Modelling Swimming Hydrodynamics to Enhance Performance

Daniel A. Marinho¹,², Abel I. Rouboa³, Tiago M. Barbosa²,⁴ and António J. Silva²,⁵,*

¹University of Beira Interior, Department of Sport Sciences (UBI, Covilhã, Portugal)
²Research Centre in Sports, Health and Human Development (CIDESD, Vila Real, Portugal)
³University of Trás-os-Montes and Alto Douro. Department of Engineering (UTAD, Vila Real, Portugal)
⁴Polytechnic Institute of Bragança. Department of Sport Sciences (IPB, Bragança, Portugal)
⁵University of Trás-os-Montes and Alto Douro. Department of Sport, Health and Exercise (UTAD, Vila Real, Portugal)

Abstract: Swimming assessment is one of the most complex but outstanding and fascinating topics in biomechanics. Computational fluid dynamics (CFD) methodology is one of the different methods that have been applied in swimming research to observe and understand water movements around the human body and its application to improve swimming performance. CFD has been applied attempting to understand deeply the biomechanical basis of swimming. Several studies have been conducted willing to analyze the propulsive forces produced by the propelling segments and the drag force resisting forward motion. CFD technique can be considered as an interesting new approach for evaluation of swimming hydrodynamic forces, according to recent evidences. In the near future, as in the present, CFD will provide valorous arguments for defining new swimming techniques or equipments.

Keywords: CFD, swimming, evaluation.

INTRODUCTION

Swimming assessment is one of the most complex but outstanding and fascinating topics in biomechanics. Computational fluid dynamics (CFD) methodology is one of the different methods that have been applied in swimming research to observe and understand water movements around the human body and its application to improve swimming performance. CFD can be considered as a new step forward to the understanding of swimming mechanisms and seems to be an interesting approach to the swimming research.

BACKGROUND OF CFD METHODOLOGY

CFD is a branch of fluid mechanics that solves and analyses problems involving a fluid flow by means of computer-based simulations. CFD methodology consists of a mathematical model that replaces the Navier-Stokes equations with discretized algebraic expressions that can be solved by iterative computerized calculations. The Navier–Stokes equations describe the motion of viscous non-compressible fluid substances. These equations arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term. A solution of the Navier–Stokes equations is called a velocity field or flow field, which is a description of the velocity of the fluid at a given point in space and time. CFD methodology is based on the finite volume approach. In this approach the equations are integrated over each control volume. It is required to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion. In addition, CFD analyses complements testing and experimentation, reducing the total effort required in the experimental design and data acquisition.

In the beginning of its application CFD was quite difficult to use. It was applied only in a few companies of high technological level, namely in the Aerospatiale Engineering or in some specific scientific research areas. It became obvious that its application had to assume a user friendly interface and to progress from a heavy and difficult computation to practical, flexible, intuitive and quick software. Therefore, the following step was to transform CFD in a new set of commercial software to be used in different applications and to help the connection between the user and the computer.

Presently, this tool is used in the resolution of complex engineering problems involving fluid dynamics and it is also being extended to the study of complex flow regimes that define the forces generated by species in self propulsion.

The basic steps of CFD analysis are:

1. Problem identification and pre-processing: (i) define the modelling goals, (ii) identify the domain that wants to model, (iii) design and create the grid.
2. Solver execution: (i) set up the numerical model, (ii) compute and monitor the solution.
3. Post-Processing: (i) examine the results; (ii) consider revisions to the model.

ADVANTAGES AND LIMITATIONS OF CFD

CFD can be used to predict fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving the set of governing mathematical equations. The results of CFD analyses can be relevant in conceptual studies of new designs, detailed product development, troubleshooting and redesign.

Lyttle and Keys [1] referred that CFD can provide the answers into many complex problems which have been unobtainable using physical testing techniques. One of the major benefits is to quickly answer many "what if" type questions. It is possible to test many variations until one arrives at an optimal result, without physical/experimental testing. CFD could be seen as bridging the gap between theoretical and experimental fluid dynamics. For example, with this methodology it is possible to study the aerodynamic of a race car before being constructed or to study the air flow inside the ventilation system of a park station, to simulate situations where a fire takes place, to analyse the ventilation and the acclimatisation of a specific building, such as an hospital where the quality of the air is quite important.

CFD was developed to model any flow fielded provided the geometry of the object is known and some initial flow conditions are prescribed. CFD is based on the use of computers to solve mathematical equation systems. However, it is essential to apply the specific data to characterize the study conditions. Therefore, in the CFD studies the subject who analyzes the problem must be considered. The scientific knowledge, the computational program which solves the equations system representing the problem, the kind of computer that executes the defined calculations in the numerical program and the person who verifies and analyses the obtained results must also be taken in account.

