AN ANALYSIS OF THE CHARACTERISTICS OF ASYMMETRIC VORTICES AND SIDE FORCES ON SLENDER BODIES AT HIGH ANGLE OF ATTACK USING A NAVIER-STOKES CODE

S. K. Jung, S. E. Je, M. J. Park, R. S. Myong, T. H. Cho
Gyeongsang National University, Korea

1.INTRODUCTION

The flow-field around slender bodies with a tangent-ogive at high angle of attack shows in general very complicated behaviors [1-5]. For example, in the high angle of attack range 25-45 degree the instability caused by small perturbations inherent in the flow generates the asymmetric vortices on the body. As a result, side forces and phantom yaw moments due to the asymmetric flow-field are observed even in vanishing sideslip. It is generally believed that the instability in this case is of convective nature, supported by the experimental observation that multiple levels of asymmetry are possible at an angle of attack. On the other hand, at any angle of attack above the critical value (about 50 degree), only two values of side force are observed. This instability is called the absolute instability in the literature [2].

In this study, a Navier-Stokes computational code is utilized to investigate mechanisms leading to the appearance of asymmetric vortices and to examine the theories of instability of the flow at high angle of attack. In particular, in order to identify the source of flow asymmetries, the side force behavior is examined by varying the roll orientation of the slender body.

2.NUMERICAL SIMULATION

It has been reported in various previous works [1,2] that high angle of attack aerodynamics has several computational difficulties that are not always found in other flow-fields. For example, it was found that symmetric or asymmetric vortices are computed in the simulations of high angle of attack flows, depending on the type of numerical algorithm. A minor modification on the symmetric flux-vector splitting algorithm, such as the diagonalization process to speed up the calculation, can change the nature of flow drastically, from symmetric configuration to asymmetric configuration.

In this study a computation program using a compressible Navier-Stokes equation, FLUENT [6], which is free from any unknown perturbation due to the algorithm asymmetry, is utilized to investigate the flow-fields at high angle of attack.

2.1.Numerical method

The FLUENT code is based on the implicit time-marching finite volume method. For the flux calculation, the flux difference splitting method proposed by Roe is employed. The RNG k-epsilon turbulence model, which is a modified model of the k-epsilon two equation turbulence model, is used throughout since it was judged to follow experimental data very closely in the case studied after the test of various models.

2.2.Model and grid generation

The object model to generate asymmetric vortices is a slender body shape. The diameter of model is set up to D and the length of nose and full shape is set up to 3.5D and 12D (Figure 1.). The structured grid around model is generated by the commercial code, GRIDGEN. A sample grid system is composed with 6 blocks and 600,000 cells. The distribution of grid points near the wall was adjusted so as to analyze the velocity profile within the viscous boundary layer (Figure 1.).

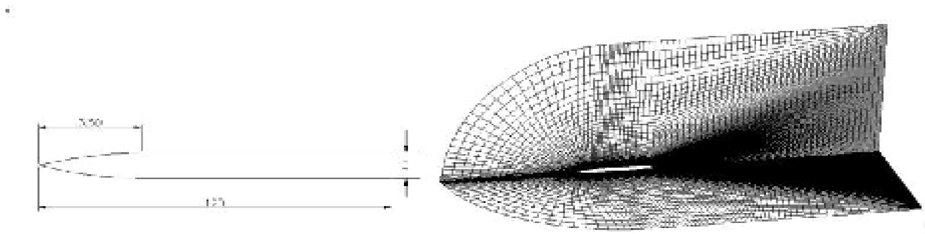

Figure 1. Model and structured mesh with 6 blocks (600,000 cells)

2.3.Flow condition and asymmetric disturbance

The flow condition of Mach number, 0.2 and Reynolds number, 3.0E+06 is chosen in the numerical simulation owing to the availability of the experimental data. The angles of attack with a range of 20-50 degrees are considered. The pressure distribution around the slender body at axial location x=6.0D is compared with the experimental data by Lamont [7]. The experiment indicates the asymmetric vortices structure at the angle of attack, 30 degrees, while the computation shows the symmetric structure, owing to the symmetric nature of the present numerical algorithm (Figure 2.).

