Turbulence model choice for the calculation of drag forces when using the CFD method

H. Zaïdi a, S. Fohanno a,*, R. Taïar b, G. Polidori a

a Université de Reims, GRESPI/Laboratoire de Thermomécanique, Moulin de la Housse, BP 1039, 51687 Reims cedex 2, France
b Université de Reims, Laboratoire d’Analyse des Contraintes Mécaniques, Moulin de la Housse, BP 1039, 51687 Reims cedex 2, France

A R T I C L E I N F O

Article history:
Accepted 5 October 2009

Keywords:
Underwater swimming
Turbulence model
Computational fluid dynamics

A B S T R A C T

The aim of this work is to specify which model of turbulence is the most adapted in order to predict the drag forces that a swimmer encounters during his movement in the fluid environment. For this, a Computational Fluid Dynamics (CFD) analysis has been undertaken with a commercial CFD code (Fluent®). The problem was modelled as 3D and in steady hydrodynamic state. The 3D geometry of the swimmer was created by means of a complete laser scanning of the swimmer’s body contour. Two turbulence models were tested, namely the standard k–ε model with a specific treatment of the fluid flow area near the swimmer’s body contour, and the standard k–ω model. The comparison of numerical results with experimental measurements of drag forces shows that the standard k–ω model accurately predicts the drag forces while the standard k–ε model underestimates their values. The standard k–ω model also enabled to capture the vortex structures developing at the swimmer’s back and buttocks in underwater swimming; the same vortices had been visualized by flow visualization experiments carried out at the INSEP (National Institute for Sport and Physical Education in Paris) with the French national swimming team.

© 2009 Elsevier Ltd. All rights reserved.

1. Introduction

The Computational Fluid Dynamics (CFD) method is increasingly used in the domain of biomechanics. This concerns a wide range of topics such as cardiovascular (Kim et al., 2006; Maurits et al., 2007), respiratory (Longest and Vinchurkar, 2007; Mylavarapu et al., 2009) or sports (Bixler and Riewald, 2002; Meile et al., 2006; Dabnichki and Avital, 2006) biomechanics.

In the specific field of swimming, the CFD method has been used these last years to study the flow around a swimmer’s body or extremities. Before being applied to swimming, this method was used by Bixler and Schloder (1996) whose aim was to simulate the flow around a disc, which had the same surface as the one of a swimmer’s hand. Since then, the CFD method has been used by several authors in the swimming field. Bixler and Riewald (2002) first used the CFD method to simulate the water flow around a swimmer’s hand and forearm. The aim of their study was to calculate the drag forces and coefficients around a swimmer’s hand and forearm in the case of different angles of attack. Rouboa et al. (2006) estimated numerically the drag and lift coefficients for a swimmer’s hand and forearm in both the steady and unsteady state cases. They also evaluated the effect of the acceleration of the hand and forearm on the generation of the propulsion force. Gardano and Dabnichki (2006) performed numerical simulations in order to highlight the importance of the flow analysis around the whole arm of a swimmer, so that the exact values of the propulsion and the drag forces were estimated. Bixler et al. (2007) used the CFD method to estimate the resistance forces encountered by the swimmer during the underwater phase and to study the effect of wearing a wetsuit on the drag forces met by the swimmer. More recently, Zaïdi et al. (2008a) studied the effect of the head position on the hydrodynamic performance during the underwater starting phase (after the start dive) or following a turn.

Examining thoroughly the literature allowed to notice that most numerical works about human swimming use the standard k–ε model of turbulence in the calculation of the drag forces. The purpose of the present work is to calculate the drag forces using both the standard k–ε and the standard k–ω turbulence models in order to certify which of the two models is best adapted to the evaluation of the drag forces encountered by the swimmer during the underwater starting phase and during the return phases after a turn.

2. Mathematical formulation

2.1. Reynolds-averaged Navier—stokes equations

The flow around the swimmer is naturally turbulent (Clarys, 1979; Ungerechts, 1983; Toussaint and Truijens, 2005; Polidori.
et al., 2006; Zaidi et al., 2008a). The turbulent flow is governed by the Reynolds-averaged Navier–Stokes equations. These equations are obtained by introducing the Reynolds decomposition, which consists in considering that, in turbulent flows, each instantaneous variable is the sum of a mean component and a fluctuating component. Then, the time-averaging of the instantaneous equations leads to the following system of averaged equations (Zaidi et al., 2008a):

Continuity equation

$$\frac{\partial \bar{\rho}}{\partial x_i} = 0$$ (1)

