advantages such as a fully electronic with no moving parts, extremely fast response, extremely thin and light-weight. There are some previous researches^{8,9)} in which the DBD plasma actuator is applied to

the active control of the asymmetric vortices over a slender body; the DBD plasma actuator has been installed to the tip of the slender body and only bang-bang control of the side force has been accomplished. The DBD plasma actuator can be easily installed to anywhere on the body surface and it is expected that the proportional control of the side force will be accomplished by multipoint installations of the actuators. However, the numerical simulations of the flow field over the slender body with the DBD plasma actuator has never studied so far.

We attempt to verify the concept of proportional control of the side force using numerical simulations based on the computational fluid dynamics (CFD). In order to do this, it is necessary to clarify the generation mechanism and transport characteristics of the vortices. The purpose of this paper is not to quantitatively be in agreement with the experimental result but to investigate the qualitative flow field over the slender body at high angle of attack. In this paper, following researches are conducted for the first step of our study.

- Numerical simulations are conducted varying the grid resolution and the height of the bump, which is an asymmetry-breaking disturbance attached to the body apex. Our numerical simulation is validated by comparison with the experimental results reported in previous works, and technical know-how for simulating the asymmetric separation flow over slender body is accumulated.
- The computational study on the asymmetric vortices over a slender body is conducted toward understanding the characteristics of asymmetric vortices to change the bump height and angle of attack.

Fig. 1. DBD Plasma actuator.

2. Characteristics of Flow Field over Slender Body at High Angle of Attack

Characteristics of the flow field over a slender body at high angle of attack can be classified into four groups by the apex angle of body and the angle of attack¹⁾ (as show in Fig. 2).

- (1) The angle of attack is sufficient low, and the axial flow is much stronger than the cross-flow. The flow separation does not occur.
- (2) The cross-flow becomes stronger with higher angle of attack, and the cross-flow separates at the side surface. The vortex filaments are shed from the side surface of the apex and are transported to the downstream along the body surface. The vortex structure remains to be symmetric.
- (3) When the angle of attack exceeds the threshold value, the vortex filaments separate from the body surface and the vortex structure becomes asymmetric. The asymmetric vortices exert strong side force on the body.

- (4) The flow field is similar to that over a cylinder, and the vortex is periodically shed.

In Fig. 2, the relations of the body apex angle θ_c and each angle of attack α are $\alpha_{SV}=1.1\sim 1.3\theta_c$, $\alpha_{AV}=2\theta_c$ and $\alpha_{UV}=70\sim 75$ degrees.¹⁾

Fig. 2. Classifications of the flow around the slender body.

3. Computational Models

3.1. Governing equations

In this study, the three-dimensional compressible Navier-Stokes equations non-dimensionalized by cylinder diameter D are employed as the governing equations. They consist of the mass, the momentum and the energy conservation laws. In the non-dimensional form, governing equations are represented as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_k}{\partial x_k} = 0, \quad (1)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_k + p \delta_{ik})}{\partial x_k} = \frac{1}{Re} \frac{\partial \tau_{ik}}{\partial x_k}, \quad (2)$$

$$\frac{\partial e}{\partial t} + \frac{\partial ((e+p)u_k)}{\partial x_k} = \frac{1}{Re} \frac{\partial u_i \tau_{ik}}{\partial x_k} + \frac{1}{(\gamma-1)Pr Re M_\infty^2} \frac{\partial q_k}{\partial x_k}. \quad (3)$$

Three basic non-dimensional number Re , Pr , and M_∞ denote the Reynolds number, the Prandtl Number and the free stream Mach number respectively:

$$Re = \frac{\rho_\infty u_\infty D}{\mu_\infty}, \quad M_\infty = \frac{u_\infty}{a_\infty}, \quad Pr = \frac{\mu_\infty C_p}{k_\infty}. \quad (4)$$

3.2. Computational schemes

LES/RANS Hybrid method is employed in this study. Because, it is considered that small vortex structures in the region near the body play significant role to decide the overall flow field. A Reynolds-averaged-Navier-Stokes (RANS) model is applied to the region near the body, whereas large-eddy simulation (LES) is applied the region away from the body. The Baldwin-Lomax turbulence model is used for the RANS computation. In standard LES approach, additional stress and heat flux terms are appended, but in implicit large eddy simulation (ILES) approach¹⁰⁾ they are not appended.