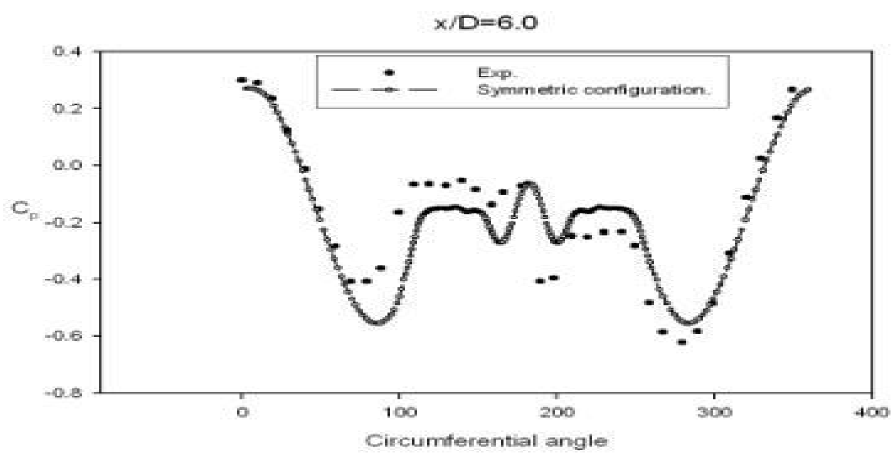

Figure 2. The comparison of the pressure distribution around a slender body($a = 30^0$).

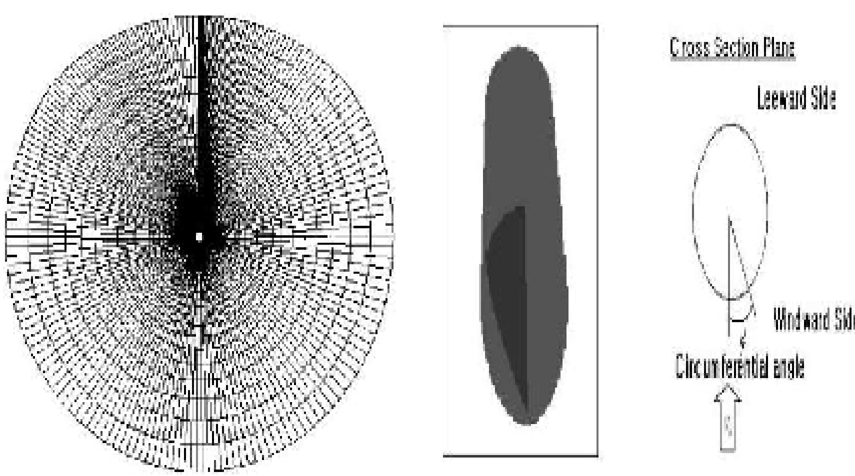

Figure 3. Asymmetrically clustered grid system and location of asymmetric roughness in the forward part of the slender body

This finding implies that some types of artificial asymmetric disturbance are necessary to simulate the asymmetric structure arising in physical flow by the numerical method.

In this study, three types of asymmetric disturbances are tested; grid, sideslip angle, and wall roughness. At first, the grid system for generating an asymmetry vortex shows at the left side in the figure 3. It means asymmetrically clustered grids of left and right side is assumed that there is a possibility to generate the different vortex pattern based on numerical errors due to the difference of grids. Then side slip angle is applied only 1 degree. It is an easy way to touch and apply for a numerical analysis. Finally, a special roughness area is applied. In the case of wall roughness, previous studies show that the mean velocity distribution near rough walls has a different intercept, in other words, an additive constant in the log-law;

$$\frac{u_P u^*}{t / r} = \frac{1}{k}\ln\left(E\frac{r u^* y_P}{m}\right) - \Delta B, \tag{1}$$

The formulas proposed by Cebeci and Bradshaw based on Nikuradse’s data are adopted to compute ΔB for each regime [8].

