THE USE OF COMPUTATIONAL FLUIDS DYNAMICS TO OPTIMISE UNDERWATER KICKING PERFORMANCE

Andrew Lyttle and Matt Keys
Western Australian Institute of Sport, Perth, WA, Australia

Elite swimmers use a variety of underwater kicking patterns in current competition with little scientific information used in their selection. The current study sought to discriminate between 2 different patterns of underwater dolphin kick (large amplitude, slow kicks versus small amplitude, fast kicks) using computational fluid dynamics (CFD). Inputs into the CFD model included an accurate 3D mapping of an elite swimmer and detailed kinematic information of the underwater kicking from a 2D analysis. Initial results of the static CFD model were in agreement with previous empirical testing of passive drag. Results of the dynamic CFD modelling and comparisons are still to be finalised.

KEY WORDS: underwater kicking, computational fluid dynamics.

INTRODUCTION: The underwater phases of swimming form a large component of the total event time in modern swimming and can often lead to the difference between a medal and finishing with the rest of the pack. An optimal underwater start and turning technique requires maximizing the distance achieved through minimizing the deceleration caused by hydrodynamic drag forces. Currently in elite competition, there exists a large range of underwater technique strategies utilized by the swimmers with very little scientific rationale applied in their selection. Previous empirical testing conducted by Lyttle et al. (2000) has examined the net force produced during underwater kicking due to the complexities in separating the propulsive force and active drag. Results were compared to prone streamlined gliding in order to prescribe an approximate velocity at which to initiate underwater kicking. The study assumed steady state (constant velocity) conditions which limited the applicability to real swimming where the body is continually accelerated and decelerated.

The use of high level computational fluid dynamics (CFD) allows variables such as the amplitude and frequency of the kicking movement and the velocity and depth interactions to be examined from a fluid dynamics perspective. CFD allows complex fluid flow regimes and geometry to be simulated using known physics, and provides visualisation of the resulting variables across the entire solution domain. This can provide answers and insights into problems which have been unobtainable using physical testing techniques. As such, CFD could be seen as bridging the gap between theoretical and experimental fluid dynamics. CFD is based on the fundamental governing equations of fluid dynamics - the continuity, momentum, and energy equations. The actual equations applied are selected with due regard for the flow regime of the simulation (eg. Navier-Stokes for viscous, Euler for inviscid etc.).

The current study seeks to discriminate between the active drag and propulsion generated in underwater kicking with the goal of prescribing an optimal kick profile in swim starts and turns. This enables objective information to be used in technique analysis.

METHODS: Required inputs into the CFD model consist of 2 important components. The first is the detailed 3D mapping of the swimmer body shape and the second is an accurate description of the movement pattern under investigation. An elite national level swimmer from the Western Australian Institute of Sport was filmed underwater from a sagittal view while performing underwater dolphin kicks at maximal effort. The swimmer performed both high amplitude, low frequency dolphin kicks and low amplitude, high frequency dolphin kicks. The kinematics of these underwater kicks were of similar magnitudes to that found in current elite competition. One of each underwater kicking pattern was selected based on similar average velocity and depth over the kick cycle between the 2 trials. A full 2D analysis was performed to determine segment kinematics for the 2 selected trials (see Figure 1).

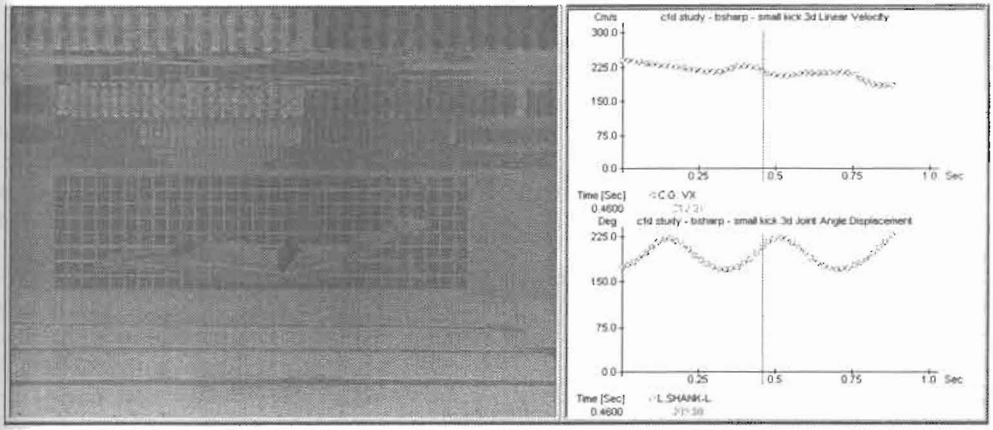

Figure 1: Sample kinematics from underwater dolphin kicking trial.