In this sense, one must consider that the CFD analyses can have some inaccurate results if there is not thorough study of the specific situation. The inserted data should not have wide-ranging estimation. On the other hand, the available computational resources can be insufficient to obtain results with the necessary precision. Previous to any simulation, the flow situation must be very well analysed and understood, as well as of the obtained results.

VALIDITY, RELIABILITY AND ACCURACY OF CFD

CFD studies are becoming more and more popular. However, a main concern still persists. Can the numerical data be comparable with experimental research? Are the numerical results accurate enough to be meaningful and therefore have ecological validity? For sport scientists who work in close connection with coaches and athletes this question is important in order to give good, appropriate and individual feedbacks for practitioners.

Several studies within different scopes attempted to verify the validity and accuracy of CFD. This numerical tool has been validated as being feasible in modelling complicated biological fluid dynamics, through a series of stepwise baseline benchmark tests and applications for realistic modelling of different scopes for hydro and aerodynamics of locomotion [2].

In bioscience, Yim et al. [3] described in detail critical aspects of this methodology including surface reconstruction, construction of the volumetric mesh, imposition of boundary conditions and solution of the finite element model. Yim et al. [3] showed the validity of the methodology in vitro and in vivo for experimental biology. Barsky et al. [4] have also demonstrated good agreement between the numerical and experimental data on DNA in flow. Moreover, Gage et al. [5] reported that computational techniques coupled with experimental verification can offer insight into model validity and showed promise for the development of accurate three-dimensional simulations of medical procedures.

In engineering, Venetsanos et al. [6] illustrated an application of CFD methods for the simulation of an actual hydrogen explosion occurred in a built up area of central Stockholm (Sweden) in 1983. The subsequent simulation of the combustion adopted initial conditions for mean flow and turbulence from the dispersion simulations, and calculated the development of a fireball. This data provided physical values that were used as a comparison with the known accident details to give an indication of the validity of the models. The simulation results were consistent with both the reported near-field damage to buildings and persons and with the far-field damage to windows.

In sports some trials have been carried-out to compare the numerical results with experimental results also. A combined CFD and experimental study on the influence of the crew position on the bobsleigh aerodynamics was conducted by Dabnichki and Avital [7]. The experimental results obtained in a wind tunnel suggested that the adopted computational method is appropriate and yields valid results. In what concerns to aquatic sports there is a lack of studies comparing experimental and CFD data. However, CFD was developed to be valid and accurate in a large scope of fluid environments, bodies and tasks, including sports. So, it is assumed that CFD have ecological validity even for swimming research.

Another important concern is related with CFD reliability. In experimental tests, the input data are not always the same and thus the outputs will vary. However, the numerical simulations allow having always the same input conditions and therefore the same outputs.

CFD IN SWIMMING: PRACTICAL CONCERNS

CFD has been applied attempting to understand deeply the biomechanical basis of swimming. Several studies have been conducted willing to analyze the propulsive forces produced by the propelling segments [e.g. 8,9] and the drag force resisting forward motion [e.g. 10,11].

Regarding the propelling forces in swimming, the main CFD results pointed out that:
1. The drag coefficient was the main responsible for the hand and forearm propulsion, with a maximum value of force corresponding to an angle of attack of 90° [12, 13].
2. An important contribution of the lift force to the overall force generation by the hand/forearm in swimming
Modelling Swimming Hydrodynamics to Enhance Performance

The Open Sports Sciences Journal, 2010, Volume 3

6. The gliding position with the arms extended at the front, with the shoulders flexed, presented lower drag coefficient values than the position with the arms placed along the trunk. Considering the breaststroke turn, Marinho et al. [11] suggested that the first gliding, performed with the arms at the front, should be emphasized in relation to the second gliding, performed with the arms along the trunk.

7. The position of the head had a noticeable effect on the hydrodynamic performances, strongly modifying the wake around the swimmer. The position with the head aligned with the body seemed to allow the swimmer to carry out the best water penetration during the underwater swimming phases, comparing with a lower and a higher head position. The head aligned with the axis of the body induces a decrease in the drag from 17% to 21%, for a range velocity from 2.20 m/s to 3.10 m/s [16].

A topic to be developed in the future is the analysis of equipments and facilities using CFD. The repercussion of training equipments (e.g., fins, boards, paddles, pull-buoys) in swim technique, as well, as the swim wear itself (e.g., swim suits, caps, goggles) and facilities (e.g., swim lane design, water flows at the surface and in the bottom of the swimming pool, materials and geometry of head and lateral walls) in performance are an outstanding research opportunity.

CONCLUSION

In summary CFD technique can be considered as an interesting new approach for evaluation of swimming hydrodynamic forces, according to recent evidences. In the near future, as in the present, CFD will provide valuable arguments for defining new swimming techniques or equipments.

REFERENCES