$\Delta B = 0,$ if $K_s^+ \le 2.25$ for the hydrodynamically smooth regime,

$\Delta B = \frac{1}{k}\ln\left[\frac{K_S^+ - 2.25}{87.75} + C_S K_S^+\right]\cdot \sin\left[0.4258\left(\ln K_S^+ - 0.8111\right)\right],$ if $2.25 < K_s^+ \le 90$ for the transitional regime,

$\Delta B = \frac{1}{k}\ln(1 + C_S K_S^+),$ if $90 < K_s^+$ for the fully rough regime,

Where,

$$K_S^+ = \left(\frac{r K_S u^*}{m}\right)$$

Here, C_S is a roughness constant and depends on the type of the roughness. The roughness constant $C_S = 0.5$ and roughness height $K_S = 1mm$ (0.26 percent of the body diameter) are used in the present study and this roughness height $K_S = 1mm$ is for the transitional regime. The method to apply the asymmetric disturbance based on the wall roughness is illustrated in Figure 3. The roughness is applied on the region with circumferential angles $180^0 < f < 240^0$.

3.VALIDATION STUDY

3.1.Slender body computation

In order to see the effect of asymmetric disturbances, the flow field around a slender body is computed for an angle of attack, 30 degree. The pressure distribution at the position x/D=2.0 turns out to be almost symmetric and thus there exists no distinction caused by the different method of applying asymmetric disturbance. However, the difference of pressure distributions of a left and right side around the slender body that asymmetric disturbances, special roughness area, sideslip angle and asymmetric grid are applied then calculated becomes pronounced at the position x/D=6.0 (Figure 4.).

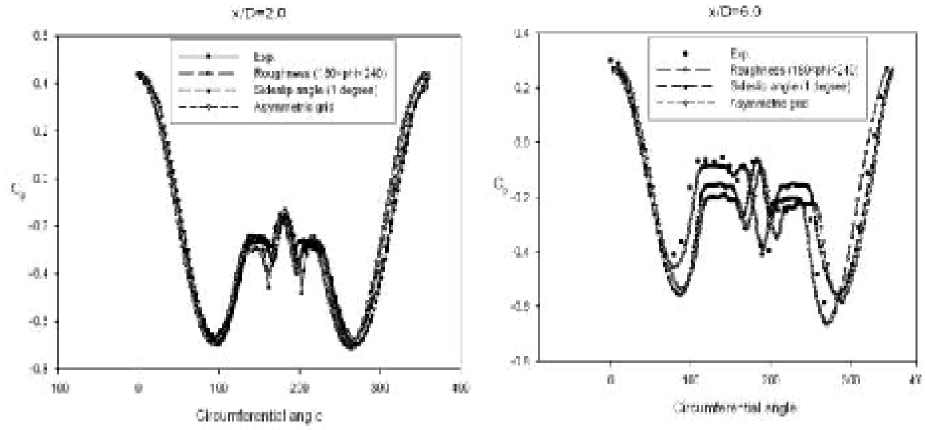

Figure 4. The comparison of the pressure distribution at x/D=2.0 and x/D=6.0 (M=0.2, Re=3.0E+06, $\alpha = 30^0$)

This indicates that the level of asymmetry is amplified along the rearward of missile. In addition, it can be noticed that the results by the present method based on asymmetric distribution of wall roughness are in good agreement with the experimental data. Moreover, asymmetric flow patterns can be confirmed through the visualization (Figure 5.). For this reason, the method is used throughout in the subsequent computations.

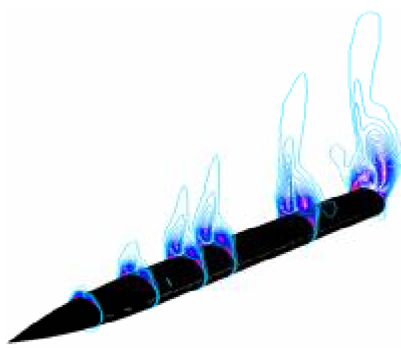

Figure 5. Pressure contours around the slender body (M=0.2, Re=3.0E+06, $\alpha = 30^0$)

3.2.Side force variation with nose roll

Even though many works have been reported in order to explain the cause of the vortex flow asymmetries arising in high angle of attack flow fields, the exact role of perturbations, flow field angularity, free stream turbulence, or surface imperfections in the asymmetries remain unclear. To circumvent this difficulty, Zilliac et al. [9] performed an experimental study on side force variation with nose roll angle. The experiment was conducted using a rotating ogive tip on the front of a cylindrical cross-section after body for a condition M=0.25, Re=3.0E+06. It was motivated by an idea that even though information of the exact circumferential location of the perturbation, by either geometric error or flow irregularity, is not available, the side force variation with the ogive roll angle can be easily measured. In this study the same problem is computed by the Navier-Stokes computational code to check the validity of the present prediction method based on the asymmetry roughness.