The 3D mapping of the swimmer was performed using a Cyberware WBX whole body laser scanner with a density of one point every 4mm. Higher resolution scans were conducted of the hands and feet using casts of these limbs. The higher resolution scans were then aligned and merged seamlessly into the full body scan to provide more accuracy at these locations. All scans were performed with the swimmer assuming a streamlined glide position with hands overlapping and feet plantar-flexed (see Figure 2).

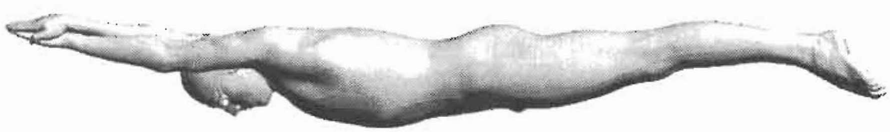

Figure 2: The 3D scanned image of the swimmer.

A computational fluids dynamics package (Fluent) was used to model the fluid flow around the upper body as well as the animated lower limbs. This CFD study was broken into two stages, a simulation of the swimmer without limb movement (static model), and one including limb movement (dynamic model). The purpose of the first stage was to allow benchmarking of swimmers drag to previous experimental drag test results (Lyttle et al., 1998, 2000). The same model was used in the second stage with the addition of user defined functions and re-meshing to provide limb movement.

Before creating the CFD model, a number of assumptions were made. This allowed the model to be solved in a reasonable time frame while still maintaining the salient characteristics of the flow. The assumptions and simplifications made in this study are listed below.

1) The model was single phase with no air/water interface. The 0.5m depth to swimmer was increased to 1.5m to reduce any confinement effects on the flow due to this assumption.

2) The width of pool included in the model was 3m. This is considered sufficiently far to have negligible effects on the results.

3) The pool floor was modelled 1.5m below the centre of the swimmer.

4) A 5m length of pool was assumed which should provide sufficient length past the swimmer to allow convergence of the model and not affect results.

5) The domain was assumed to be moving at the average speed of the swimmer so that the swimmer remains relatively stationary. This was achieved via an upstream inlet, a downstream outlet and moving side walls.

The CFD process requires geometric construction of the simulation to define the extent of the domain to be investigated. This was achieved by subtracting the swimmer (3D solid model

generated from the laser scan) from a block representing the section of pool being simulated. The laser scanning of the swimmer included approximately 1 million surface grid points defining triangular faces capturing the surface geometry. This information was processed to extract 288 NURBS curved surfaces forming a 3D solid model of the swimmer.
The domain surfaces were meshed with varying mesh densities to pick up the detail around highly curved areas while still maintaining a workable mesh size. The current surface mesh on the swimmer was 60,000 triangular surface elements and the total simulation comprises approximately 2 million control volumes. The figure below represents the surface mesh around the head of the swimmer (see Figure 3).

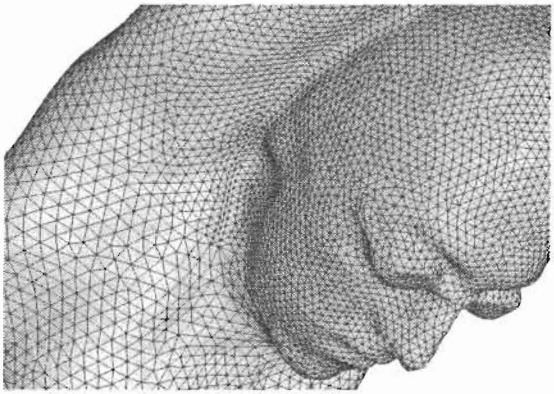

Figure 3: Sample surface mesh around the swimmer's head.

Boundary conditions are applied to simulate the adjacent conditions outside the domain. The assumed domain cross section gives an area ratio of 9m2 pool to 0.13m2 area of the swimmer, 70 times greater which allows for minimal increase in velocities around the swimmer due to the confined nature of flow in the domain.
The governing equations of fluid flow for the problem were integrated over the control volumes of the solution domain. Finite difference approximations were substituted for the terms in the integrated equations representing the flow processes. This converted the integral equations into a system of algebraic equations that were solved using iterative methods. Convergence of the solution was judged by inspecting and plotting summed residuals of the variables over the domain and variables at critical monitor points.
The second stage analysis (dynamic model) was completed by breaking the limb movements down into discrete time steps and having the simulation 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 used as the starting point for the next time step. The results of the analysis were viewed using the post processing visualisation tools to present any of the variables at locations of interest.