Navier–Stokes (momentum) equations

$$\frac{\partial}{\partial x_j} \left( \rho \bar{U}_j \right) = - \frac{\partial \bar{p}}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \rho \bar{U}_i \bar{U}_j \right) - \rho \frac{\partial}{\partial x_i} \bar{U}_j \frac{\partial}{\partial x_j} \bar{U}_i - \rho \bar{U}_i \frac{\partial}{\partial x_j} \bar{U}_j + \rho \bar{f}_i$$ (2)

with \( \bar{U}_i(t) = \bar{U}_i + u_i \), the instantaneous velocity component in the \( i \) direction (m/s), \( \bar{U}_j \) the mean (time-averaged) velocity component in the \( j \) direction (m/s), \( u_i \) the fluctuating velocity component in the \( i \) direction (m/s), \( i, j \) are the directions, \( \rho \) the fluid density (kg/m³), and \( \mu \) the fluid dynamic viscosity (kg/ms).

2.2. Turbulence modelling

2.2.1. Boussinesq hypothesis

New terms of double correlations between the fluctuations of the velocity components (\( -\rho \bar{U}_i \bar{U}_j \)), called Reynolds stresses, appear in the averaged equations. These new terms are additional unknowns that need to be modelled in order to close the system of equations to be solved. The Boussinesq hypothesis can be used to model the Reynolds stresses (Zaidi et al., 2008a). This hypothesis consists in the introduction of a turbulent viscosity (\( \mu_t \)) that directly links the correlations with the time-averaged velocity components.

Boussinesq hypothesis

$$-\rho \bar{U}_i \bar{U}_j = \mu_t \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k$$ (3)

with \( \mu_t \) the turbulent viscosity (kg/ms); \( \kappa = (1/2)\rho \bar{U}_i \bar{U}_i \) the turbulent kinetic energy per unit mass (m²/s²); \( \delta_{ij} \) the Kronecker symbol; \( \delta_{ij} = 1 \) if \( i=j \) and \( \delta_{ij} = 0 \) if \( i \neq j \).

The turbulent viscosity is not a property of the fluid itself but depends on the dynamic characteristics of the turbulent flow. In this work, the turbulent viscosity is modelled by means of first-order models based on the time-averaged dynamical characteristics of the turbulent flow (Zaidi et al., 2009). The two models chosen for this study are the standard \( k-\epsilon \) model, which is widely used in CFD, and the standard \( k-\omega \) model, which is well-suited to detect recirculations (Zaidi et al., 2009). Each model requires two additional transport equations.

2.2.2. Standard \( k-\epsilon \) model

The standard \( k-\epsilon \) model is widely used in CFD. As concerns the CFD analysis of human swimming, most studies carried out in this field have used this model (Bixler and Riewald, 2002; Rouboa et al., 2006; Gardano and Dabnichki, 2006; Zaidi et al., 2008a).

The equations for the standard \( k-\epsilon \) model are as follows:

$$\frac{\partial \bar{U}_i \bar{k}}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu_t \frac{\partial \bar{k}}{\partial x_i} \right) + P_k - \epsilon$$ (4)

$$\frac{\partial \bar{c}_e \bar{k}}{\partial x_i} = \frac{1}{\rho} \frac{\partial}{\partial x_i} \left( \mu_t \frac{\partial \bar{c}_e \bar{k}}{\partial x_i} \right) + c_1 \frac{\epsilon}{k} P_k - c_2 \frac{c_e^2}{k}$$ (5)

with \( \sigma_k, \sigma_\epsilon, c_1, c_2, \sigma_t \): constants; \( \epsilon \): dissipation rate of the turbulent kinetic energy per unit mass (m²/s³)

$$P_k = \frac{\mu_t}{\rho} \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \frac{\partial \bar{U}_j}{\partial x_j}$$

Production rate of turbulent energy per unit mass (m²/s³)

The turbulent viscosity is calculated by the following relation:

$$\mu_t = \rho \sigma_k \frac{k^2}{\epsilon}$$ (7)

where \( \sigma_k \) is a constant.

The constants of the standard \( k-\epsilon \) model have been determined from experiments (Cousteix, 1989). These values are summarized in Table 1.

Furthermore, turbulent flow characteristics are significantly affected by the presence of the swimmer’s body while the standard \( k-\epsilon \) model assumes a fully turbulent flow in the whole fluid domain. Therefore, a specific law, namely the “non-equilibrium wall-function law”, has been chosen for the treatment of the fluid flow area near the swimmer’s body contour (Bixler and Riewald, 2002; Bixler et al., 2007; Zaidi et al. 2008a, 2009). In the following, the generic term “wall” (commonly used in CFD) will be employed to designate the swimmer’s body contour.