Instead, a high-order low-pass filter selectively damps only the poorly resolved high-frequency waves. In this study, ILES

approach is adopted. This filtering regularization procedure provides an attractive method to the use of standard sub-grid-scale (SGS) models.

With regard to the spatial differencing, sixth-order compact scheme is employed, while sixth-order tri-diagonal filter is together used for numerical stability.^{11,12} For time integration, a kind of implicit method Lower-Upper Symmetric Alternating Direction Implicit and Symmetric Gauss-Seidel (ADI-SGS)^{13,14} is used. This algorithm uses same kind of idea of Four-Factored Symmetric Gauss-Seidel (FFSGS)¹⁵ which adopts both ideas of the Lower-Upper Symmetric Alternating Direction Implicit (LU-ADI) and the Lower-Upper Symmetric Gauss-Seidel (LU-SGS). To ensure the time accuracy, backward second-order difference formula is used for time integration whereas three sub-iterations¹⁶ are adopted.

4. Computational Setup

4.1. Flow conditions

Free stream Mach number is $M_\infty=0.3$. This freestream Mach number is slightly higher than that in the experiment by Nishida et al.¹⁷ However, the value is low enough that the compressibility of fluid is almost negligible. Therefore, the flow field obtained by our simulation is comparable to that obtained by the experiment. Other flow conditions are experimental condition by Nishida et al.¹⁷ Reynolds number based on the cylinder diameter is $Re=3.0 \times 10^4$. The Prandtl number set to be 0.72.

4.2. Computational models

In this study, a slender body shape and computational conditions in the simulation are the same as those in the experiment by Nishida et al. for comparison with the experimental results.¹⁷ The slender body shape considered in the simulation is like a pencil, which consists of a cone and a cylinder (height ratio 1:1). The detailed dimension and shape are shown in Fig. 3 and Table 1. The body apex angle is 18 degrees, and therefore, according to the discussion in the Sec. 2, the asymmetric vortices appear over the body at the angle of attack of from 38 to 70 degrees.

Fig. 3. Computational model.

Table 1. Dimension of the slender body.

L	0.421[m]
L_{cone}	0.211[m]
$L_{cylinder}$	0.210[m]
D	0.072[m]
R_N	0.0028[m]
θ_{cone}	9[deg]

4.3. Computational grid

Three different grid resolution types of computational grid are generated to clarify grid resolution dependency. Generated computational grid called “Fine grid”, “Medium grid”, and

“Coarse grid” in order of high resolution. The grid point of Fine grid is $\sqrt{2}$ times that of Medium grid each direction. On the other hand, Grid points of Coarse grid is $1/\sqrt{2}$ times that of Medium grid each direction.

Figure 4 shows the Fine grid. The fine computational grid consists of 219 axial (j) points, 186 circumferential (k) points, and 101 normal (l) points. The minimum grid spacing of body surface in the laminar boundary layer needs to take smaller than $y_{min} = 0.08/\sqrt{Re}$.

Fig. 4. LES/RANS Hybrid computational grid.

4.4. Geometrical disturbance (bump)

It has been reported in previous studies^{18,19} that a symmetry-breaking imperfection is needed to make the separation vortices asymmetry. In the actual flight condition or the wind tunnel experiment, the freestream turbulence and imperfection of the body shape break the symmetry. To simulate the symmetry-breaking imperfection geometrical disturbance (bump) is added near the body apex according to the reference.¹⁸ This disturbance represents various disturbances such as the small surface roughness on experimental models and freestream turbulence. Different four height bumps are shown in Fig. 5. These four shapes are generated to find out the effect of the bump height. The height of the bump is the only varying parameter. The height of the bump ranges from $0.033D$ to $0.01D$, the length of the bump was $0.05D$ and it was located at 90 degrees circumferentially from the windward meridian.

Fig. 5. Four types of the bump at the body apex.

5. Validation Study of the Computational Methods

First, we investigate the dependency of the solution on the computational setup and validate the numerical technique by comparing with the previously reported experimental results.

5.1. Grid convergence

Figure 6 shows the computational results without bump for three grid resolutions. As the grid resolution increases smaller vortices, which are instantaneously fed, can be captured at the LES region. However, there is not much difference among flow structures obtained by the three computational grids from the view point of the time averaged flow. Therefore, it can be concluded that the grid resolution is high enough to discuss

the time averaged flow field. Hereafter, the Fine grid is used in order to resolve fine-scale flow structure in instantaneous flow fields.

