NUMERICAL SIMULATIONS OF UNSTEADY FLOW AROUND AN ENTRY CAPSULE

Axelle Viré[1]

[1]*Université Libre de Bruxelles, Avenue F.D. Roosevelt 50, 1050 Brussels, Belgium, Email: avire@ulb.ac.be*

ABSTRACT

Unsteady near-wake characteristics of a sphere-cone bluff body in incompressible flow are investigated numerically, using a hybrid spectral/finite element solver. Direct Numerical Simulations (DNS) are performed at a Reynolds number Re=1000. Comparison with experiments at higher Re [1] confirms that results are independent of the Reynolds number for Re>1000. Solutions also seem to support the hypothesis of F.Y. Wang, *et al.* [2]: oscillations of the wake closure point could be responsible for the appearance of additional swirl structures in the wake. Moreover, simulations indicate the existence of an unsteady lateral force at a 15° angle of attack.

Key words: Unsteady flow; reentry; wake; flow visualization; DNS.

1. INTRODUCTION

At low speeds, before the splashdown (or touchdown), the turbulent flow field around an entry capsule induces unsteady forces on the capsule. In particular, even if the capsule geometry is axisymmetric, the flow field is not because of three-dimensional effects which manifest themselves as ring vortices in the azimuthal direction. The understanding of these phenomena is essential to ensure both static and dynamic stability. In addition, parachute deployment plays an important role in the stabilization of the capsule. The aim of the present study is to analyse the flow field around an entry capsule at low Reynolds numbers numerically, using a hybrid spectral/finite element solver. Apollo is chosen as the representative geometry. Results are compared qualitatively with water tunnel experiments at low speeds and quantitatively with literature at higher Reynolds numbers.

2. MODEL

The scaled drawing of the model is shown in Fig. 1. The heat shield forward position corresponds to the zero angle of attack and incidences increase in the clockwise direction.

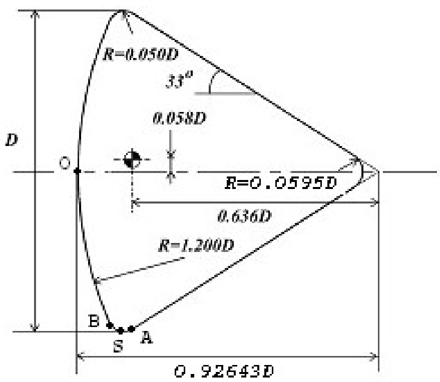

Figure 1. Apollo capsule model.

3. WATER TUNNEL VISUALIZATIONS

A preliminary experimental study is conducted in the WT1 water tunnel of the Von Kármán Institute for Fluid Dynamics (VKI). The periodicities of the eddies in the near wake are deduced from the video recordings and no quantitative measurement device is used. The maximum speed in the 240mm x 120mm section of the tunnel is 0.15 m/s. Two small-scale models are used. Both are characterized by a heat shield diameter d=50mm but they differ from the dye injection holes position. The first model, equiped with dye injection holes at the junction between the spherical and the conical part, is used to visualize the shear-layer instability. The shear-layer is characterized by a pseudo-periodic motion of ring vortices (Fig. 2). The pairing of two or three vortices can be observed as illustrated in Fig. 3. Using the heat shield diameter as the non-dimensional length, the Strouhal number St of this Kelvin-Helmoltz instability is approximately equal to 2.1.

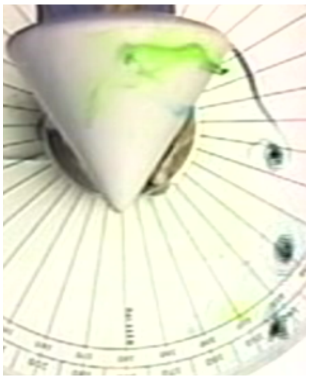

Figure 2. Shear-layer instability (Re=3700).

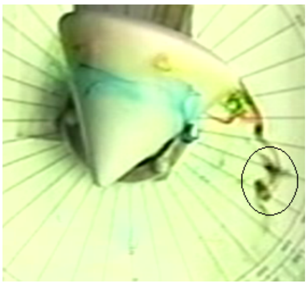

Figure 3. Pairing of two vortices (Re=3000).