2.2.3. Standard \( k-\omega \) model

One of the weak points of the standard \( k-\epsilon \) turbulence model concerns the modelling of near-wall region where the dissipation rate (\( \epsilon \)) is difficult to specify. The standard \( k-\omega \) model aims at overcoming this difficulty by solving the transport equation of the specific dissipation rate (\( \omega \)) instead of the dissipation rate (\( \epsilon \)). This model is well-suited to wall-bounded flows like the flow around the body contour of the swimmer.

The standard \( k-\omega \) model transport equations are

$$\frac{\partial}{\partial x_j} (\rho k \bar{\omega}_j) = \frac{\partial}{\partial x_j} \left( \gamma_k \frac{\partial \bar{c}_o}{\partial x_j} \right) + P_k - Y_k$$ (8)

$$\frac{\partial}{\partial x_j} (\rho \omega \bar{U}_j) = \frac{\partial}{\partial x_j} \left( \gamma_\omega \frac{\partial \bar{c}_o}{\partial x_j} \right) + P_o - Y_o$$ (9)

where \( \Gamma_k, \Gamma_\omega \) are the effective diffusivities of \( k \) and \( \omega \), respectively. \( Y_k, Y_o \) are the turbulent dissipations of \( k \) and \( \omega \), respectively.

The turbulent viscosity is then calculated by the following equation:

$$\mu_t = \sigma \frac{k^2}{\omega}$$ (10)

where \( \sigma \) is a coefficient bringing a correction to the turbulent viscosity for low Reynolds numbers.

3. Numerical method

The numerical simulations of the flow were carried out by means of the Fluent® commercial CFD code. This code is based on the Patankar finite volume method (Patankar, 1980) to solve the system of equations governing the fluid flow around the swimmer.
3.1. Construction of the swimmer’s geometry

The subject chosen to realize the study is a national-level female swimmer. The equipment used in the construction of her 3D geometric model is a Konica Minolta® scanner, which is commanded by a laser scanning process. The scanner allows the creation of data files containing all the coordinates of all the points defining the layer on the surface of the swimmer’s body. The RapidForm® software has been used in the construction of the swimmer’s geometry using the data file containing the clouds of points coming from the scanner. Fig. 1 shows the front and the back of the swimmer before and after the scanning.

3.2. Construction of the fluid domain around the swimmer

After constructing the swimmer’s geometry, the next stage consists in building the fluid domain around the swimmer. The size of the fluid domain was chosen with respect to the parametric study by Zaidi et al. (2008a) especially concerning the upstream and downstream distances from the body, namely 3 m upstream and 9.6 m downstream. Fig. 2 schematizes the size of the fluid area built around the swimmer.

3.3. Grid of the fluid domain

The grid of the fluid area was realized by means of the TGrid® software; it is a progressive three-dimensional grid, refined near the surface of the swimmer and loose when it is far so that the computational time will not grow heavily. Fig. 3 shows an example of the meshing of the external boundaries of the fluid domain.

Fig. 4 shows the surface meshing of the swimmer’s body and a cross-section of the fluid area around the swimmer.

When a non-structured grid is used, the flow is not lined up with the grid cells. Therefore, in order to limit the numerical dissipation, second-order numerical schemes are used. The convergence criterion is chosen as equal to $10^{-5}$ (Zaidi et al. 2008a).

3.4. Boundary conditions

The boundary conditions adopted for the numerical simulations are as follows:

- At the entrance of the fluid domain: a longitudinal uniform velocity profile ($U_0$) is imposed.
- At the exit of the fluid area: mass conservation law (all the gradients equal zero).
- On the upper, lower, left, and right borders of the fluid domain: the condition of symmetry is imposed.
- On the surface of the swimmer: the no-slip condition is imposed.

Remark. Physical observations show that a viscous fluid such as water remains in contact with solid surfaces at the body/fluid interface. The no-slip condition ensures that this observation is respected by imposing no relative motion or slip velocity between the fluid (water) and the swimmer’s body at the body/fluid interface.

Fig. 1. Back and front views of the swimmer before and after the scanning.

Fig. 2. Fluid domain built around the swimmer.
4. Results and discussions

4.1. Turbulence model choice—Drag forces

To simplify the analysis, the underwater swimming situation is considered, so that the wave drag is negligible. In such a situation, the total drag is composed of the skin-friction drag and the profile drag. At first, the total drag is calculated for the same velocities as that used in Bixler et al. (2007), namely $U_0=1.5$, 1.75, 2 and 2.25 m/s. Both turbulence models (standard $k$-$\varepsilon$ and standard $k$-$\omega$) are used for the present 3D analysis.

