Proceedings of
The Fourth National Congress on Fluids Engineering
August 23-25, 2006, Kyungju, Korea

회전하는 구의 공력특성에 수치해석적 연구

S.V.Deshpande* · 이영기** · 김희동***

A Computational Study of Aerodynamic Characteristics of Spinning Sphere

S.V.Deshpande*, Y.-K.Lee** and H.-D.Kim***

Keywords :Drag *Coefficient(항력계수), Separation Line(박리선), Spinning Effect(회전효과)*

Abstract

Computational Study of a sphere subjected to free stream flow and simultaneously subjected to spinning motion is carried out. Three dimensional compressible Navier-Stokes equations are solved using fully implicit finite volume scheme. SST(Shear Stress Transport) *k-ω* turbulence model is used. Aerodynamic characteristics being affected are studied. Validation of the numerical process is done for the no spin condition. Variation of drag coefficient and shock wave strength with increase in spinning rate is reported. Changes in the wake region of the sphere with respect to spinning speed are also observed.

1. INTRODUCTION

A Sphere subjected to linear motion and spinning motion simultaneously is studied. The direction of spin given to the sphere distinguishes the flow problem. Axis of spin is parallel to the free stream flow. Aerodynamic characteristics change by virtue of spinning of sphere. Also because of formation of shear layers, flow separation, shock waves boundary layer interaction, flow field is affected. An attempt has been made to understand and explore the flow problem. Flow problem holds application in aeroballistics of bullets, submunitions, parachutes etc. and also in rocket engines where the turbine and compressor components rotate about an axis parallel to the direction of the flow of air.

An attempt has been made to solve the flow problem numerically. Three dimensional Navier-Stokes equations are solved using commercial software FLUENT. Fully implicit finite volume scheme with SST(shear stress transport) *k-ω* turbulence model is used.

Research interests in the simulation of high-speed flow over spinning bodies of revolution have basically been focused on the cross-flow separation at low spinning rate associated with flight stability [1-5]. To the authors' knowledge, only few papers were published regarding the physics of flight bodies with the axis of spin, parallel to the direction of flight. Wieselsberg et al.[1] carried out an experimental study that estimated a variation of drag coefficients of a sphere with respect to the critical Reynolds number and rotation parameter. Schlichting[2] defined a rotation parameter as the ratio of the circumferential velocity to the free stream velocity, which is dominant dimensionless quantity used for determining the drag coefficient and the location of separation point. Hoskin et al.[3] reported the effect of rotational speed of a sphere on the location of separation point. Kim et al.[4] conducted a numerical study to examine the variation in flow characteristics over a sphere at very low Reynolds numbers. The researches explain only incompressible flow physics of aerodynamic bodies at low spinning rates. At high flight speeds and spinning rates, however, aerodynamic characteristics have not been investigated for even a simple object such as sphere or cylinder. It would be very useful to make this point clear from an academic point of view but not for a practical purpose.

2. NUMERICAL SIMULATION

2.1 Testing Model

A sphere of diameter (D) 100 mm is used for the present study. Fig.1 shows the schematic diagram of the testing model with the Cartesian coordinate system. In the figure, M_∞ and ω are the Mach number and angular velocity of sphere and subscript ∞ indicates freestream. The origin of the computational domain is located at the center of model front face, i.e. the stagnation point. The sphere spins about the x-axis which is the same as the flight axis. In order to model the moving zone, a rotating reference frame is used in computation. In this approach, a rotational speed is given to the x-axis in the clockwise direction. An effect of wall roughness on flow physics has not been taken into account in the present study.

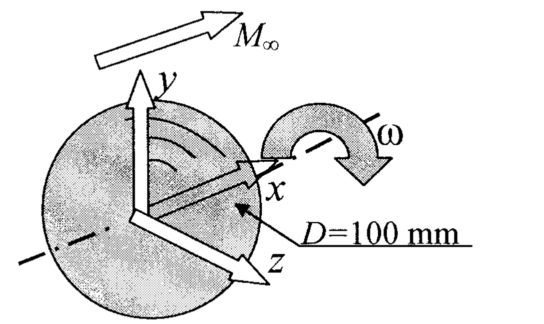

Fig.1 Testing model and Coordinate system

* 안동대학교 대학원 기계공학과, srikanth@anuis.andong.ac.kr
** 안동대학교 기계공학부, yklee@andong.ac.kr
***안동대학교 기계공학부, kimhd@andong.ac.kr

