17

Modelling Hydrodynamic Drag in Swimming using Computational Fluid Dynamics

Daniel A. Marinho[1,2], Tiago M. Barbosa[2,3], Per L. Kjendlie[4],
Narendra Mantripragada[2,5], João P. Vilas-Boas[6,7], Leandro Machado[6,7],
Francisco B. Alves[8], Abel I. Rouboa[9,10] and António J. Silva[2,11]
[1]University of Beira Interior. Department of Sport Sciences (UBI, Covilhã)
[2]Research Centre in Sports, Health and Human Development (CIDESD, Vila Real)
[3]Polytechnic Institute of Bragança. Department of Sport Sciences (IPB, Bragança)
[4]Norwegian School of Sport Sciences. Department of Physical Performance (Oslo)
[5]IIT Kharagpur. Department of Aerospace Engineering (Mumbai)
[6]University of Porto, Faculty of Sport (FADEUP, Porto)
[7]Centre of Research, Education, Innovation and Intervention in Sports (CIFI2D, Porto)
[8]Technical University of Lisbon. Faculty of Human Kinetics (FMH-UTL, Lisbon)
[9]University of Trás-os-Montes and Alto Douro. Department of Engineering (UTAD, Vila Real)
[10]Research Centre and Technologies of Agro-environment and Biological Sciences (CITAB, Vila Real)
[11]University of Trás-os-Montes and Alto Douro. Department of Sport Sciences, Exercise and Health (UTAD, Vila Real)
[1,2,3,6,7,8,9,10,11]Portugal
[4]Norway
[5]India

1. Introduction

In the sports field, numerical simulation techniques have been shown to provide useful information about performance and to play an important role as a complementary tool to physical experiments. Indeed, this methodology has produced significant improvements in equipment design and technique prescription in different sports (Kellar et al., 1999; Pallis et al., 2000; Dabnichki & Avital, 2006). In swimming, this methodology has been applied in order to better understand swimming performance. Thus, the numerical techniques have been addressed to study the propulsive forces generated by the propelling segments (Rouboa et al., 2006; Marinho et al., 2009a) and the hydrodynamic drag forces resisting forward motion (Silva et al., 2008; Marinho et al., 2009b).

Although the swimmer's performance is dependent on both drag and propulsive forces, within this chapter the focus is only on the analysis of the hydrodynamic drag. Therefore, this chapter covers topics in swimming drag simulation from a computational fluid dynamics (CFD) perspective. This perspective means emphasis on the fluid mechanics and

Source: Computational Fluid Dynamics, Book edited by: Hyoung Woo OH,
ISBN 978-953-7619-59-6, pp. 420, January 2010, INTECH, Croatia, downloaded from SCIYO.COM

CFD methodology applied in swimming research. One of the main aims for performance (velocity) enhancement of swimming is to minimize drag forces resisting forward motion, for a given trust. This chapter will concentrate on numerical simulation results, considering the scientific simulation point-of-view, for this practical implication in swimming.

In the first part of the chapter, we introduce the issue, the main aims of the chapter and a brief explanation of the CFD methodology. Then, the contribution of different studies for swimming using CFD and some practical applications of this methodology are presented. During the chapter the authors will attempt to present the CFD data and to address some practical concerns to swimmers and coaches, comparing as well the numerical data with other experimental data available in the literature.

2. Fluid mechanics and CFD methodology

2.1 Background

CFD is a branch of fluid mechanics that solves and analyses problems involving a fluid flow with 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 early days of its application, CFD was quite difficult to use. It was only applied by a few high technological level companies, 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 improve the user interface.

Presently, this tool is used in the solution 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 animal 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 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.

2.2 Advantages and limitations

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 (2006) referred that CFD can provide the answers into many complex problems that have been unobtainable using physical testing techniques. One of its 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. CFD can be applied in several research fields, such as: architecture, engineering, medicine, and sports. For example, with this methodology it is possible to: (i) study the aerodynamics of a racing car before it being constructed; (ii) to study the air flow inside the ventilation system of a park station; (iii) to simulate situations where a fire takes place or; (iv) 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 filed 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. The scientific knowledge, the computational program which solves the equation 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, followed by the careful analysis of the obtained results.

2.3 Validity, reliability, accuracy

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 feed-backs for practitioners.

Several studies with 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 (Liu, 2002).

In bioscience, Yim et al. (2005) 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. (2005) showed the validity of the methodology *in vitro* and *in vivo* for experimental biology. Barsky et al. (2004) have also demonstrated good agreement between the numerical and experimental data on tethered DNA in flow. Moreover, Gage et al. (2002) 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 one can cite, for example, Venetsanos et al. (2003) 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 tests have been performed 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 (2006). The experimental results obtained in a wind tunnel suggested that the adopted computational method is appropriate and yields valid results. In 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 usually assumed that CFD has ecological validity even for swimming research.

