

71 回転楕円体まわりの3次元剥離流れの数値シミュレーション

推進性能部 李 信亨、日野孝則

1. Introduction

Until recently, computational fluid dynamics (CFD) techniques have shown application results only for limited conditions that are relatively far from "utilizable in actual design procedures." As the demand of CFD technology in simulations for various real world situations increases, however, it is inevitable to develop and improve CFD codes, so that more complicated and realistic physical problems can be dealt with.

The present study is being focused on CFD applications to maneuvering problems, which necessitates more and more studies to meet various rules and regulations, such as the ones imposed by the International Maritime Organization. The objectives of the present study are, therefore, (a) development of an unsteady Reynolds averaged Navier-Stokes (RANS) method for maneuvering problems; (b) evaluation of the method for a model problem, which can be described as an extreme case of submarine maneuvers; (c) providing guidelines for future code development and application to design procedures.

2. Model Problem and Mathematical Formulation

The model problem of the present study is the turbulent flow around a 6:1 prolate spheroid in the pitch-up maneuver. The pitch-up maneuver is a simple linear ramp from 0 to 30 degrees in 11 non-dimensional time units. The computational conditions are set after the experimental conditions (Wetzel and Simpson, 1998), such that the spheroid is pitched about its center and Reynolds number ($Re = U_0 L / \nu$), defined in terms of free stream velocity U_0 , spheroid length L , and kinematic viscosity ν , is set to be 4.2×10^6 . Experimental data from Hoang et al. (1994), Wetzel (1996), and Wetzel and Simpson (1997, 1998) are used for comparison and validation.

The mathematical equations for the present study are written in the Cartesian coordinate system fixed to the body, and therefore the inertia forces due to coordinate system transformation, i.e., from the space-fixed to the body-fixed, should be added as a

body force term. The inertia forces due to the transformation are, using vector notation,

$$\bar{f} = -2\bar{\Omega} \times \bar{U} - \bar{\Omega} \times (\bar{\Omega} \times \bar{r}) - \frac{\partial \bar{\Omega}}{\partial t} \times \bar{r} - \frac{\partial \bar{V}_s}{\partial t} \quad (1)$$

where $\bar{\Omega}$ and $\bar{V}_s$ are the angular and translation velocity vectors of the body fixed coordinate system, respectively, and $\bar{U}$ and $\bar{r}$ are flow velocity and position vectors in the body fixed coordinate system, respectively. For the present study,

$\bar{\Omega} = \left[0 \quad 0 \quad \frac{\partial \alpha}{\partial t} (= 0.0476) \right]^T$ is normalized by U_0/L , where α is the pitch angle.

The governing equations are continuity and unsteady 3D RANS equations for incompressible flow, written as

$$\nabla \cdot \bar{U} = 0 \quad (2)$$

$$\frac{D\bar{U}}{Dt} + \nabla p - \nabla \cdot \tau + \bar{f} = 0 \quad (3)$$

where p is pressure normalized by ρU_0^2 , and τ is the shear stress tensor.

The one-equation turbulence model by Spalart and Allmaras (1994) is employed for the present study. A modified version of Dacles-Mariani et al. (1995) is also implemented for steady flows to evaluate the influence of the eddy viscosity estimation on the overall solution behavior. The modification is not applied to unsteady flows, since it has yet to be rigorously justified.

The solution domain resembles a half of an egg with extent $-2.0 \leq x \leq 4.0$, $-2.0 \leq y \leq 2.0$, $0 \leq z \leq 2.0$ and the origin at the body center. Note that ϕ is 0° at the symmetry plane on the windward side and 180° on the leeward side. The boundary condition on each boundary is: (a) on the body, the no-slip condition with $\partial p / \partial n = 0$ (steady flow) or $\partial p / \partial n = f_n$, where f_n is the face-normal component of the body force (unsteady flow) is imposed; (b) on the outer boundary up to $x = -0.5$, $\bar{U}(\bar{r}_g) = -\bar{V}_s$ (steady flow) or $\bar{U}(\bar{r}_g) = -\bar{V}_s - \bar{\Omega} \times \bar{r}_g$ (unsteady flow), where $\bar{r}_g$ is the position vector in the space fixed coordinate system, with zero-gradient pressure

and free-stream eddy viscosity of $0.01/Re$ is imposed; (c) on the remaining outer boundary, zero-gradient perturbation velocity, i.e., $\partial \bar{U}(\bar{r}_g)/\partial n = 0$ (steady flow) or $\partial [\bar{U}(\bar{r}_g) - \bar{\Omega} \times \bar{r}_g]/\partial n = 0$ (unsteady flow), with $p = 0$ and zero-gradient eddy viscosity is imposed; and (d) on the symmetry plane, $\partial(U, V, p, v_i)/\partial n = W = 0$ is imposed. The initial condition for unsteady flow calculations is a corresponding steady flow solution at $\alpha = 0$.