2. 2 Numerical Methods

A commercial CFD (computational fluid dynamics) code, FLUENT 6 is adopted to analyze a complicated fully three-dimensional flow passing over a spinning sphere, including shock waves, strong shear flows and viscous dominant regions by large separation depending on flight conditions under consideration. The three dimensional, compressible, mass-averaged Navier-Stokes equations were solved to examine the effect of the rotational speed on the flow characteristics of the sphere, using a fully implicit finite volume scheme, a second order upwind scheme and the SST k-ω turbulence model which was developed based on Menter's[6] effective blending of k-ω model in the near wall region and the free stream dependence of the k-ε model in the far field. It has been known that this turbulence model has an ability to provide reliable simulation of flows with an adverse pressure gradient and shock waves and near wall flows.

2.3 Computational Grid and Boundary Condition

The structured grid system used in the present computation is given in Fig.2. The layout shows the grid structure on the wall surface and of the near-field in the xy-plane. The grid system must be built in consideration of flow physics, which are of great importance for accurate simulation, under limited computational resources. Grids are therefore clustered in the regions with a large gradient where a shock wave, shear layers and wake flow are expected to occur and near the sphere surface The grid system with hexahedral cells consists of about 440,000 nodes, which were required to get grid independent solutions.

Fig.3 shows the schematic diagram of the present computational domain which is built basically with an O-type grid system and boundary conditions. The pressure far-field condition applied to the outer boundaries should be far enough from the model so as to meet free stream conditions and thus to offer better convergence. The domain has a circular free stream boundary apart from the model in a distance of 30D with an extended region up to 80D downstream of the sphere. In the domain, all free- stream boundaries are identified with the pressure far-field condition. From some preliminary tests, this type of free stream condition gave better convergence than when the pressure outlet condition was applied to the right side face of free stream boundaries. To specify the free stream condition, the Mach number and static properties were applied to the boundaries. In the present computation, for simplicity, the static pressure and temperature are assumed as 101325 Pa and 288.15 K, respectively. For the sphere surface, no-slip and adiabatic wall conditions are used.

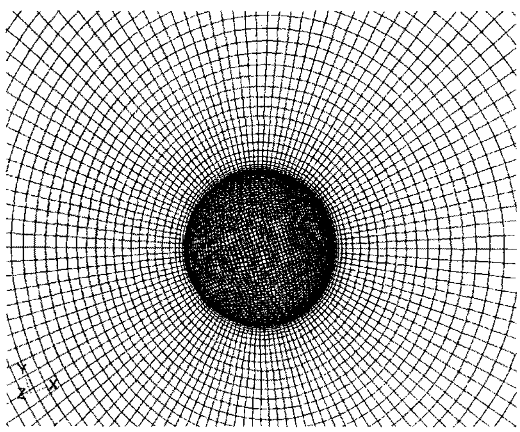

Fig.2 Structures Grid System

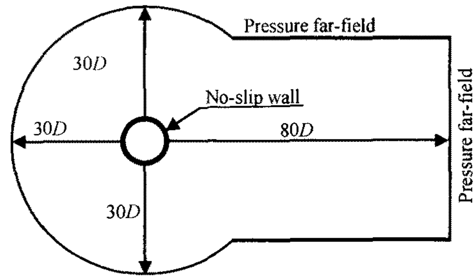

Fig.3 Boundary conditions and domains

2.4 Testing Conditions and Analysis

To provide various flow characteristics over rapidly spinning sphere, the free stream Mach number and rotational speed are changed in the range from 0.7 to 2.0 and 0 to 1,000,000 RPM respectively. Considering the ordinary rotational speed of bullets (around 300,000 RPM), the maximum rotational speed is extremely high which is somewhat beyond practical interest. The range of rotational speed, however, needs to be large enough because most significant changes in aerodynamic characteristics are obtained at very high rotational speeds in very high speed flight.