Another important concern is that of 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.

3. Hydrodynamic drag

3.1 Definition

Swimming is characterized by the intermittent application of a propulsive force (thrust) to overcome a velocity-dependent water resistance (hydrodynamic drag). The thrust is generated by a combination of arm, leg and body movements and lead to variations of thrust and velocity. Different fluctuations in thrust, drag and velocity among different techniques and different level of skills contribute to the highly variable performance in swimming. Swimming performance can be studied by analysing the interaction of propelling and resistive forces. In this sense, a swimmer will only enhance performance by minimizing resistive forces that act on the swimming body at a given velocity and/or by increasing the propulsive forces produced by the propelling segments. Furthermore, a third performance enhancing factor would be to do this with a minimal enhancement of physiological or energetic costs.

Hydrodynamic drag can be defined as an external force that acts in the swimmer's body parallel but in the opposite direction of his movement direction. This resistive force is depending on the anthropometric characteristics of the swimmer, on the characteristics of the equipments used by the swimmers, on the physical characteristics of the water field, and on the swimming technique (Vilas-Boas, 1996).

The hydrodynamic drag resisting forward motion (D) can be expressed by the Newtonian equation:

$$D = \tfrac{1}{2}\ C_D\, \rho\, S\, v^2 \tag{1}$$

where: ρ represents the fluid density, C_D represents the drag coefficient, S represents the projection surface of the swimmer and v represents the swimming velocity.

The total drag consists of the frictional, form and wave drag components. Frictional drag is depending on water viscosity and generates shear stress in the boundary layer. The intensity of this component is due to the wetted surface area of the body, the characteristics of this surface and the flow conditions inside the boundary layer. Form drag is the result of a pressure differential between the front and the rear of the swimmer, depending on the velocity, the density of water and the cross sectional area of the swimmer. Near the water surface, due to the interface between two fluids of different densities, the swimmer is constrained by the formation of surface waves leading to wave drag (Toussaint & Truijens, 2005).

It is accepted that frictional drag is the smallest component of total drag, especially at higher swimming velocities. However, this drag component should not be disregarded in elite level swimmers. In this sense, issues such as sports equipments, shaving and the decrease of immersed body surface should be taken into account. In addition, form and wave drag represent the major part of total hydrodynamic drag, thus swimmers must emphasize the most hydrodynamic postures during swimming (Toussaint, 2006; Marinho et al., 2009b).

The evaluation of the intensity of the hydrodynamic drag during swimming represents an important aim in swimming biomechanics. Drag determined by towing a non-swimming subject through the water (passive drag) has been studied for a long time (Karpovich, 1933). Passive drag analysis does not consider the drag that the swimmer creates when he produces thrust to overcome the drag, i.e., during actual swimming (active drag). Nevertheless, passive drag seems to be a simple way to investigate the contribution of each drag component to total drag. In addition, passive drag could be used to evaluate drag during parts of the swimming event, namely during the gliding after starts and turns, when the swimmer is "passively" gliding underwater.

3.2 Minimizing drag after start and turns

Minimizing the hydrodynamic drag should be a main concern during swimming. After the push-off during turns and after the block start, the swimmer travels underwater. The first part of this underwater swimming is usually performed without any movements of the propelling segments. Indeed, Lyttle and Keys (2006) reported that at velocities higher than 2.40 m/s it is more efficient for the swimmer to maintain a streamline position than to initiate underwater kicking. This situation is due to the swimmer creating more active drag than propulsion while kicking compared to remaining in a streamlined posture, leading to wasted energy and/or negative acceleration of the swimmer. According to these statements, Lyttle and Keys (2006) suggested that it may be more beneficial to maintain a streamlined position during gliding.

In this sense, the evaluation of the hydrodynamic drag during gliding after starts and turns represents an important question to be addressed. The position of the body segments, such as the head (Zaidi et al., 2008) and the arms (Marinho et al., 2009b), but also different postures adopted during the underwater gliding (Marinho et al., 2009c) have been evaluated using CFD.