The second small-scale model allows the visualization of the "fish tail motion" downstream of the wake. The Strouhal number evaluation is quite difficult due to rapid dispersion of coloured liquid. However, St=0.19 constitutes a good approximation. The periodicity of this oscillation motion in the vicinity of the wake closure point is one order of magnitude smaller than that of the shear-layer instability.

4. NUMERICAL SIMULATIONS

Numerical simulations are performed using a hybrid spectral/finite element solver developed for Direct Numerical Simulations (DNS) of incompressible turbulent flows around axisymmetric geometries. A Fourier series is used to represent flow variations in the azimuthal direction whereas a finite element discretization of the equations is applied in the meridional plane. As a consequence, the resolution of a three-dimensional problem is decomposed into a series of two-dimensional problems.

Two angles of attack are studied, 0° and 15°, at a Reynolds number Re=1000. Computations simulate 100 seconds (real time). Results are compared with experimental data available in the literature [1] at higher Reynolds numbers in terms of periodicities, pressure distributions, force and moment coefficients.

4.1. Mesh

The domain is a cylindrical box. The heat shield center is located at 5d downstream of the inlet surface. The domain length equals 26d and the domain height with respect to the capsule axis of rotation is 6d as shown in Fig. 4. The mesh contains 26,000 nodes and 51,452 triangular elements.

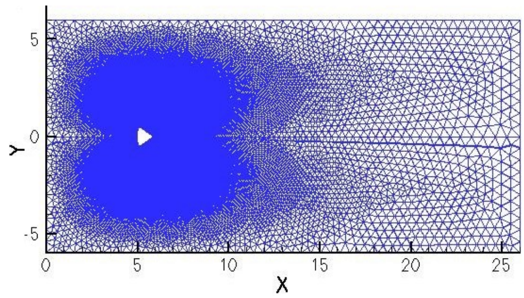

Figure 4. Mesh.

4.2. Periodicity

Numerically, the Strouhal number of the low frequency mode is estimated to be St=0.165-0.200 whereas mesh cells are not sufficiently small to capture the high frequency shear-layer instability.

4.3. Pressure distribution

The pressure distribution is maximum at the heat shield stagnation point and decreases in streamwise direction as illustrated in Fig. 5. Its sign changes to negative before the end of spherical part. On the conical part, the distribution is uniform. Pressure coefficient C_p is compared with literature experimental data [1] obtained at $2.10^5 \leq Re \leq 1,2.10^6$.

4.4. Aerodynamic coefficients

The convention chosen for aerodynamic coefficients is depicted in Fig. 6, where the x-axis coïncides with the capsule axis of rotation. D

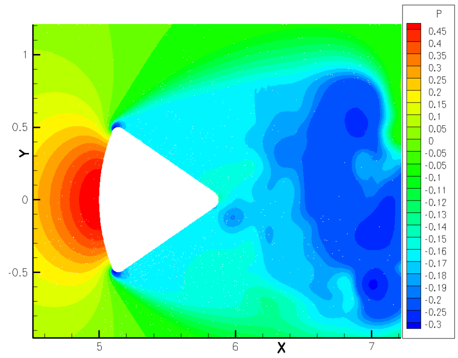

Figure 5. Pressure distribution $P - P_\infty$ at 0°

stands for the drag, L the lift, Fz the lateral force, Cx the rolling moment, Cy the yawing moment and Cz the pitching moment (positive for a nose-up configuration).

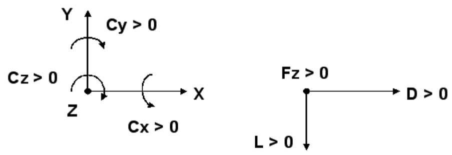

Figure 6. Aerodynamic coefficients convention.

At a zero angle of attack, the configuration is symmetric which is why perturbations are initially added to allow three-dimensional effects to start. The analysis of the perturbations influence confirms that asymptotic values of aerodynamic coefficients are quite unsensitive to the perturbations. However, at a zero angle of attack, if no perturbation is introduced, the flow field is two-dimensional and the drag coefficient is twice smaller than in a flow in which three-dimensional effects appear due to the addition of perturbations (three-dimensional flow field). Moreover, in a three-dimensional flow field, aerodynamic coefficients experience unsteady variations which suggest that three-dimensional effects are responsible for a deterioration of aerodynamic characteristics of the capsule and also influence its stability.