In the present computation, basically, solutions were considered converged when the residuals of all governing equations dropped to 1.0×10^{-5}. The mass imbalance was also checked for flow inlet and outlet boundaries. The mass imbalance was defined by the ratio of the imbalanced amount of mass to the incoming mass. With the main convergence criterion, it has been kept to be less than ±0.0001%

Spinning speeds (RPM)	M_∞	$\mathrm{Re} = \frac{\rho U_\infty D}{\mu}$
0	0.7	1.63×10^6
100,000		
300,000	1.2	2.8×10^6
500,000		
700,000	2.0	4.66×10^6
1,000,000		

Table 1 Flow Conditions employed in the present study

Table.1 gives the spinning speed range considered. All the spinning speeds are tested for each of the free stream Mach numbers given.

3. RESULTS AND DISCUSSION

3.1 VALIDATION

Validation of the computational results is done by comparing with experimental results obtained by Spearman et.al[7]. Total drag coefficients are compared for no spin condition of the sphere for various free stream Mach numbers. Qualitative and quantitative agreement between the two is seen. Fig 4 represents the graph obtained during validation.

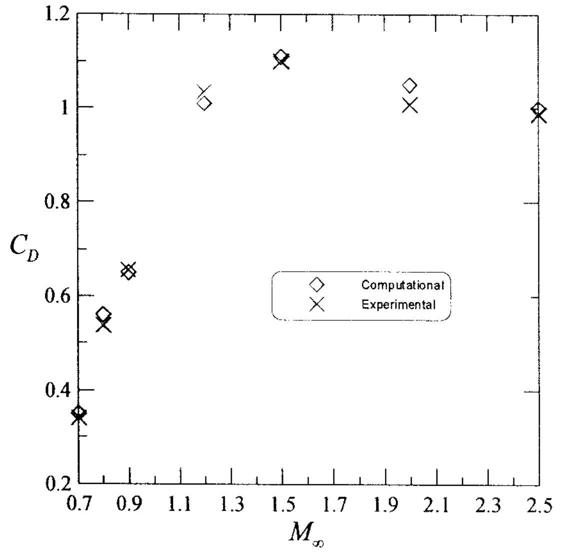

Fig.4 Graph for Validation

3.2 RESULTS

After validating, effect of spin on the total drag coefficient is considered. Fig5 shows the variation where total drag coefficient is plotted against the spinning speed. For subsonic free stream velocities, drag coefficient value increases whereas for supersonic free stream velocities there is no relative change in drag observed.

Fig.6 shows the effect of spinning on the strength of shock wave and its thickness. A line is considered upstream of the sphere and static pressure variation are plotted to investigate the same. It is found that the strength of the shock wave, which is defined as the ratio of static pressure upstream of shock wave to that of down stream remains the same for the spinning speed range considered. Also it can be observed that shock front thickness increases with increase in spinning speed which makes the static pressure change more gradual than the case for no spin.

Third aspect important for study is the shifting of separation line with change in spinning speed. As it is universally known that predicting a separation line for three dimensional compressible external flow becomes equally difficult as solving the flow problem

