SPORTS APPLICATIONS FOR CFD ANALYSIS

Yong Ma[1,2], Weitao Zheng[1,2] and Jiurui Han[1]
[1]Wuhan Institute of Physical Education, Wuhan, China
[2]School of Transportation, Wuhan University of Technology, Wuhan, China

Computational Fluid Dynamics (CFD) and the sports applications of CFD were introduced. It was discussed that the level of sports games was enhanced by calculating the performance of equipment by use of CFD. The results indicated that by use of CFD, the simulative computation can be finished in the short period, the relation of the fluid dynamics and the figure of equipment was gained.

KEY WORDS: computational fluid dynamics, performance, equipment

INTRODUCTION: Fluid is a very important branch of sports biomechanics. Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow by means of computer-based simulation. The technique is very powerful and spans a wide range of application areas (Subramani, 2000).
Many sports games are related to fluid, for example, sailing, swimming, rowing, formula one (F1) and so on. It is important to note that for sports games, typical differences of speed between the winning and losing boats are about 1%-2%. Under such conditions, it is clear that a high level of precision is required to predict equipment performance to sufficient accuracy. Such precision places strong demands on both experimental and numerical methods used to determine the forces acting on the equipment. Modern sports games rely on the use of numerical flow simulations to obtain a competitive edge. The computation of the complex hydrodynamic and aerodynamic flow about sports games is obtained by CFD (Subramani, 2000).

COMPUTATIONAL FLUID DYNAMICS: The CFD process begins with geometry, which has to be either created or imported from an external CAD package. The 3D CAD then forms the framework around which a mesh is constructed. The mesh is a set of small blocks, which fill the volume through which the fluid flow. The mesh can contain millions of blocks (or cells). Next, a CFD code applies the governing equations of fluid flow, the Navier-Stokes Equations, to each cell within the mesh. The computer process communicates information across all the cells and proceeds in an iterative manner towards solving the problem (Sakir Bal, 1999; A K. Wiersma, 1979). After much computation a solution is reached where forces and mass flow balance in every cell, and across the whole flow domain. Once the calculation is finished, the CFD solution contains all the pressures and velocities both on and off the surfaces of the object being studied, this is usually millions of numbers. Making sense of such a large data set is not a trivial task and the science of Computer Visualization has developed to deal with this challenge. Modern visualization packages allow the engineer to interactively explore the wealth of data created by the CFD codes. Visualization is the key to fully exploiting the potential of CFD (Subramani, 2000; Zou, Z.J, 1994).
Rapid analysis of designs with Computational Fluid Dynamics aids in the pursuit of maximum aerodynamic efficiency, providing a greater understanding of airflow which can be invaluable for improving performance and producing competitive, efficient, winning products.
Most of the numerical simulations undertaken to date in this field have been based on potential flow theory, which reduces the complexity of the Navier-Stokes equations governing the flow and, consequently, the computational resources required. In particular, a large effort has been devoted to develop reliable tools (such as the panel method) for the computation of the wave resistance, as well as the lift and drag. In some cases the basic hypothesis of the theory (inviscid flow) is satisfactory fulfilled, however, in a number of situations it has been shown that viscosity plays a fundamental role that can not be neglected (Rhie C.M, 1983). Nowadays, computational resources exist that allow the numerical simulation of the complex

viscous flow around three-dimensional bodies, thus improving the accuracy of the computed flow solutions.

SPORTS APPLICATIONS FOR CFD ANALYSIS: When considering a CFD project, computer resources, in-house expertise, employee work-load and the frequency of the need for CFD are all factors to consider. CFD offers a range of Consultancy Services designed to meet the needs of any external aerodynamics and general fluid mechanics problems (Subramani,2000; A.H. Day,1991; Zou, Z.J, 1995).