For a three-dimensional flow field, non-dimensional force coefficients are compared with experimental data at Re=60,000 from the literature [1]. Numerical simulations indicate that, at a 15° angle of attack, unsteady phenomena also appear in the lateral direction, along the z-axis, and this has not been observed during the wind tunnel experiments [1]. In fact, the lateral force coefficient equals zero during 25 seconds before starting to oscillate. In Fig. 7, the dotted line concerns a 345° angle of attack whereas the other one is obtained for the symmetric case (15°). Even if the oscillations amplitude is small, to the author's knowledge, this phenomenon has never been referred in the literature. Further simulations should be conducted at different incidences in order to support the present solutions.

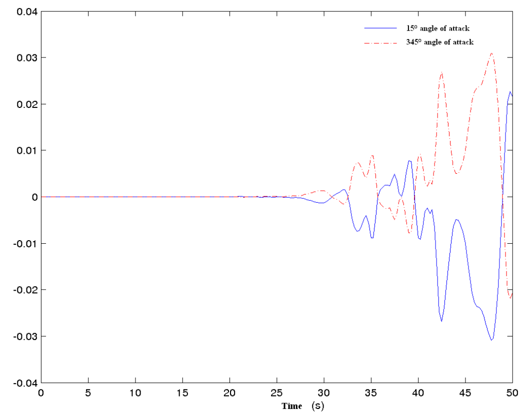

Figure 7. Lateral force coefficient.

Non-dimensional moment coefficients are compared with experimental data obtained in the literature at Re=110,000 [1]. Both rolling and yawing coefficients with respect to the theoretical center of gravity position are equal to zero. The pitching moment analysis confirms that a 15° angle of attack corresponds to the trim position when the heat shield faces the stream.

4.5. Flow visualizations

A movie was made based on numerical visualizations. This approach allows the comparison with experimental flow visualizations in water tunnel. Large recirculation regions are observed in the near-wake. Also, after the destruction of the vortex rings, a part of the vortices rolling-up in the upper portion moves into the near-wake and becomes part of the flow convected on the lower portion of the capsule. As suggested in [3], this phenomenon may disturb the vehicle stability by introducing an amplification of the incidence oscillation magnitude. Furthermore, instantaneous numerical visualizations illustrate large oscillations of the wake closure point and the apparition of additional swirl structures which do not belong to vortex rings. F.Y. Wang, *et al.* [2] suggests that these vortices could be the consequence of the capsule experiencing unsteady lift. In fact, the "fish tail" motion of the wake closure point influences forces that act on the body and a new starting vortex is subsequently deposit into the wake. This study seems to support the proposed mechanism.

5. CONCLUSIONS

Unsteady flow around an Apollo capsule has been computed in Direct Numerical Simulations (DNS) using a new hybrid spectral/finite element solver. Results show very good agreement with the quantitative wind tunnel and qualitative water tunnel experiments. The study also confirms that the flow around the Apollo capsule is largely independent of Reynolds number for Re>1000, as the flow separation is determined by the geometry. The implementation of Large-Eddy Simulations (LES) subgrid models could allow simulations to be performed at higher Reynolds numbers.

Moreover, this study reveals interesting conclusions concerning unsteady phenomena which act on the body. An unsteady force in the lateral direction is shown for the first time by the present direct numerical simulations. A more detailed analysis should be done to confirm these observations and to study their influence on the capsule stability.

Apollo was chosen as the representative geometry to compare results with those in the literature. Good agreements with the literature give us the sufficient confidence to apply similar simulations to other capsule shapes such as the Huygens. The impact of external perturbations such as wind gust loads could also be the subject of further studies.

6. ACKNOWLEDGMENTS

The numerical study has been done at the Université Libre de Bruxelles (Belgium) as part of a student final year research project. The Experimental work has been conducted in Von Kármán Institute for Fluid Dynamics (Belgium). I thank Professor Gérard Degrez and Yves Detandt who supervised my work at the university and Sébastien Paris, research engineer at the Von Kármán Institute. Special thanks to Dr. Özgur Karatekin for the interest he took in my work.