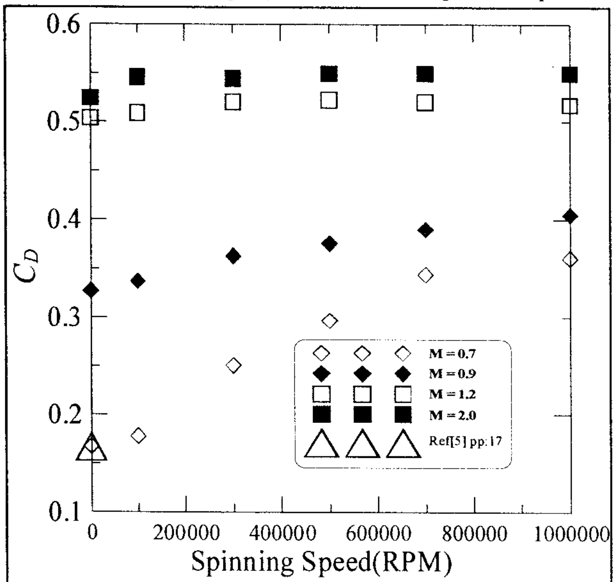

Fig.5 Drag versus spinning speed

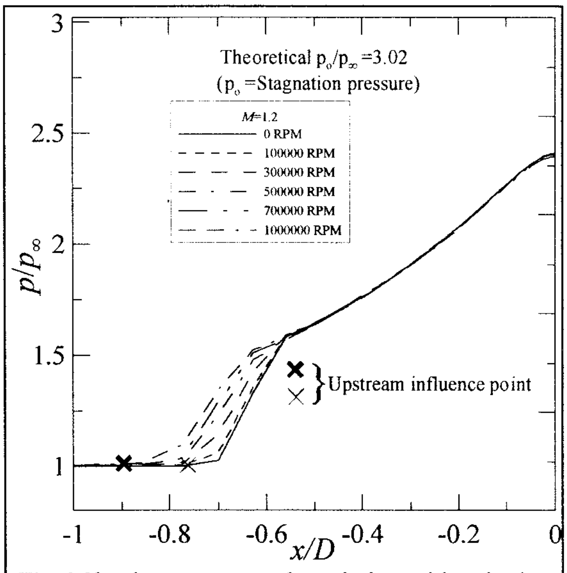

Fig.6 Shock wave strength variation with spinning

itself. After extended literature survey on the topic, wall shear stress distribution on the sphere was expected to through some light on the topic pointing out regions of zero wall shear stress which means no fluid sticking on to the surface of the fluid. Similar approach was followed and results are shown in Fig.7. Values of wall shear stresses represented by the contours are not shown because, our region of interest is the blue region which corresponds to the zero wall shear stress. Incase of subsonic free stream velocities, blue region, both on the upstream and downstream of the sphere surface is observed whereas for supersonic free stream velocities, blue region is only found along the downstream surface of sphere as it is the case with normal flow over sphere. The reason for such a phenomenon may be attributed to the ratio of circumferential velocity to the free stream velocity. Incase of the highest spinning speed and subsonic free stream velocity (M_∞=0.7 and 1,000,000 RPM), the ratio is greater than unity which means that circumferential velocity dominates over free stream velocity. Incase of supersonic free stream velocities (M_∞=2.0 and 1,000,000 RPM), free stream velocity dominates over the circumferential velocity even for the highest spinning speed considered. Also the centrifugal force corresponding to the circumferential velocities will influence the flow around the sphere.

The wake region for all the cases is studied. Wake region studies just ascertain the points mentioned in the study of drag coefficient. For subsonic free stream velocities, the wake region is affected in the sense that the low pressure region in the wake narrows down with the increase in spinning speed. Where as for supersonic free stream velocities, there is no change in the wake region for the entire speed range considered. The effect of the ratio of circumferential velocity to the free stream velocity has its effect on the wake region being affected or otherwise.

4. CONCLUDING REMARKS

The effect of spinning speed on a translating sphere at subsonic and supersonic Mach numbers has been studied. The static pressure distribution along the sphere surface at various cases considered for relatively low spinning speeds (<100,000 RPM) showed no significant changes for M_∞=0.7. At higher RPM, the static pressure distribution does change. This means that after a certain limit of spinning speed the static pressure distribution starts changing.