Zaidi et al. (2008) evaluated the effect of the head position on hydrodynamic performance, analysing three head positions: (i) head aligned with the body, (ii) a lower head position, and (iii) a higher head position. These three situations were numerically analysed with the swimmer completely submerged in a prone position. Flow velocities of 1.40, 2.20 and 3.10 m/s were used during the simulations. The main results showed that the head position adopted during the gliding should be a main concern of swimmers, since it alters the flow around the body. The head position aligned with the body presented around 20 % less drag than the other two positions for velocities of 2.20 and 3.10 m/s. For a velocity of 1.40 m/s, the difference is small, although the drag force for the lower head and higher head positions is higher than in the aligned head position. It was also interesting to note that the higher head position presented higher values than the lower head position for the three different flow velocities. The numerical results show that the position of the head plays a very important role for high swimming velocities. They reveal that the position of the head has a noticeable effect on the hydrodynamic performances, strongly modifying the wake around the swimmer. Based on a two-dimensional analysis, Zaidi et al. (2008) proposed an optimal position of the head of a swimmer in underwater swimming. However, restrictions inherent to the use of a two-dimensional steady flow model to investigate a really unsteady three-dimensional flow must be kept in mind when analyzing these results.

Marinho et al. (2009b) attempted to analyze the underwater phase after start and turns in a specific technique. In fact, in breaststroke, the first part of the underwater phase is performed with the arms extended at the front of the body, whereas the second gliding is performed with the arms aside the trunk. Therefore, Marinho et al. (2009b) developed a three-dimensional model representing a male adult swimmer in these two gliding positions. The simulations were carried-out with the model placed at a water depth of 0.90 m with flow velocities from 1.60 to 2.0 m/s. The drag coefficient of the position with the arms extended at the front presented lower values than the position with the arms aside the trunk. In fact, the model with the arms at the front of the body presented about 60 % of the hydrodynamic drag values of the position with the arms aside the trunk. It was also interesting to notice that the friction drag component was very similar in both body positions. The pressure drag component was the responsible for the differences between the models, suggesting that the streamlined position with the arms extended at the front of the body lead to decreasing the negative hydrodynamic effects of the human body morphology, especially near the head and shoulders of the swimmer. For the breaststroke underwater phase after start and turns, it was concluded that the first glide, performed with the arms at the front, must be emphasized in relation to the second glide, performed with the arms along the trunk. Vilas-Boas et al. (2009) attempted to analyse the same situations but through inverse dynamics. This procedure was based on the experimental velocity to time gliding curve and the swimmers' inertia performed during the first and second gliding positions of the breaststroke underwater stroke. We were very pleased to observe similar results obtained through CFD and through inverse dynamics. Regarding drag coefficient, Vilas-Boas et al. (2009) reported that the position with the arms at the front (position adopted during the first gliding) presented about 65 % of the drag coefficient values of the position with the arms aside the trunk (position adopted during the second gliding).

Another interesting research concerning the underwater gliding and the most advantageous postures to improve performance was conducted by Marinho et al. (2009c). These authors developed four two-dimensional models to analyse this phase: (i) a ventral position with the arms at the front of the model, (ii) a ventral position with the arms aside the trunk, (iii) a dorsal position with the arms at the front, and (iv) a lateral position with the arms at the front. All these body positions can be used in high level events during the underwater gliding. The four selected postures can be applied to a real swimming situation after the starts and turns, as: the gliding phase in front crawl, butterfly and the first gliding in breaststroke (prone position with the arms extended at the front), the second gliding in breaststroke (prone position with the arms aside the trunk), the gliding in backstroke (dorsal position with the arms extended at the front) and, in some techniques/phases during the gliding in front crawl (lateral position with the arms extended at the front). Marinho et al. (2009c) found that the body postures with the arms extended at the front presented lower drag values than the body posture with the arms aside the trunk. Furthermore, the lateral position was the one in which the drag force was lower. The prone and the dorsal positions (both with the arms extended at the front) presented similar values. Thus, the position with the arms extended at the front (perhaps performed in a lateral position) must be the one adopted after stars and turns. Nevertheless, this issue demands further research using three-dimensional CFD models.

Although the aim of Bixler et al. (2007) study was not the evaluation of different body postures during gliding, this research represented an important contribution to CFD validation in swimming research. Bixler et al. (2007) studied the accuracy of CFD analysis of the passive drag of a male swimmer in a submerged streamlined position. The authors compared the drag force of a real swimmer, a three-dimensional model of this swimmer and a real mannequin based on the digital model. Bixler et al. (2007) found drag forces determined from the digital model using the CFD approach to be within 4 % of the values assessed experimentally for the mannequin, although the mannequin drag was found to be 18 % smaller than the real swimmer drag. Indeed, the Bixler et al. (2007) study has underlined the validity and accuracy of CFD approach in swimming research.