The use of advanced techniques has become essential in the field of sport. Since a number of years, computational methods - in particular, CFD - have been successfully applied to the design of sport, such as sailing boats, swimming and Formula One (F1). Even though experimentation remains the tool most commonly used by designers to obtain accurate values of the hydrodynamic and aerodynamics forces about sports games, numerical simulations have some major advantages. In particular, they are relatively inexpensive and fast to use, so that it is possible to test and select different candidate geometries before setting up models for the towing tank or wind tunnel. Moreover, they allow the Visualization of several quantities - such as the flow streamlines, the wave profiles or the pressure distribution - that are very difficult to obtain from experiments. This is a very useful aid for the designer to understand the physics of the flow phenomena, at least from a qualitative point of view.

There is a classical example of using CFD, sailing. To challenge the best in this field of sailing, it is necessary to achieve a high standard of technological knowledge, innovation and training. CFD has allowed the sailing to obtain a competitive edge in an application area where small performance improvements result in significant time gains (Rhie C.M, 1983; A.H. Day, 1991; J.H. Milgram,1998; M. Caponnetto,1998,). Detailed numerical studies of sailing are being undertaken in three principle areas: hydrodynamic flow around the hull and the boat appendages, aerodynamic flow around the mast and sails, and the generation of waves on the water surface.

The forces acting on a sailing boat are the aerodynamic force applied to the mast and the sails, the hydrodynamic force applied to the hull and appendages and the force due to gravity. For ideal steady motion, the sums of these forces and of their associated moments are both equal to zero. The sails develop a thrust and a lateral force that are respectively equal to the hydrodynamic resistance and lift generated by the hull, keel and rudder. Moreover, the aerodynamic lateral force and the hydrodynamic lift generate a heeling moment that must be compensated by the righting moment of the hull.

CFD have been employed for a number of years as a design optimization tool for high-performance racing yachts. The mathematical equations describing three-dimensional incompressible turbulent flow are a complex set of coupled partial differential equations, whose resolution requires both accurate and robust numerical methods and substantial computational resources. Applying these equations to sailing is further complicated by the complex physical modeling required to account for hydrodynamic and aerodynamic flow, wave generation on the water surface, and fluid-structure interaction with the mast and sails.

The assumption of inviscid, irrotational flow leads to "potential flow" that can be conveniently solved using traditional panel methods. However, to obtain a competitive edge in an application area where small performance differences can result in significant gains, it is important to account for more complex flow behavior. "Advanced" numerical methods are generally based on the complete viscous flow equations using turbulence modeling, the so-called Reynolds-Averaged Navier-Stokes (RANS) equations. Solving these equations provides detailed insights that - when combined with standard numerical methods, experimental testing, empirical techniques and experience can suggest ways to improve boat performance (Subramani, 2000; Zou, Z.J,1994; Zou, Z.J,1995).

The reduction of drag on the appendages of the sailing boat has been an important area of study for a number of years. For this project, numerical flow simulations are being conducted for different bulb-keel-winglet configurations in order to determine the shape with the least drag (within the applied constraints of weight, structural strength and lift). Such a study

performed for a variety of sailing conditions requires not only numerous simulations but also a significant effort in analyzing the results.
For a number of years, upwind configurations (mainsail and genoa) have been studied using traditional panel methods (J.H. Milgram, 1998; A.H. Day, 1997; M.-C. Sawley, 1997; P.J. Richards, 2001). Due to the strong influence of viscous effects, downwind configurations (mainsail and spinnaker) require the use of RANS simulations, and have therefore received considerably less attention. Nevertheless, the fact that downwind sails are less well understood implies that there is significant potential to improve their efficiency. For the present studies, the sails are considered to be rigid, and therefore the effects of aero-elasticity neglected. Calculations using FLUENT treat the flow around the sails and exposed hull on boat, and also the interaction between two identical boats. By calculating the streamlines, surface pressure and global forces on the boat, the basic physical phenomena are qualitatively and quantitatively examined. In particular, it has been observed that for sailing conditions, the flow on the leeward side of the spinnaker remains partially attached. The spinnaker therefore acts as a combination of a parachute (with the lift aligned with the direction of thrust) and a vertical wing (drag aligned with the direction of thrust) (J.H. Milgram, 1998; P.J. Richards, 2001).
A boat hull is subject to two main resistance components: wave drag and viscous drag (P.J. Richards, 2001b, Ecer, Akin, et. al, 1975). The viscous component is well understood, can be accurately approximated by empirical formulae, and in relation to the hull geometry is most sensitive to the total wetted area. The wave resistance is more difficult to model and is more sensitive to the curvature of the hull geometry (P.S. Jackson. 1982; Cao Y., Schultz W.& Beck R., 1990). In light air conditions, the viscous forces dominate; however as the wind strengthens the larger boat speed leads to a rapid increase in the wave drag that effectively limits the maximum boat speed. Wave drag is being studied for different hull shapes under various sailing conditions using the Princeton code. The blunt bow shape of an IACC boat generates breaking waves that are not possible to treat using the surface tracking technique employed by the code. Modifications have thus been made to suppress numerically the breaking waves. Comparison with experimental data from the towing tank show that the modified code gives accurate values for both resistance components across a wide range of hull speeds. The resistance curves display the same qualitative behavior as predicted by theory (Sakir Bal, 2000; Zou, Z.J, 1995; P.J. Richards, 2001; Ecer, Akin, et. al, 1975).