CFD results are compared with two series of experiments carried out by Bixler et al. (2007) and Vennell et al. (2006). In their study, Bixler et al. (2007) experimentally determined total drag forces in a submerged streamlined position. As concerns Vennell et al. (2006), their study mainly focused on the contribution of the wave drag for several tow depths up to 1 m. However, they also
observed that the wave drag was no longer significant for tows deeper than 0.7 m at a swimmer’s speed of 2 m/s. Therefore, from their experimental results, they could deduce the total drag for fully immersed bodies corresponding to the underwater swimming situation of the present study. These fully immersed drag results will be used for comparison with the present CFD results.

Fig. 5 shows a comparison between the total drag obtained with the present numerical study using both the standard $k$–$\varepsilon$ and the standard $k$–$\omega$ turbulence models and the experimental ones obtained by Bixler et al. (2007) and Vennell et al. (2006). Fig. 5 shows that, with a 3D geometry and the standard $k$–$\omega$ turbulence model, the numerical results do match with the experimental ones obtained from the tests carried out by Bixler et al. (2007) and Vennell et al. (2006). On the other hand, one notes that the standard $k$–$\varepsilon$ model underestimates the resistance force faced by the swimmer during the underwater phase. The conclusion is that the standard $k$–$\omega$ model is the best one to predict the drag forces.

After choosing the standard $k$–$\omega$ turbulence model to calculate the total resistance forces met by the swimmer, the numerical simulations were carried out again for the same velocity values as the ones chosen by Zaïdi et al. (2008a), that is 1.4, 2.2, and 3.1 m/s at the entrance of the fluid domain.

Fig. 6 presents the evolutions of the skin-friction drag, the profile drag and the total drag as a function of the swimmer’s speed. Fig. 6 shows that the profile drag is the main component of drag and also that the skin-friction drag is not negligible. The skin-friction drag actually represents 20.9% of the total drag for a swimmer’s speed of 1.4 m/s and 16.4% for a speed of 3.1 m/s. These results show that the contribution of the skin-friction drag
to the total drag decreases with increase in the speed. This matches the results obtained by Bixler et al. (2007). The latter indicate that the skin-friction drag represents 27% of the total drag for a speed of 1.5 m/s and 25% for a speed of 2.25 m/s.

Remark: In Fig. 7 a comparison is shown between the numerical results of the present 3D study with the k-ω model and the 2D study carried out by Zaïdi et al. (2008a, 2009) using both the k-ε and k-ω turbulence models. To complete this comparison, the 3D experimental results obtained by Bixler et al. (2007) and Vennell et al. (2006) are also presented. One observes that the total drag values calculated in the case of the 2D study are much higher than the ones calculated in the case of the 3D study, whatever model of turbulence has been used. So, one can conclude that the 2D study appears to be limited as it does not enable to evaluate correctly the drag forces, and that the 3D study is unavoidable in swimming analysis.

4.2. Vortex structures visualized with the standard k-ω turbulence model

The numerical simulation, using the standard k-ω turbulence model, allowed capturing separation zones level with the back and the buttocks (Figs. 8 and 9) of the swimmer. To prove that the k-ω model combined with the 3D CFD analysis qualitatively enables to visualize the flow, flow visualization experiments have been conducted at the INSEP with top-level swimmers from the France swimming team (Zaïdi et al., 2008b). The visualization method used was the tuft method; the tufts accurately follow the flow direction and are able to indicate the separation zones. These structures were visualized level with the back and the buttocks. Figs. 8 and 9 show a comparison between the vortical structures visualized at the INSEP and the ones captured by the numerical calculation using Fluent®. The observation shows that the fluid situated level with those two swirling areas is driven by a rotating motion. Near the buttocks, one notes two symmetrical vortical parietal areas, characteristic from contra-rotating cells.

5. Conclusion

In this work, the CFD method was used to specify which model of turbulence is the most adapted in order to predict the resistance forces that the swimmer faces during the underwater swimming phase and when swimming back after a turn. Two models were tested: the k-ε standard model and the k-ω standard turbulence models.

The k-ω standard model accurately predicts the drag forces while the k-ε standard model underestimates the values of the drag force. Moreover, in such complex fluid dynamics problem where complex phenomena appear, the k-ω model was the only one to be able to model vortex structures and separation areas. This work has also shown that the 2D study is insufficient and that the 3D study is inevitable. Considering the results obtained in this study, the use of the CFD method combined with the k-ω standard model of turbulence is encouraging to improve the achievement in the field of human swimming.

Conflict of interest

The authors (Hanane Zaïdi, Stéphane Fohanno, Redha Taïar and Guillaume Polidori) do not have any conflict of interest

References


