Applied Mathematics and Computation 302 (2017) 48–57

Contents lists available at ScienceDirect

Applied Mathematics and Computation

journal homepage: www.elsevier.com/locate/amc

Preliminary numerical investigation in open currents-water swimming: Pressure field in the swimmer wake

Fabien Beaumont, Redha Taïar*, Guillaume Polidori

Laboratoire de Thermomécanique GRESPI- EA4694, Université de Reims, 51687 Reims cedex 2, France

ARTICLE INFO

Keywords:
Open-water swimming
Drafting
Computational fluid dynamics (CFD)
k-ω turbulence model
Drag forces

ABSTRACT

The drafting in swimming is a sports practice usually used during the open water race of triathlon. Indeed, it is well known that swimming drafting behind or close to the lead swimmer induces a reduction in drag, once the draft swimmer is located in the low pressure field of the lead swimmer's wake. For this purpose, in order to determine the best position for the draft swimmer, a preliminary perfect knowledge of the lead swimmer's wake is necessary in presence or not of currents. The present preliminary study focuses this point before developing further analyses on reducing drag and enhancing performance in drafting by considering both lead and draft swimmers. In this work, the computational fluid dynamics (CFD) method was used in order to evaluate the hydrodynamic drag during the swimmer's displacement in presence of currents. The standard k-ω turbulence model was chosen to predict the resistance forces with currents-negative ($\alpha = 0°$: aligned position of the swimmer's body relative to the main flow direction) and cross-currents (angle of incidence of the swimmer's body relative to the main flow direction, respectively 10 and 20°). The analysis of the CFD results have shown that the best position for a draft swimmer was found to be directly behind the lead swimmer (close to the toes) and very close to the side of the lead swimmer when drafter's head is between shoulders and hip level of the leader.

1. Introduction

The open water swimming is practiced outdoor in stretches of water such as lakes, rivers … All those swimming events held in open water are covered by long-distance swimming (it's the reason why it is called open water swimming). The triathlon begins by the swimming event and helped making more popular the open water swimming. What is very different from swimming in a pool is the presence of current, wave, wind, and poor visibility. Swimmers may head off course when covering large distances. It is generally easy to swim with current-positive (i.e., current-assisted), the currents which push you forward. Inversely, current-negative or cross-currents (i.e., current-handicapped) swims are harder to handle, in particular when swimmers are advised of their negative effects on their progression [1]. Swimming against a current can take considerably longer to cover a same distance and may generate much more fatigue than swim in calm water. Drafting can be to the swimmer's advantage in an open water swim. It is now well known that swimming drafting leads to a reduction in energy spent to overcome drag forces (friction, wave and pressure) and enhances gains time in competitive swimming [2,3]. In swimming, the flow past a swimmer induces a trailing adverse pressure gradient area in its near wake.

* Corresponding author.
E-mail address: Redha.taiar@univ-reims.fr (R. Taïar).

http://dx.doi.org/10.1016/j.amc.2016.12.031
0096-3003/

Swimming drafting behind and close to another swimmer consequently leads to a reduction in pressure drag induced by the lead swimmer.

Drafting or staying on the slipstream of the swimmer in front of is legal in the swim part of a triathlon. The slipstream is the area behind the swimmer as they progress through the water. In this work, it can be done the assumption that, whether the direction of currents changes during the swimmer's displacement, there will be substantial differences in the magnitude of the total drag force as well as differences in the relative contribution of the components forces, viscous, form and wave drag. These differences would be mainly due to the changes in exposed frontal area with increasing the angle between the swimmer's and the main water flow direction [4].

The drag force components generated by the swimmer's displacement have been analyzed by means of different experimental methods [5,6]. However, in reason of the difficulties involved with experimental research, quantitative data obtained through these studies have to be considered cautiously [7]. An alternative approach that can be used to analyze the swimmer's displacement is the employ of a numerical simulation method, such as computational fluid dynamics (CFD). Bixler et al. [7] proposed a study whose data showed the accuracy and validity of this method used in the swimming research field. Since then, this numerical method has been regularly used to analyze the water flow around the human body and the forces which occur during the swimmer's displacement [8–13]. More recently, Zaïdi et al. [14] have investigated the flow around a 3D swimmer geometry using a CFD code. In order to predict the resistance forces during the underwater swimming phase, the 3D analysis was performed using two models of turbulence, namely the standard k-ε turbulence model and the standard k-ω turbulence model. The comparison between experimental data of drag forces and numerical results has evidenced that the standard k-ω model accurately predicts the drag forces while the standard k-ε model underestimates their value, both in 2D and 3D analyses. Regarding the swimming drafting where several swimmers are considered, most of the studies conducted on this topic are experimental ones [15]. Actually, only one study has focused on 2D swimming drafting by numerical ways [13].

Most of CFD studies on the swimmer's displacement were conducted in swimming pool conditions (i.e. without current and with a water temperature which is constant) and results have shown differences due to body shape, body position or swimmer velocity. However, no CFD study has focused on the open-water swimming and its specificity (currents, waves, cold water…). Therefore, the present study was undertaken to quantify the effect of currents-negative on forces acting on the swimmer's body in open water swimming using the CFD method.

This work is a logical continuation of previous studies by Zaïdi et al. [14,16,17]. In our current study, the standard k-ω turbulence model is used to predict the resistance forces in open water swimming with currents-negative and cross-currents.

Fig. 1 presents the three position of swimmer's body (relative to the water flow direction) that we investigated in the present work. In the first case ($\alpha = 0°$), the athlete swims parallel to the water flow direction but in its opposite direction (i.e. counter current). In the other two studied position, the athlete swims in an opposite direction to that of the flow but with an angle of incidence of respectively 10 and 20° (i.e. cross current). In a swimming pool, α has always a zero value not depending on the swimmer direction, and also not depending on his time-varying motion (the swimmer moves in a fluid which is always at rest). In lakes or in the see where lateral currents can occur, α has a non-zero value. In our numerical methodology, we choose as realistic hypothesis a steady-state lateral flow. In that case, the steady calculation remains sufficient to investigate the flow field.

In order to simplify the simulation, the swimmer's body is totally immerged and the effect of surface wind waves, generally present on water surface in open water competition, is not considered. In such a case, we do the assumption that the contribution of wave drag on total drag is less predominant. Even if this situation is different from that observed at the water surface, the results of this work will gives us some clues for further studies on swimming drafting.

Therefore, a preliminary perfect knowledge of the lead swimmer's wake is necessary in presence or not of currents to determine the best position for the draft swimmer. The present preliminary study focuses this point before developing further analyses on reducing drag and enhancing performance in drafting by considering both lead and draft swimmers.

2. Mathematical formulation

2.1. Reynolds-averaged Navier–stokes equations

It is assumed that the flow around the swimmer is fully turbulent [5,18-21]. The Reynolds-averaged Navier–Stokes equations are used to describe turbulent flows. These equations are obtained by introducing the Reynolds decomposition, which consists in considering that, in turbulent flows, an instantaneous quantity is decomposed into its time-averaged and fluctuating quantities. Then, the time-averaged solutions to the Navier–Stokes equations can be written [21]:

Continuity equation

$$\frac{\partial}{\partial x_i}\left(\overline{U_i}\right) = 0 \tag{1}$$

Navier–Stokes (momentum) equations

$$\frac{\partial}{\partial x_j}\left(\rho \overline{U_i U_j}\right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i}\right) - \rho \overline{u_i u_j}\right] \tag{2}$$