CONCLUSIONS: Sports biomechanics involves fluid that many sports games are related to. Numerical method by using CFD is an effective tool that is applied to the field of sports. The present study has presented a number of different approaches that have been undertaken for the numerical simulation of the flow around the equipment of sports games. It has been show that, despite the extremely complex flow behavior present in the field of sports games, information gained by using CFD can be gained that is invaluable for design purposes for researchers, athletes, coaches and so on.

REFERENCES:
Subramani, A.K., Paterson, E.G., Stern, F. CFD calculation of sinkage and trim. *Journal of Ship Research* 2000; 44:59-82.
Sakir Bal. A potential based panel method for 2-d hydrofoils. *Ocean Engineering* 1999; 26: 343-361.
A K. Wiersma. On the optimization of the thrust of a yacht sailing to windward. *J. Eng. Math* 1979; 13: 289.
Zou, Z.J. A-3D panel method for the radiation problem with forward speed. *Proc. of 9th International Workshop on Water Waves and Floating Bodies*, Kuju, Oita, Japan, 1994; 251-254.
Rhie C.M., Chow W.L. Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA Journal* 1983; 21: 1525-1532.
A.H. Day. Sail optimization for high speed craft. Trans. RINA 1991; 133: 65.

Zou, Z.J. A-3D numerical solution for a surface-piercing plate oscillating at forward speed. *Proc. of 10th International Workshop on Water Waves and Floating Bodies*, Oxford, UK, 1995; 285-288.

J.H. Milgram. Fluid mechanics for sailing vessel design. *Annual Review of Fluid Mechanics* 1998; 30: 613-653.

M. Caponnetto, A. Castelli, B. Bonjour, P.-L. Mathey, S. Sanchi, M.L. Sawley. America's Cup yacht design using advanced numerical flow simulations. *EPFL Supercomputing Review* 1998: 10: 24-28.

A.H. Day. The Design of Yacht Sailplans for Maximal Upwind Speed. In: *Proc. 12th Chesapeake Sailing Yacht Symp., US Naval Academy*, Annapolis. MD 1995; 97-116.

M.-C. Sawley. FAST2000, UN projet toutes voiles dehors. *EPFL Supercomputing Review* 1997; 9:18-20.

P.J. Richards, A. Johnson, A. Stanton., America's Cup downwind sails - vertical wings or horizontal parachutes. *Journal of Wind Engineering and Industrial Aerodynamics* 2001; 89: 1565-157.

Ecer, Akin, Eichers, Joseph, Bratanow. Theodore, Analysis of three-dimensional potential flow around a ship hull. J. Hydronautics 1975; 9: 64-68.

P.S. Jackson. A 3-D aeroelastic sail model. In: Proc. the Science of Sail Design. London. Ont.. Canada 1982; 55-66.

Cao Y., Schultz W., Beck R. Three-dimension, unsteady computations of nonlinear waves caused by underwater disturbances. 18th Sympo. on Naval Hydro., 1990: 417-425.

Acknowledgement

This work has received financial support from the National Natural Science Foundation of China (Grant No. 10272085) and the Natural Science Foundation of Hubei Province (Grant No. 2005ABA280).