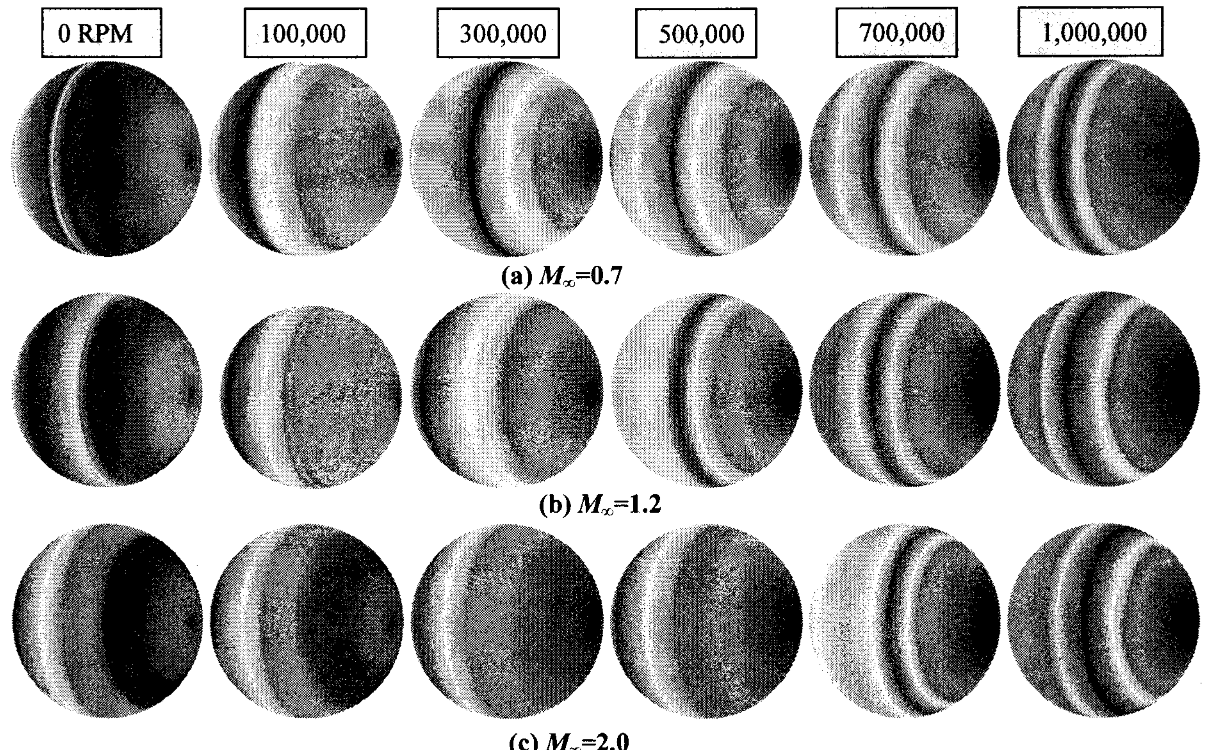

Fig.8 Wall Shear stress contours on the surface of the sphere

Probably that limit has not been crossed for M_∞=2.0. For spinning speeds greater than 1,000,000 RPM we may expect the static pressure distribution to be changed. As of now this part remains as a scope for further studies.

The results obtained show that the spinning of the sphere in the streamwise direction has a significant effect on the aerodynamic characteristics. The drag coefficient varies in proportion with the spinning speeds for subsonic freestream velocities while it is almost constant in case of supersonic freestream velocities.

The wall shear stress behavior with respect to spinning speeds shows that centrifugal force during spinning and the ratio of circumferential velocity to free stream velocity has significant effect on the aerodynamic characteristics. The complex flow phenomenon needs further in-depth analysis in order to capture the apt flow physics involved. Observing the zero shear stress regions both in the frontal face and rear face of the sphere, it is interesting to study the pressure distribution on planes considered upstream as well.

References

[1] Wieselsberger,.C, 1927, "Uber Den Luftwiderstand Bei Gleichzeitiger Rotation des Versuchskorpers". Phys. Z.28,pp 84-88

[2] Schlichting, H, 1953 "Die Laminare Stromung um Einen Axial Angestromten Rotierenden Drehkorper", Vol .XXI, No.4. Translated to English "Laminar Flow About a Rotating Body of Revolution in an Axial Airstream", NACA TM 1415

[3] Hoskin,N.E, 1955, "The Laminar Boundary Layer on a Rotating Sphere." Fifty years of Boundary Layer Research (W. Tollmien and H. Gortler, ed/), Braunschweig, pp 127-131.

[4] Dongjoo Kim, Haecheon Choi : Laminar Flow past a Sphere Rotating in Streamwise Direction .:JFM (2002)Vol, 461, pp. 365-386

[5] Schlichting. H, 1979 : "Boundary Layer Theory" pp 243-244

[6] Menter. F.R, 1994 "Two Equation Eddy Viscosity Turbulence Models for Engineering Applications" AIAA Journal, Vol 32(8):pp 1598-1605

[7] Lorey Spearman.M and Dorothy Brasswell.O. 1993. "Aerodynamics of a sphere and an oblate spheroid for Mach numbers from 0.6 to 10.5 including some effects of test conditions." NASA TM 109016