0.94 Surface pressure 1.06
-1.5 X axis vorticity 1.5

(a) Instantaneous flow (Fine grid)

(b) Instantaneous flow (Medium grid)

(c) Instantaneous flow (Coarse grid)

(d) Time-averaged flow (Fine grid)

(e) Time-averaged flow (Medium grid)

(f) Time-averaged flow (Coarse grid)

Fig. 6. Surface pressure and x direction vorticity several grid resolutions at $\alpha=50$ degrees.

5.2. Effect of bump

In this section, the effect of the existence of the bump on the flow structure is discussed. Figure 7 shows the comparison between the flow structure without the bump and that with the large bump; the two-dimensional cross-flow structure at the distance $x/D=4.5$ from the body apex and the angle of attack of 40 degrees. The schematic of the asymmetric vortexes structure reported in the previous study²⁾ is also shown in Fig. 7 for comparison. It can be observed in Fig. 7 that the almost symmetric vortex structure is obtained for the case without the bump, whereas the strong asymmetric vortex structure is obtained for the case of the large bump. The strong asymmetric flow with the large bump is qualitatively in good agreement with that reported in the previous study.²⁾

Fig. 7. Flow structure in cross-flow plane.

Figure 8 shows the computed two-dimensional cross-flow structure and the PIV measurement result of flow field by Nishida et al.¹⁷⁾ at the distance $x/D=4.5$ from the body apex, and the angle of attack of 40 degrees. Computational results for 4 types of the bump are shown in Fig. 8, and the result for the case of the medium bump is qualitatively in good agreement with the experimental result.

Fig. 8. Flow structures in cross-flow plane.

Figure 9 shows the side force coefficient plotted against the angle of attack; both the simulation results and the experimental results by Nishida et al.¹⁷⁾ are shown. It can be observed the difference between the side force coefficient of numerical simulation with large bump and that of experiment.¹⁷⁾ The computational result is expected to be made closer to the experimental result by regulating the bump size between medium and small.

Fig. 9. Side force coefficient, C_Y v.s. Angles of attack.

As discussed in this section, the numerical technique used in this study can correctly simulate the characteristics of the time averaged flow field over the slender body at high angle of attack. Although the quantitative difference is observed between the numerical and experimental results, it is indicated as reported in the previous work¹⁹⁾ that the computational result can be made closer to the experimental result by choosing the property-sized bump. The bump is needed to simulate the flow field over the slender body at high angle of attack from the qualitative viewpoint, and it is numerical technique based on the physical basis. Therefore, it is concluded that our numerical method has sufficient capability to address our research purpose.

6. Characteristics of Flow Field and Side Force

Next, the computational study on the asymmetric vortices over a slender body is conducted toward understanding the

characteristics of asymmetric vortices to change the bump height and angle of attack.

6.1. Effect of bump height

In this section, the effect of bump height on time averaged flow-fields and aerodynamic characteristics are discussed.

The simulation is conducted with progressively reducing the bump height each 10^5 steps. The large bump is switched to the medium bump at 10^5 steps. The medium bump is switched to the small bump at 2×10^5 steps. The small bump is switched to the without bump at 3×10^5 steps.

Figures 10 to 12 show the computational results for four types of the bump height. The angle of attack fixed at $\alpha=40$ degrees.

Time averaged x -directional vorticity distributions and the surface pressure distributions are shown for each bump height in Fig. 10. The strong asymmetric vortex structure can be observed in Figs. 10 (a) and (b). At (a), the disturbance created by the bump at the nose grows rapidly during the transportation by the flow and results in the strong asymmetric vortex structure indicated by the red arrow. Removal of the bump makes the asymmetric flow return to the undisturbed symmetric structure (see Fig. 10 (d)). This result confirms that the asymmetric flow structure is the result of a convective instability.²⁾ At (a) and (b), the vortex filament, which is generated at the bump (starboard side of the body), is separated from the body surface during the convection along the body (the separation points are indicated by the arrow numbered 1 in Fig. 10 (a) and the arrow numbered 2 in Fig. 10 (b)). The separation point of the vortex filament is estimated from visualization results of x directional vortex structure and iso-surface of second invariant of the velocity gradient tensor.

At the separation point of the vortex filament, the new third vortex filament is generated. The vortex filament from the body nose separates at upper position of the body (closer position of the nose) when the bump height higher.