3. Numerical Method

A numerical method was developed and implemented in a computer code for the solution of the unsteady RANS mathematical formulation and modeling described in the previous section. The main flow solver was developed by Hino (1997) and a variety of validation tests including free-surface flows around practical ship hull forms were carried out (Hino, 1999; Rhee and Hino, 2000a). The code's capabilities were extended to unsteady flow computations following Rogers et al. (1991) and results of fundamental test cases were reported in Rhee and Hino (2000b). Also detailed steady flow results of 3D turbulent separation around a prolate spheroid with the modification of the SA model are available in Rhee and Hino (2000c). In the present study, the unsteady flow computation procedures are refined and a body force term and boundary conditions for general maneuvering motions are included.

In the spatial discretization of the governing equations, an artificial compressibility is introduced into the continuity equations to couple a pressure field with the corresponding velocity one. The finite volume method is adopted for spatial discretization. First, the computational domain is meshed into unstructured polyhedral cells. Cells of various shapes can be used for the volume meshing and the faces of each cell can be triangles (for tetrahedral) or rectangles (for hexahedra) or combinations (for prisms and pyramids). Flow variables are stored at the center of each cell. For inviscid fluxes, the second order upwind scheme based on the flux-difference splitting of Roe (1986) with the MUSCL approach is employed. Viscous fluxes are evaluated by the second order central scheme. After the spatial discretization, time derivative terms are discretized using Euler backward and the second order backward scheme for pseudo- and physical time derivative terms, respectively. The resulting linear equation is solved by the Symmetric Gauss-

Seidel iteration. The pseudo-time iteration continues until the averaged pressure residual between pseudo-time iteration, i.e., continuity equation imbalance, reaches a convergence criterion, three orders of magnitude drop in the present study, or the iteration number reaches its pre-set maximum.

Owing to the abrupt start and stop at the beginning and end of the pitch-up maneuver, a special care should be taken in the evaluation of the body force term and a second-order accurate central difference scheme in time is employed in the present study.

In order to exploit the simplicity of the geometry, hexahedral cell grids were generated using GRIDGENTM software by mixed algebraic/elliptic method. The average spacing off the body surface in the normal direction is about 1×10^{-5} .

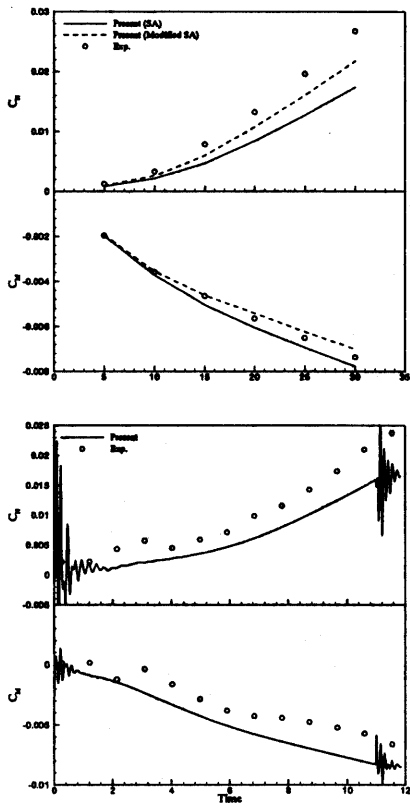


Figure 1. Normal force and pitch moment

4. Results and Validation

The simulation results are analyzed using the global force and moment, separated flow field observation, separation location, and pressure and skin friction coefficients. Comparison is made with available experimental data and the difference of flow features between steady and unsteady pitch-up maneuver flows is discussed.