RESULTS AND DISCUSSION: Summary results of the kinematic analysis are listed below for the 2 underwater kicking conditions and are compared with data from International swimmers (Arellano et al., 2002). Results demonstrate the clear difference in amplitude and frequency between the 2 selected underwater kicks. A comparison with data from Arellano et al. (2002) shows a higher underwater kicking aptitude for the subject in this study, which may have been due to the inclusion of females within the average international data.

Table 1 Descriptive kinematic variables.

Derived Kinematic Variables	Large/Slow Kick	Small/Fast Kick	Average International
Kick Amplitude (vertical displacement of toe) (m)	0.54	0.42	0.62
Horizontal distance of toe in 1 kick cycle (m)	0.98	0.77	0.76
Average Horizontal CM Velocity (ms^{-1})	2.16	2.13	1.61
Maximum Knee Flexion (°)	121.7	139.9	113.7
Kick Frequency (Hz)	2.27	2.63	2.14
Strouhal Number	0.57	0.52	0.81

Preliminary results of stage 1 analysis (Static Model) are presented below (see Figure 4). All velocities are relative to the swimmer that is travelling at 2ms-1 in the horizontal direction. Pathlines are used to visualise the flow of massless particles in the problem domain. These particles have been released from a surface upstream of the hands and demonstrate the varying velocities of the flow over the swimmer. The contour plots are also coloured by velocity magnitude and demonstrate the effected area of flow around the swimmer.

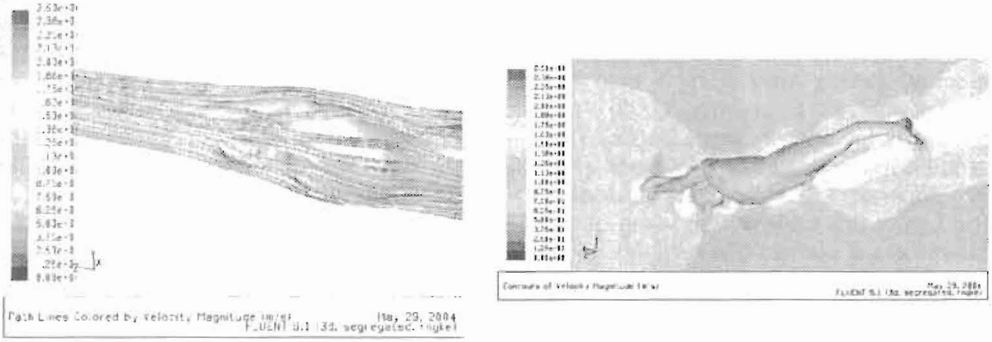

Figure 4: Visualisation of results of the static model.

CONCLUSIONS: Initial results have indicated that the CFD model is providing valid and reliable results. This will enable us to differentiate flow lines and calculate propulsive and resistive coefficients. Once the CFD model has been completed, alterations to the inputs and model constraints can also be investigated to examine the effects of modifications in underwater kicking technique as well as morphology variations. This sort of analysis can then be used as a precursor to further investigations into this area by building on the fluid dynamics model. The objective information provided by this type of analysis equips the sports scientists with the tools to more accurately provide advice on technique modifications in order to gain the extra edge at the elite level.

REFERENCES:

Lyttle, A.D., Blanksby, B.A.B., Elliott, B.C., & Lloyd, D.G. (1998). The effect of depth and velocity on drag during the streamlined glide. Journal of Swimming Research, 13, 15-22.

Lyttle, A.D., Blanksby, B.A.B., Elliott, B.C., & Lloyd, D.G. (2000). Net forces during tethered simulation of underwater streamlined gliding and kicking techniques of the freestyle turn. Journal of Sports Science, 18, 801-807.

Arellano, R., Pardillo, S., & Gavilan, A. (2002). Usefulness of the Strouhal Number in evaluating human underwater undulatory swimming. In J.C. Chatard (Ed.), Proceeding of the IX International Symposium on Biomechanics and Medicine in Swimming (pp 213-218). France: University of Saint-Etienne.