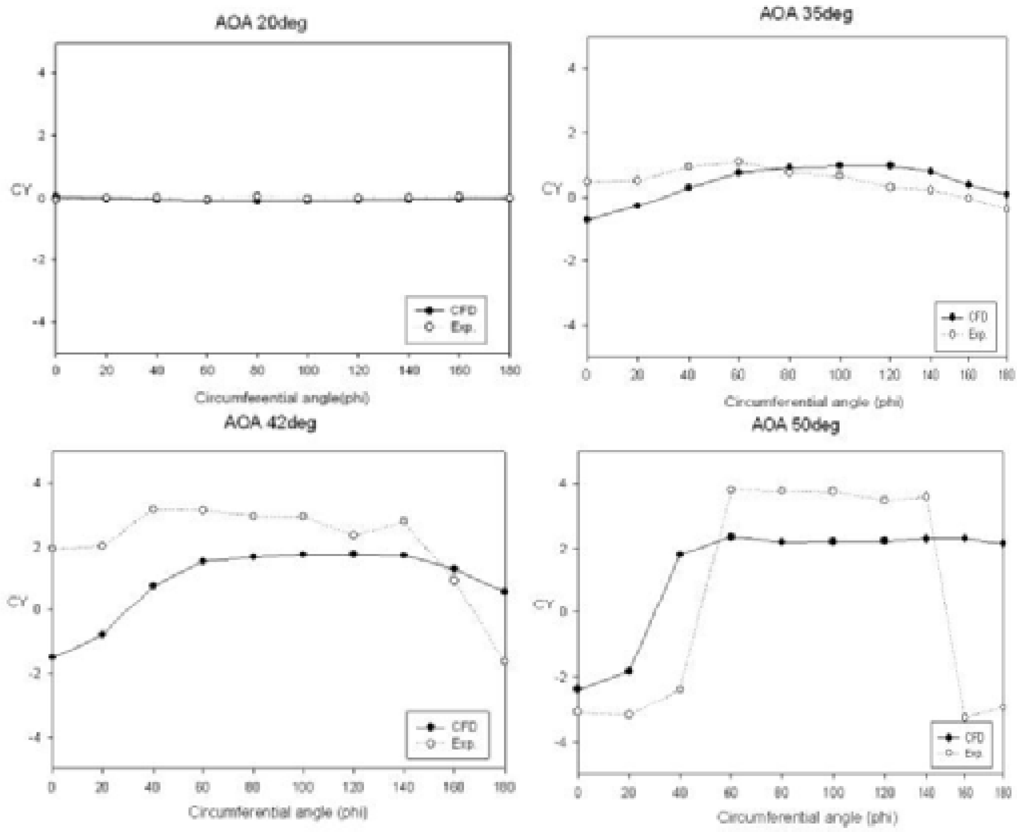

Figure 6. Side force variations with nose roll angle ($a = 20^0$, $a = 35^0$, $a = 42^0$, $a = 50^0$).

At a medium angle of attack, $a = 20^0$, there was no asymmetry both in the experiment and the prediction (M=0.4 and Re=6.0E+06). In high angles of attack, $a = 35^0, 42^0$, the side force varied continuously with the roll angle and, in other word, any value of side force between the positive and negative maximum values is possible. This implies that the asymmetry mechanism within this range of angle of attack is due to the convective instability. On the other hand, as the angle of attack reached the critical value, side forces of either a positive or negative maximum value but with no intermediate values are observed both in the experiment, $a = 50^0$, and the CFD calculation, indicating that the asymmetry in this case is due to the absolute instability. On the whole, the Navier-Stokes results are shown to be in qualitative agreement with the experiment in capturing the general features of the flow asymmetry. Therefore, it can be concluded that the present methodology based on the Navier-Stokes computational model and the asymmetric roughness is a valuable prediction tool for the study of the characteristics of asymmetric vortices and side forces in high angle of attack flow (Figure 6.).