As mentioned above, Lyttle and Keys (2006) studied the underwater phase in swimming. However, contrarily to the previous mentioned studies, these authors were able to perform a CFD analysis providing limb movement. This movement was completed by breaking the limb movements down into discrete time steps and having the package solve the flow field for that position before moving on to the next position. The volume mesh was also updated at each time step with the previous flow field being the starting point at the next time step. Therefore, the authors were able to evaluate two different dynamic dolphin kicking techniques used also during the underwater phase: (i) a large/slow kick (0.54 m of kick amplitude and 2.27 Hz of kick frequency), and (ii) a small/fast kick (0.42 m of kick amplitude and 2.63 Hz of kick frequency). Lyttle and Keys (2006) simulated velocities of 1.50, 2.18 and 2.40 m/s and reported that both kicking techniques have a similar effect at 2.40 m/s. For velocities lower than 2.40 m/s the large/slow kick appears more effective, with about 4 % better efficiency at 2.18 m/s and about 18 % more efficiency at 1.50 m/s. Although these data showed that the large/slow kick has produced better results, one should be aware that these results are based only on the two kicking patterns analyzed and can not be generalized to the large number of possible kicking patterns used by the swimmers. Lyttle and Keys (2006) also showed the benefits of using a modelling approach in the area of technique modification strategies. To illustrate the capabilities of the CFD approach, various simulations were carried-out by varying ankle movement in order to examine the effects on the swimmer's net thrust. The main results showed that while the swimmer is travelling at 2.18 m/s, a 10º increase in ankle plantar flexion created a 16.4 N greater peak propulsive force during the kick cycle. However, with 10º more dorsi-flexion, the peak drag increased by 31.4 N, showing that increasing ankle flexibility will increase the efficiency of the stroke. Nevertheless, this information should be carefully read, since this analysis referred to a specific male swimmer.

3.3 Tandem effects

The tandem concept in swimming is related to situations where a swimmer displaces himself immediately behind another. During swimming events, these tandem effects do not occur since swimmers performed alone in their own lane. However, in open water competitions this situation is very common. Furthermore, during swimming training, due to time and space issues, several swimmers train in the same lane, performing significant parts of total swimming volume "in roundabout" (Silva et al., 2008).

Some experimental studies reported that the distance between swimmers significantly influences the energy cost of the swimmer submitted to the suction effect (Bassett et al., 1991; Hausswirth et al., 1999, 2001; Chatard and Wilson, 2003) and it helps proper technique maintenance when fatigue appears (Chollet et al., 2000). Hence, CFD analysis of the tandem effects on drag force represented an opportunity to help swimmers and coaches improving training sets and also improving performance. Another important aim is to clearly understand the hydrodynamic differences of swimming in the front of the group or behind another.

Silva et al. (2008) aimed to determine the effect of tandem distance on the drag coefficient in swimming. A k-epsilon turbulent model was implemented in the commercial code Fluent® and applied to the fluid flow around two swimmers in a tandem situation. CFD simulations were conducted for various distances between swimmers (from 0.50 to 8.0 m) and swimming velocities (from 1.60 to 2.0 m/s). Silva et al. (2008) computed the drag coefficient for each of the distances and velocities. As expected, these authors found that the relative drag coefficient of the back swimmer was lower (about 56 % of the leading swimmer) for the smallest inter-swimmer distance (0.50 m). This value increased progressively until the distance between swimmers reached 6.0 m, where the relative drag coefficient of the back swimmer was about 84 % of the leading swimmer. Due to some limitations of this study, mainly the simulation domain having small dimensions, it was not possible to numerically accomplish one aim of this study: to determine the distance in which both swimmers performed in the same hydrodynamic conditions, i.e., the distance in which the drag coefficient of the back swimmer is equal to the drag coefficient of the leading swimmer. In fact, for distances higher than 6.0 m, the values of the drag coefficient of the back swimmer remained constant. In this sense, to calculate the distances in which the drag coefficient of the back swimmer's equalled the value of the leading swimmer, a fitting of the drag coefficient curves of the back swimmer was carried out (according to a polynomial function of the values found until the 6.0 m distance). Silva et al. (2008) indicated that the drag coefficient of the back swimmer was equal to that of the leading swimmer at distances ranging from 6.45 to 8.90 m, depending on flow velocity, concluding that these distances allow the swimmers to be in the same hydrodynamic conditions during training and competitions. Regarding specific swimming training sets, Silva et al. (2008) suggested that the back swimmer must start swimming only when the leading swimmer reaches a 10 m distance from the starting wall, rather than the 5 m distance commonly used in training. Nevertheless, concerning open water competitions, the athletes could take important advantages of swimming in a drafting situation.

Although the important findings of the study of Silva et al. (2008), it presented some limitations that should be improved in future studies. The model used during the CFD simulations was a two-dimensional model and the analysis was performed with the model totally submerged. Further studies are needed to evaluate these tandem effects with more realistic models (three-dimensional models) and with the models at the water surface. The inclusion of the interface between air and water seems to be an important concern to be accomplished in the future. Additionally, movements of the propelling segments can be added to the simulations.