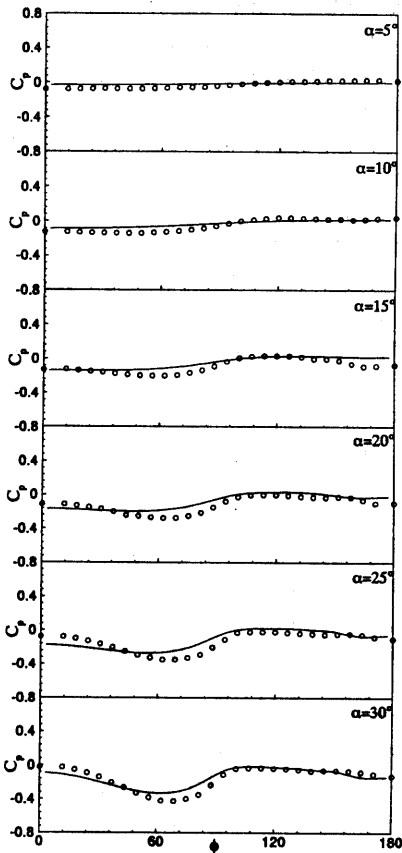


Figure 2. Pressure coefficient at $x/L=0.90$

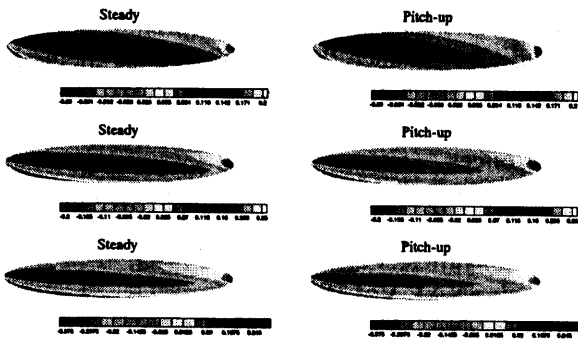


Figure 3. Pressure contours on the body at $\alpha=10^\circ$ (top), 20° (middle), and 30° (bottom)

Figure 1 displays normal force $\left(C_N = \text{Normal force} / \frac{1}{2} \rho U_o^2 L^2\right)$ and pitch moment $\left(C_M = \text{Pitch moment} / \frac{1}{2} \rho U_o^2 L^3\right)$ coefficients for steady and pitch-up maneuver with experimental data (Wetzel and Simpson, 1997). In both cases, the errors, which is defined as the difference between experimental data and computational results hereafter, in C_N and C_M increase up to 38.2% with increasing α or equivalently time, implying the difficulty of

simulating strongly separated flows accurately at large pitch angles. The abrupt start and stop of the pitch-up maneuver cause large oscillations at the beginning and end of the maneuver in both experimental and computational results, although experimental data are shown at several points only.

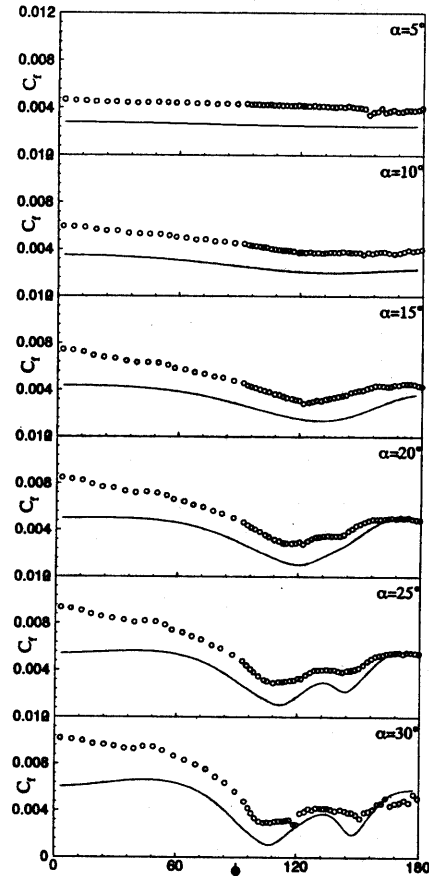


Figure 4. Skin friction coefficients at $x/L=0.882$

Pressure coefficients $\left(C_p = p / \frac{1}{2} \rho U_o^2\right)$ at $x/L=$

0.90 are presented in Figure 2 for increasing α . Experimental data (Hoang et al., 1994) are also shown for comparison. The overall agreement between computational and experimental results is quite good, and the flow development during the pitch-up maneuver is well predicted, especially the large pressure variation on the leeward side. The leeward side vortices induce strong and cohesive swirling motions on the body surface, which results in suction peaks in the leeward side at $\alpha=20^\circ$ through 30° . Toward the trailing edge of the body, both the computational and experimental results show flattened C_p , indicating detached vortices. As expected from the large errors in C_N prediction, however, suction peaks are under-estimated at higher α and near the trailing edge, which is related

to the weaker vortices in the flow and attributed to the over-estimation of eddy viscosity.