(a) Large bump

(b) Medium bump

(c) Small bump

(d) Without bump

Fig. 10. Surface pressure and x direction vorticity in time averaged flow field at $\alpha=40$ degrees.

Figures 11 and 12 show the time history of the side force coefficient and that of the yawing moment coefficient when the center of gravity is at 55% from the body axis; the bump height was progressively changed in the simulation, respectively. As shown in Fig. 11, the nonlinear change in the side force is caused by the change of the bump height. On the other hand, as shown in Fig. 12, the yawing moment coefficient is monotonically decreased by the change of the bump height but is almost nonlinearly changed.

Fig. 11. Time history of side force coefficient.

Fig. 12. Time history of yawing moment coefficient.

The reason of the nonlinear change in the side force can be explained by focusing on the local side force distributions along the body axis. The sign of the sectional side force inverts at the separation point of the vortex filament, where the third vortex filament is generated as shown in Fig. 13. This characteristic of the sectional side force distributions is in good agreement with the knowledge obtained in a previous experimental study.^{3,20)} Therefore, the side force is comparable value between with the large bump and the medium bump instead of stronger asymmetry in the vortex structure with the large bump. As the results, the relationship between the side force and the bump height is nonlinear. This result indicates that the flow field should be carefully checked even if the simulation result of the side force is quantitatively in good agreement with the experimental result.

Fig. 13. Sectional side force coefficient of several bump sizes.

6.2. Effect of angle of attack

In this section, the effect of angles of attack on time averaged flow fields with large bump is discussed.

Time averaged x -directional vorticity distributions and surface pressure distributions at various angles of attack are shown in Fig. 14. As shown in the figure, the flow fields are asymmetric in all cases. It can be observed that the third vortex filament indicated by the all red allows in Fig. 14 is separated from the body surface. Furthermore, it can be observed that the asymmetry of the vorticities becomes stronger as the angle of attack becomes higher, and the third vortex filament indicated by the arrow numbered 3 in Fig. 14 (d) and the arrow numbered 4 in Fig. 14 (e) shed from the apex separates away from the body at upper position (closer position to the apex) when the angle of attack becomes higher; the third vortex filament is generated at upper position of the body when the angle of attack becomes higher.

The side force coefficient of the computational result is expected to be made closer to that of the experimental result by choosing the properly-sized bump is indicated in Sec.5.1.2.

Note, however, that the relationship between bump size and side force coefficient is nonlinear (see Fig. 11). These flow-structure changes depending on the angle of attack are qualitatively consistent with the results obtained by the previous researches.^{2,3)}

0.94 Surface pressure 1.06
-1.5 X axis vorticity 1.5

Fig. 14. Surface pressure and x direction vorticity in time averaged flow field at various angles of attack.

7. Summary

In this paper, we analyzed the asymmetric vortices in the flow over the slender body using three-dimensional RANS/LES hybrid CFD simulations.

First, the technical know-how for simulating the asymmetric separation flow over slender body was validated by parametric study of the grid resolution and the bump height. As the grid resolution increases, there is not much difference among flow structures obtained from the view point of the time averaged flow. It can be concluded that the grid resolution is high enough to discuss the time averaged flow field. In addition, the qualitatively good agreement correlation between the numerical and the experimental results is obtained by adding geometrical disturbance (bump) to the body apex. The quantitative difference between computational result and experimental result is thought to be caused by too large bump.

The computational result is expected to be made closer to the experimental result by choosing the properly-sized bump.

According to these results, it is concluded that the numerical method adopted in this study has sufficient capability to address our research purpose.

Next, the computational study on the asymmetric vortices over a slender body is conducted toward understanding the characteristics of asymmetric vortices to change the bump height and angle of attack. When the angle of attack or the bump becomes higher, the asymmetry of vorticities becomes stronger. The vortex filament shed from the apex separates away from the body at upper position (closer position to the apex), a third vortex filament is generated at the separation point of the vortex filament. The sign of the sectional side force is inversed at the separation point of the third vortex filament. This behavior of the flow field leads to the nonlinear relation between the bump height and the side force. There is a possibility that the amplitude of the side force does not change even if the bump height changes.

8. Future Works

We will conduct numerical simulations of the flow field over the slender body with the DBD plasma actuator, and investigate the flow control effect on the separation vortices.

The effective layout and operation method of the DBD plasma actuator for proportional control of the side force will be discussed. The results obtained by the simulation will be validated by experiments.