Although the most common tandem effects are related to queue displacements, it has been reported that lateral side effects can also be observed during swimming (Janssen et al., 2009). For instance, during competitions it is usual to observe competitors swimming near the lane rope, in a lateral position and a little bit behind the swimmer of the near lane. It is supposed that this option can benefit the swimmer that follows in this lateral and behind position. However, in a recent experimental study, Janssen et al. (2009) reported that at the side of a passive lead swimmer, passive drag was significantly increased by 9 %, and at the side of an active lead swimmer it increased by 8 %. This is in opposition to the significantly reduction of passive drag observed behind a passive lead swimmer (20 %) and behind an active lead swimmer (9 %). Therefore, it should be interesting to perform a similar procedure using CFD methodology.

3.4 Form, friction and wave drag components

The contribution of form, friction and wave drag components to total drag during swimming is an interesting topic in sports biomechanics. Data available from several experimental studies show some difficulties involved in the evaluation of the contribution of each drag component. However, CFD has the advantage of allowing the computation of these drag components, letting the user to perform the desired simulations and to know exactly what is the intensity of form, friction and wave drag in different swimming simulation situations (Bixler et al., 2007).

Nevertheless, to the best of our knowledge, there is no CFD studies that have been able to compute wave drag in swimming. In fact, CFD studies in swimming were only able to analyse form and friction drag components, since the models were placed underwater. Bixler et al. (2007) simulated a human body placed at a water depth of 0.75 m, Zaidi et al. (2008) positioned the model 1.50 m below the water surface, whereas Marinho et al. (2009b) used a model at a water depth of 0.90 m. These distances were experimentally proven to be sufficient to exclude the wave drag component from the simulations (Lyttle et al., 1999; Vennell et al., 2006). Therefore, it seems very interesting to improve CFD simulations, including the analysis of hydrodynamic drag when wave drag is a real phenomenon. To achieve this purpose the inclusion of the air and water in the same computational domain is required.

Bixler et al. (2007) found that friction drag represented about 25 % of total drag when the swimmer is gliding underwater; stating that although form drag was dominant, friction drag should be taken in consideration by swimmers and coaches. Zaidi et al. (2008) also found an important contribution of friction drag to the total drag. These authors, used a two-dimensional model with the head in different positions, and found that friction drag represented about 20 % of the total drag. In the study of Zaidi et al. (2008) it was very interesting to note that the position with the head aligned with the body presented lower form and friction drag values than the lifted up head position and the lowered head position. Although the lifted up and lowered head positions presented similar values of friction drag, the lifted up position presented higher form drag values than the lowered head position, showing that the lifted up head position lead to a higher pressure gradient around the swimmer during the gliding.

Marinho et al. (2009b), when analysing two different gliding postures, found a bit lower values of friction drag. However, these authors reported contributions of 13 % and 8 % for the friction drag component in the position with the arms extended at the front and in the position with the arms along the trunk, respectively. Nevertheless, Marinho et al. (2009b) showed that differences in the drag force between these two body positions were only related to different form drag values. Indeed, the absolute values of friction drag were about the same in the two gliding positions. Thus, this finding underlined the idea that the so-called streamline position allowed decreasing the pressure gradient around the swimmer body during the underwater gliding.

It is important to emphasize that, if the swimmer model was at the water surface, rather than gliding underwater, the contribution of each drag component is expected to be different. For instance, one should be aware that in this situation wetted area would be lower, thus decreasing friction drag.

3.5 Equipments

The influence in performance of the equipments used during swimming is not a clear issue and, usually, the methodological design involved during the experiments lead to some difficulties in the data analysis.

However, the numerical analysis with CFD can be a good approach to overcome this problem. For instance, different human body models can be tested wearing different swimsuits, fins, paddles, caps, goggles, to evaluate the effects in hydrodynamic drag.

The most known application of CFD in swimming research is probably the numerical analysis of different types of swimsuits and the testing procedures that lead to the development of new swimsuit models (Fluent, 2004). These data suggest that the polyurethane generation of swimsuits can significantly improve performance due to the reduction in hydrodynamic drag.