Figure 3 shows the pressure contours on the body surface at $\alpha=10^\circ$, 20° , and 30° for both the steady and pitch-up maneuver cases. For the pitch-up maneuver cases, the suction peaks values are higher and shifted leeward compared to those in steady cases. Even though the effective pitch angle (Ericsson, 1992), which is approximately 3° at most near the leading and trailing edges, is considered, the comparison clearly shows that the unsteady flow pressure cannot be correctly predicted using the corresponding steady or quasi-steady results.

Skin friction coefficient $\left(C_f = \tau_{wall} / \frac{1}{2} \rho U_o^2 \right)$ at $x/L=0.882$ are presented in Figure 4 for increasing α . Experimental data (Wetzel, 1996) are also shown for comparison. The overall agreement and trend of computational results compares favorably with the experimental data, although the errors seem to be larger than those of C_p comparisons. The errors increase with increasing x/L even at small α and on the windward side, emphasizing the significance of turbulence modeling in the skin friction prediction for a body with strong cross flows around. Also the values at local minima are under-estimated, supporting the argument of over-estimated eddy viscosity. The agreement in the locations of local minima indicate that the flow development patterns are well predicted, while the computational results show somewhat slower separation formation, i.e., approximately 5° shift leeward.

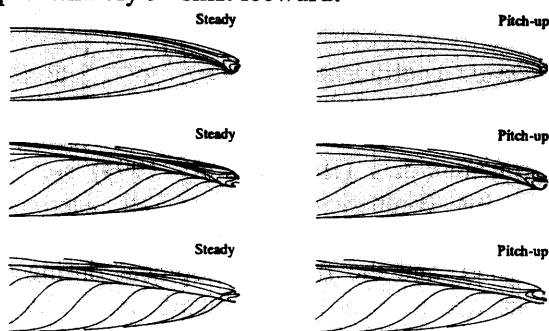


Figure 5. Limiting streamlines at $\alpha=10^\circ$ (top), 20° (middle), and 30° (bottom)

The difference in the flow development patterns for steady and pitch-up maneuver cases can be viewed in Figure 5 by the limiting streamlines near the trailing edge at $\alpha=10^\circ$, 20° , and 30° for both the steady and pitch-up maneuver cases. The pitch-up maneuver case results clearly display the delayed separations, and the less steep gradient of limiting

streamlines confirm the history effect of particle movement in the unsteady flow. This trend also strengthens the argument that in an unsteady flow the separation pattern itself can be quite different from its counterpart in an equivalent steady configuration.

5. Concluding Remarks

An unsteady RANS method was developed and implemented for the turbulent separated flow around a maneuvering 6:1 prolate spheroid. A body force term is added in the governing equations to account for the maneuvering motion in a body fixed coordinate system. The computational results for both the steady and unsteady flows compare favorably with experimental data, and the trend of flow development and difference between steady and pitch-up maneuver cases are correctly predicted. Considering the clear difference of unsteady flow patterns from steady ones, it is confirmed that an unsteady solution approach must be used for maneuvering problems. As the pitch angle increases and toward the trailing edge, however, the computational results under-estimate normal force and pitch moment, as well as suction pressure and skin friction peak values. An improvement of turbulence modeling is deemed necessary to resolve these deficiencies. A faster solution algorithm would also enhance the usability of the method for practical design procedures.

References

- Dacles-Mariani et al., AIAA J., Vol.33, No.9, 1995.
- Ericsson, in Progress in Astronautics and Aeronautics, Vol.141, AIAA, 1992.
- Hino, J. SNAJ, Vol.182, 1997.
- Hino, J. SNAJ, Vol.186, 1999.
- Hoang et al., AIAA-94-0197, 1994.
- Rhee, Hino, Proc. Gothenburg 2000 Workshop, Sweden, 2000a.
- Rhee, Hino, Proc. 4th ICHD, Yokohama, 2000b.
- Rhee, Hino, J. SNAJ, Vol.188, 2000c.
- Roe, Ann. Rev. Fluid Mech., Vol.18, 1986.
- Rogers et al., AIAA J., Vol.29, No.4, 1991.
- Spalart, Allmaras, La Recherche Aerospatiale, No.1, 1994.
- Wetzel, <http://www.aoe.vt.edu/aoe/physical/dyppir/dypdiss.htm>, 1996.
- Wetzel, Simpson, AIAA J., Vol.36, No.11, 1998.
- Wetzel, Simpson, AIAA-97-0618, 1997.