REFERENCES

1. Karatekin Ö., *Aerodynamics of a planetary entry capsule at low speeds*, PhD. Thesis Université Libre de Bruxelles / Von Kármán Institute for Fluid Dynamics, 2002.
2. Wang F.Y., Karatekin Ö. and Charbonnier J.-M., An experimental study of the flow-field around an Apollo capsule at low speed, *In AIAA 36th Aerospace Sciences Meeting and Exhibit*, Reno (NV), USA, 1998.
3. Karatekin Ö., Wang F.Y., Charbonnier J.-M., Aerodynamic stability of planetary entry capsules in subsonic flow, *International Symposium on Atmospheric Reentry Vehicles and Systems*, Arcachon, France, 1999.
4. Carati D., *Large-eddy simulation for turbulent flows: from mathematical fundations to practical applications*, Thèse d'agrégation ULB, 2001.
5. Charbonnier J.-M., *Plans de la capsule Apollo: sonde oscillante - ensemble et bouclier*, Von Kármán Institute for Fluid Dynamics, 1997.
6. de la Llave Plata M., *Flow field investigation behind a re-entry capsule in VKI-WT1 water tunnel*, Stagiaire report, Von Kármán Institute for fluid dynamics, 1998.
7. Erre A., *Experimental and numerical investigation of a low Reynolds number flow around a 2D Apollo capsule*, Stagiaire report, Von Kármán Institute for fluid dynamics, 1999.
8. Karatekin Ö., Wang F.Y. and Charbonnier J.-M., Terminal aerodynamics of a planetary entry capsule, *2nd Ankara International Aerospace Conference*, Ankara, Turkey, 1998.
9. Karatekin Ö., Wang F.Y. and Charbonnier J.-M., Aerodynamic stability of planetary entry capsules, *3rd European Symposium on Aerothermodynamics for Space Vehicles*, ESTEC, Noordwijk, The Netherlands, 1998.
10. Karatekin Ö., Wang F.Y. and Charbonnier J.-M., Visualisation of an unsteady wake, *In 8th International Symposium on Flow Visualization*, Sorrento, Italy, 1998.
11. Karatekin Ö., Wang F.Y. and Charbonnier J.-M., Near-wake of a three-dimensional bluff body at angles of attack, *In ASME Paper FEDSM98-5185, Proceedings 1998 ASME Fluids Engineering Division Summer Meeting*, Washington DC, 1998.
12. Karatekin Ö., Wang F.Y. and Charbonnier J.-M., Characterisation of a bluff body wake using LDV and PIV techniques, *9th International Symposium on Applications of Laser Techniques to Fluid Mechanics*, Lisbon, Portugal, 1998.
13. Karatekin Ö., Voets R., Charbonnier J.-M. and Wang F.Y., Incompressible aerodynamics of a planetary entry capsule, *17th International Congress on Instrumentation in Aerospace Facilities*, Toulouse, France, 1999.
14. Sagaut P., *Large eddy simulation for incompressible flows - An introduction*, 3rd edition, Springer, 2005.
15. Viré A., *Étude de l'écoulement autour des capsules spatiales*, Mémoire de fin d'études, Université Libre de Bruxelles, Brussels, Belgium, 2006.
16. Wang F.Y., Charbonnier J.-M., Karatekin Ö. and Paris S., The utilization of low speed facili-

ties in transonic stability of reentry vehicles research - An evaluation, *In AIAA 16th Applied Aerodynamics Conference*, Albuquerque (NM), USA, 1998.

17. Wang F.Y., Karatekin Ö. and Charbonnier J.-M., Low-speed aerodynamics of a planetary entry capsule, *Journal of Spacecraft and Rockets*, 36(5):659-667, 1999.

18. Wang F.Y., Karatekin Ö., Charbonnier J.-M. and Paris S., Utilization of low-speed experiments in transonic capsule stability research, *Journal of Spacecraft and Rockets*, 36(5):774-776, 1999.