It is usually accepted that drag-reducing suits can reduce skin friction, with an effect similar to shaving (Sharpe & Costill, 1998; Pendergast et al., 2006). Nevertheless, Mollendorf et al. (2004) revealed that total drag decreased by 3 % to 10 % mostly due to decreased form drag in textile suits. These experimental data suggest that the water flow was tripped by frictional drag, remained attached to the swimmer body, thus decreasing form drag (Polidori et al., 2006; Marinho et al., 2009b). Pendergast et al. (2006) stated that studies of the effects of a drag reducing textile suit on active drag at low to moderate velocities failed to show a clear benefit, although at the fastest velocity the textile suit reduced the drag of some swimmers (Sanders et al., 2001; Toussaint et al., 2002). Other authors used physiological approaches, and the results were controversial as well (Starling et al., 1995; Roberts et al., 2003). All these data suggests that CFD can represent an additional solution to clear this issue and help swimmers improve their performance. CFD has the advantage to show the water flow around the swimmer body, allowing understanding if the water really remained attached to the body. Moreover, an important issue that has not been systematically evaluated is the compression effects due to swimsuits. The compression of the body may decrease the area projected in the frontal plane, being this variable a major determinant of form drag. Furthermore, the advantage of swimsuits upon wobbling body masses represent an important opportunity to future research in this field, especially after the FINA changing rules regarding swimsuits to 2010 swimming events.

Another interesting concern is related to the flow visualization around the fins. Swimmers wearing fins can swim much faster than without this device. Research on the flow characteristics around the fins still needs more attention (Tamura et al., 2002). On the other hand, the effects on hydrodynamic drag by wearing different fin types (e.g., on size and/or flexibility) and with different kick movements can allow enhanced performance during training sets with fins and also during fin swimming events.

Following this line of research, effects of different caps and goggles can be tested using numerical simulation techniques. The effects of different swimming pools on drag (depth, width lane, number of lanes, lane ropes, and lane position) should also be attempted in the near future.

4. Future research in swimming using CFD

Throughout this chapter, several future ideas have been presented to improve the application of CFD in swimming research. One of our major aims is to be able to evaluate biomechanical situations that can be used by coaches and swimmers to swim faster and, thus to enhance performance. Therefore, the effective evaluation of active drag should be one of the first concerns in future studies using CFD. At this point, it seems important to analyse the intensity of active drag in the four competitive strokes within a wide range of swimming velocities. On the other hand, CFD can compute the contribution of friction, form and wave drag components to total drag. This issue will only be possible if the simultaneous simulation of the interface between air and water within the same CFD domain is achieved.

The tandem issues also represent a significant matter that can be more deeply understood. If the above mentioned questions are solved, one can simulate several current tandem situations that occur during training but also during competitions, as in open-water events where tandem situations are common. Rear and lateral positions occur in this type of events and the lateral position (in an adjacent lane) can also be possible in swimming events, thus its effects on hydrodynamic drag must be quantified.

As mentioned above, the analysis of the effects of different equipments and facilities on hydrodynamic drag seems to be an interesting and an important issue to be dealt in future studies.

5. Conclusion

During this chapter, the authors attempted to present some important studies that have been conducted in swimming research using CFD. Although there are some limitations of these studies, it seems that this numerical tool should not be disregarded. CFD can be used to evaluate several hydrodynamic issues, hence helping swimmers moving faster. In the current work some issues regarding the effect of hydrodynamic drag on swimming performance were discussed. We believe that we were able to show the practical applications of CFD to swimmers and their coaches.

Moreover, several questions to be addressed in future investigations were reported. These concerns represent an important step forward to bridge the gap between theory and practice, allowing even more the scientific knowledge to be available to swimmers and coaches.

6. Acknowledgement

This work was supported by the Portuguese Government by Grants of the Science and Technology Foundation (PTDC/DES/098532/2008).

7. References

Barsky, S.; Delgado-Buscalioni, R. & Coveney, P.V. (2004). Comparison of molecular dynamics with hybrid continuum-molecular dynamics for a single tethered polymer in a solvent. *Journal of Chemical Physics*, 121, 2403-2411.

Bassett, D.R.; Flohr, J.; Duey, W.J.; Howley, E.T. & Pein, R.L. (1991) Metabolic responses to drafting during front crawl swimming. *Medicine and Science in Sports and Exercise*, 23, 6, 744-747.

Bixler, B. & Schloder, M. (1996). Computational fluid dynamics: an analytical tool for the 21st century swimming scientist. *Journal of Swimming Research*, 11, 4-22.

Bixler, B.; Pease, D. & Fairhurst, F. (2007). The accuracy of computational fluid dynamics analysis of the passive drag of a male swimmer. *Sports Biomechanics*, 6, 81-98.

Chatard, J.C. & Wilson, B. (2003). Drafting distance in swimming. *Medicine and Science in Sports and Exercise*, 35, 7, 1176-1181.

Chollet, D.; Hue, O.; Auclair, F.; Millet, G. & Chatard, J.C. (2000). The effects of drafting on stroking variations during swimming in elite male triathletes. *European Journal of Applied Physiology*, 82, 5-6, 413-417.

Dabnichki, P. & Avital, E. (2006). Influence of the position of crew members on aerodynamics performance of two-man bobsleigh. *Journal of Biomechanics*, 39, 2733-2742.