4.APPLICATION

To verify the asymmetric disturbance, the specific roughness area of asymmetric disturbances was applied to a guided missile. Different roughness areas were applied on the nose and asymmetric vortices generated at the nose were amplified in flow directions (Figure 7.). There exist large differences of asymmetric vortices on the both sides of the missile due to the interaction of vortices and fins. Table 1 shows the results of the CFD and wind tunnel, showing good agreement with the exception of the axial force coefficient. Where, CN, CA and CY mean a coefficient of normal force, axial force and side-force and the AoA is an angle of attack.

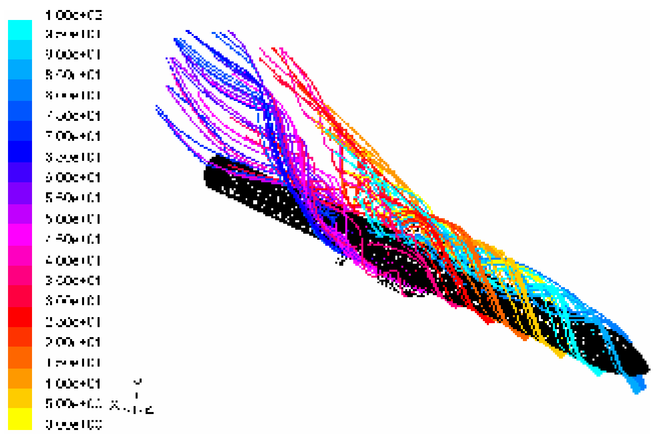

Figure 7. Streamlines around the guided missile for the speed of paticles.

Table 1. Comparison of experimental and CFD results (roll angle=450, M=0.4, AoA=300).

	CN	CA	CY
Experiment	6.1959	-0.086	0.3957
CFD	5.9284	0.1502	0.3049

5.CONCLUSION

In this work, a Navier-Stokes computational code is utilized to investigate mechanisms leading to the appearance of asymmetric vortices and to examine the theories of instability of the flow at high angle of attack. In particular, as a method to apply the physical disturbance for the computations, an asymmetry inducing method based on wall roughness is developed. In order to check the validity of the present prediction method, the side force behavior at high angle of attack is examined by varying the roll orientation of the slender body. It turned out that the Navier-Stokes results are shown to be in qualitative agreement with the experiment in capturing the general features of the flow asymmetry. Indeed, the CFD results based on a special roughness area applied to the guided missile showed quite a good agreement with the wind tunnel data.

Acknowledgements: This research is supported by the Second Stage Brain Korea 21 and NURI Project.

REFERENCES

[1]. P. Champigny, *High angle of attack aerodynamics*, AGARD Special Course on Missile Aerodynamics, (1994).

[2]. R. M. Cummings, J. R. Forsythe, S. A. Morton, and K. D. Squires, *Computational challenges in high angle of attack flow prediction*, Progress in Aerospace Science, Vol. 39, pp 369-384, (2003).

[3]. F. Lesage, J. Nicolle, and M. A. Boulianne, *Navier-Stokes computations of high angle of attack missile flowfields*, AIAA paper 2000-4212, (2000).

[4]. S. Westmoreland, and G. Gebert, *Strake effectiveness for controlling out-of plane loading on missile configuration*, AIAA paper 2000-0386, (2000).

[5]. C. C. Yuan, and R. M. Howard, *Effects of forebody strakes on asymmetric vortices of a vertically-launched missile*, AIAA paper 91-2864, (1991).

[6]. FLUENT 6.1 User's Guide, Fluent inc., (2003)

[7]. P. J. Lamont, *Pressure around an inclined ogive cylinder with laminar, transitional or turbulent separation*, AIAA Journal, Vol. 20, No. 11, pp 1492-1499, (1982).

[8]. T. Cebeci, and P. Bradshaw, *Momentum Transfer in Boundary Layers*, Hemisphere Publishing Corporation, New York, (1977).

[9]. G. G. Zilliac, *Asymmetric vortices on a slender body of revolution*, AIAA Journal, Vol. 29, No. 5, pp 667-675, (1991).