Acknowledgments

Computation time was provided by the Japan Aerospace Exploration Agency (JAXA) Supercomputer System. This research was partially supported by KAKENHI; Grant-in-Aid for Research Activity Start-up (21860028).

References

- 1) Ericsson, L. E. and Reding, J. P.: Steady and Unsteady Vortex-Induced Asymmetric Loads on Slender Vehicles, *J. Spacecraft and Rockets*, **18** (1981), pp.97-109.
- 2) David H. B.: The Asymmetric Vortex Wake Problem – Asking the Right Question, *AIAA Paper 2006-3553*, (2006).
- 3) Hunt, B.L., Corp, N. and CA, H.: Asymmetric Vortex Forces Wakes on Slender Bodies, *AIAA Paper 1982-1336*, (1982).
- 4) Panel, M.P., Tilmann, C.P. and Ng, T. T.: Closed-Loop Missile Yaw Control via Manipulation of Forebody Flow Asymmetries, *Journal of Spacecraft and Rockets*, **41** (2004), pp.436-443.
- 5) Bernhardt, J.E. and Williams, D.R.: Proportional Control of Asymmetric Forebody Vortices, *AIAA J.*, **36** (1998), pp.2087-2093.
- 6) Tsubakino, D. and Tanaka, K.: Effective Layout of Plasma Actuators for a Flow Separation Control on a Wing, *AIAA Paper 2007-474*, (2007).
- 7) Asada, K., Fujii, K.: Computational Analysis of Unsteady Flow-field Induced by Plasma Actuator in Burst Mode, *AIAA Paper 2010-5090*, (2010).
- 8) Liu, F., Luo, S., Gao, C., Meng, X., Hao, J., Wang, J. and Zhao, Z.: Flow Control over a Conical Forebody Using Duty-Cycled Plasma Actuators, *AIAA J.*, **46** (2008), pp.2969-2973.
- 9) Matsuno, T., Kawazoe, H., Nelson, R. C., and Corke, T. C.: Forebody Vortex Control on High Performance Aircraft Using PWM-Controlled Plasma Actuators, *Proceedings of the 26th ICAS Congress*, (2008).
- 10) Fujii, K.: Efficiency Improvement of Unified Implicit Relaxation/Time Integration Algorithms, *AIAA J.*, **37**(1999), pp.125-128.
- 11) Gaitonde, D.V. and Visbal, M.R.: Further Development of a Navier-Stokes Solution Procedure Based on Higher-Order Formulas, *AIAA Paper 1999-0557*, (1999).
- 12) Gaitonde, D. V. and Visbal, R. M.: Pade-Type Higher-Order Boundary Filters for the Navier-Stokes Equations, *AIAA J.*, **38**(2000), pp.2103-2112.
- 13) Nishida, H., Nonomura, T.: ADI-SGS Schemes on Ideal Magnetohydrodynamics, *Journal of Computational Physics*, **228**(2009), pp. 3182-3188
- 14) Iizuka, N., and Fujii, K.: Evaluation of Dynamic Base Pitching Moment of a Blunt ReEntry Capsule in Transonic Region, *Proceedings of 57th International Astronautical Congress*, IAC-06-D2P.1.9, (2006).
- 15) Chakravarthy, S. R.: Relaxation Methods for Unfactored Implicit Upwind Schemes, *AIAA Paper 1984-0165*, (1984).
- 16) Visbal, M. R. and Rizzetta, D. P.: Large-eddy Simulation on General Geometries Using Compact Differencing and Filtering Schemes, *AIAA Paper 2002-288*, (2002).
- 17) Nishida, H.: Study on Active Flow Control of Separation Flow behind Slender Body in High Angle of Attack, *Annual Meeting of Japan Society of Fluid Mechanics*, (2010).
- 18) Degani D., Levy Y.: Asymmetric turbulent vortical flows over slender bodies, *AIAA J.*, (1992), pp.2267-73.
- 19) Levy, Y., Hesselink, L. and Degani, D.: A Systematic Study of the Correlation Between Geometrical Disturbances and Flow Asymmetries, *AIAA Paper 1995-0365*, (1995).
- 20) Deng, X., Wang, Y. and Chen, X.: Deterministic Flow Field and Flow Structure Model of Asymmetric Vortices over Slender Body, *AIAA Paper 2003-5475*, (2003).