Fluent (2004). Speedo goes for gold with CFD. *Fluent News*, 13, 1, 4-6.

Gage, K.L.; Gartner, M.J.; Burgreen, G.W. & Wagner, W.R. (2002). Predicting membrane oxygenator pressure drop using computational fluid dynamics. *Artificial Organs*, 26, 600-607.

Hausswirth, C.; Lehenaff, D.; Dreano, P. & Savonen, K. (1999). Effects of cycling alone or in a sheltered position on subsequent running performance during a triathlon. *Medicine and Science in Sports and Exercise*, 31, 4, 599-604.

Hausswirth, C.; Vallier, J.M.; Lehenaff, D.; Brisswalter, J.; Smith, D.; Millet, G. & Dreano, P. (2001). Effect of two drafting modalities in cycling on running performance. *Medicine and Science in Sports and Exercise*, 33, 3, 485-492.

Janssen, M.; Wilson, B.D. & Toussaint, H.M. (2009). Effects of drafting on hydrodynamic and metabolic responses in front crawl swimming. *Medicine and Science in Sports and Exercise*, 41, 4, 837-843.

Karpovich, P. (1933). Water resistance in swimming. *Research Quarterly*, 4, 21-28.

Kellar, W.P.; Pearse, S.R.G. & Savill, A.M. (1999). Formula 1 car wheel aerodynamics. *Sports Engineering*, 2, 203-212.

Liu, H. (2002). Computational biological fluid dynamics: digitizing and visualizing animal swimming and flying. *Integrative and Comparative Biology*, 42, 1050-1059.

Lyttle, A. & Keys, M. (2006). The application of computational fluid dynamics for technique prescription in underwater kicking. *Portuguese Journal of Sport Sciences*, 6, Suppl. 2, 233-235.

Lyttle, A.D.; Blanksby, B.A.; Elliott, B.C. & Lloyd, D.G. (1999). Optimal depth for streamlined gliding, In: *Biomechanics and Medicine in Swimming VIII*, K.L. Keskinen, P.V. Komi & A.P. Hollander (Eds.), 165-170. Gummerus Printing, Jyvaskyla.

Marinho, D.A.; Rouboa, A.I.; Alves, F.B.; Vilas-Boas, J.P.; Machado, L.; Reis, V.M. & Silva, A.J. (2009a). Hydrodynamic analysis of different thumb positions in swimming. *Journal of Sports Science and Medicine*, 8, 1, 58-66.

Marinho, D.A.; Reis, V.M.; Alves, F.B.; Vilas-Boas, J.P.; Machado, L.; Silva, A.J. & Rouboa, A.I. (2009b). The hydrodynamic drag during gliding in swimming. *Journal of Applied Biomechanics*, 25, 3, 253-257.

Marinho, D.A.; Carvalho, F.; Barbosa, T.M.; Reis, V.M.; Alves, F.; Rouboa, A. & Silva, A.J. (2009c). The determination of drag in the gliding phase in swimming, In: *Proceedings of the 27th International Conference on Biomechanics in Sports*, R. Anderson, D. Harrison & I. Kenny (Eds.), 620. University of Limerick, Limerick.

Mollendorf, J.C.; Termin, A.C.; Oppenheim, E. & Pendergast, D.R. (2004). Effect of swim suit design on passive drag. *Medicine and Science in Sports and Exercise*, 36, 6, 1029-1035.

Pallis, J.M.; Banks, D.W. & Okamoto, K.K. (2000). 3D computational fluid dynamics in competitive sail, yatch and windsurfer design, In: *The Engineering of Sport: Research, Development and Innovation*, F. Subic & M. Haake (Eds.), 75-79. Blackwell Science, Oxford.

Pendergast, D.R.; Capelli, C.; Craig Jr, A.B.; di Prampero, P.E.; Minetti, A.E.; Mollendorf1, J.; Termin, A. & Zamparo, P. (2006). Biophysics in swimming. *Portuguese Journal of Sport Sciences*, 6, Suppl. 2, 185-189.

Polidori, G.; Taiar, R.; Fohanno, S.; Mai, T.H. & Lodini, A. (2006). Skin-friction drag analysis from the forced convection modeling in simplified underwater swimming. *Journal of Biomechanics*, 39, 2535-2541.

Roberts, B.S.; Kamel, K.S.; Hedrick, C.E.; MLean, S.P. & Sharpe, R.L. (2003). Effect of Fastskin suit on submaximal freestyle swimming. *Medicine and Science in Sports and Exercise*, 35, 519-524.

Rouboa, A.; Silva, A.; Leal, L.; Rocha, J. & Alves, F. (2006). The effect of swimmer's hand/forearm acceleration on propulsive forces generation using computational fluid dynamics. *Journal of Biomechanics*, 39, 1239-1248.

Sanders, R.B.; Rushall, H., Toussaint, H., Steager, J. & Takagi, H. (2001). Bodysuit yourself but first think about it. *American Swimming Magazine*, 5, 23-32.

Sharpe, R.L. & Costill, D.L. (1998). Influence of body hair removal on physiological responses during breaststroke swimming. *Medicine and Science in Sports and Exercise*, 21, 576–80.

Silva, A.J.; Rouboa, A.; Moreira, A.; Reis, V.; Alves, F.; Vilas-Boas, J.P. & Marinho, D. (2008). Analysis of drafting effects in swimming using computational fluid dynamics. *Journal of Sports Science and Medicine*, 7, 1, 60-66.

Starling, R.D.; Costill, D.; Trappe, T.A.; Jozsi, A.C.; Trappe, S.W. & Goodplaster, B.H. (1995). Effect of swimming suit design on the energy demands of swimming. *Medicine and Science in Sports and Exercise*, 27, 1086–1089.

Tamura, H.; Nakazawa, Y.; Sugiyama, Y.; Nomura, T. & Torii, N. (2002). Motion analysis and shape evaluation of a swimming monofin. *The Engineering Sports*, 4, 716-724.

Toussaint, H.B. (2006). Technology applied to optimise training for improving of front crawl performance. *Portuguese Journal of Sport Sciences*, 6, Suppl. 2, 205-210.

Toussaint, H. & Truijens, M. (2005). Biomechanical aspects of peak performance in human swimming. *Animal Biology*, 55, 1, 17-40.

Toussaint, H.M.; Truijens, M.J.; Elzinga, M.J.; van de Ven, A.; de Best, H.; Snabel, B. & de Groot, G. (2002). Effect of a Fast-skin "body" suit on drag during front crawl swimming. *Sport Biomechanics*, 1, 1–10.

Venetsanos, A.G.; Huld, T.; Adams, P. & Bartzis, J.G. (2003). Source, dispersion and combustion modelling of an accidental release of hydrogen in an urban environment. *Journal of Hazardous Materials*, 105, 1-3, 1-25.

Vennell, R.; Pease, D.L. & Wilson, B.D. (2006). Wave drag on human swimmers. *Journal of Biomechanics*, 31, 664-671.

Vilas-Boas, J.P. (1996). *Bases Mecânicas da Natação*. Faculty of Sport Sciences and Physical Education of Porto University, Porto.

Vilas-Boas, J.P.; Costa, L.; Fernandes, R.; Ribeiro, J.; Figueiredo, P.; Marinho, D.; Silva, A.; Rouboa, A. & Machado, L. (2009). Determination of the drag coefficient during the first and second glide positions of the breaststroke underwater stroke. *Journal of Applied Biomechanics*, in press.

Yim P.; Demarco, K.; Castro, M.A. & Cebral, J. (2005). Characterization of shear stress on the wall of the carotid artery using magnetic resonance imaging and computational fluid dynamics. *Studies in Health Technology and Informatics*, 113, 412-442.

Zaidi, H.; Taiar, R.; Fohanno, S. & Polidori, G. (2008). Analysis of the effect of swimmer's head position on swimming performance using computational fluid dynamics. *Journal of Biomechanics*, 41, 1350-1358.

Computational Fluid Dynamics
Edited by Hyoung Woo Oh

ISBN 978-953-7619-59-6
Hard cover, 420 pages
Publisher InTech

Published in print edition January, 2010

How to reference

Daniel A. Marinho, Tiago M. Barbosa, Per L. Kjendlie, Narendra Mantripragada, João P. Vilas-Boas, Leandro Machado, Francisco B. Alves, Abel I. Rouboa and António J. Silva (2010). Modelling Hydrodynamic Drag in Swimming using Computational Fluid Dynamics, Computational Fluid Dynamics, Hyoung Woo Oh (Ed.), ISBN: 978-953-7619-59-6, InTech, Available from: http://www.intechopen.com/books/computational-fluid-dynamics/modelling-hydrodynamic-drag-in-swimming-using-computational-fluid-dynamics

InTech Europe
University Campus STeP Ri
Slavka Krautzeka 83/A
51000 Rijeka, Croatia
Phone: +385 (51) 770 447
Fax: +385 (51) 686 166
www.intechopen.com

InTech China
Unit 405, Office Block, Hotel Equatorial Shanghai
No.65, Yan An Road (West), Shanghai, 200040, China
中国上海市延安西路65号上海国际贵都大饭店办公楼405单元
Phone: +86-21-62489820
Fax: +86-21-62489821