Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 00 (2013) 000–000

Procedia Engineering

www.elsevier.com/locate/procedia

The 2014 conference of the International Sports Engineering Association

Fluid Mechanics in rowing: the case of the flow around the blades

Yoann ROBERT[a], Alban LEROYER[a,*], Sophie BARRÉ[b], François RONGÈRE[a], Patrick QUEUTEY[a], Michel VISONNEAU[a]

[a]*LHEEA, Ecole Centrale de Nantes, CNRS UMR 6598, BP 92101, 44321 Nantes Cedex 3, France*
[b]*CREPS Pays de la Loire, 4 Place Gabriel Trarieux, 44319 Nantes Cedex 3, France*

Abstract

The aim of this research is to develop hydrodynamic models to enhance the knowledge of the propulsion efficiency in rowing. The flow around a rowing blade is a complex phenomenon characterised by an unsteady 3D behaviour, with violent free surface deformation including breakup and with a flexible shaft driven by a 6-DOF movement. The study uses experimental results obtained on an instrumented boat to perform CFD computations. All parameters are considered except the minor role played by the spin rotation of the shaft. The numerical results fit fairly well with experimental data given a high number of uncertainties. Once CFD computations fully validated, more accurate parametric models could be built and integrated in a rowing simulator which will help coaching staff in analysing and improving performance and training of athletes. Another considered possibility is the direct coupling between the rowing simulator and the CFD code.

Selection and peer-review under responsibility of the Centre for Sports Engineering Research, Sheffield Hallam University.

Keywords : rowing ; oar blade; free surface flow ; computational fluid dynamics (CFD) ; fluid-structure interaction (FSI)

1. Introduction

Fluid mechanics plays a key role in nautical sports, mostly in complex situations. This is particularly true for rowing, for which the fluid flow knowledge is required for both the boat hull and the oar blades. The movement of oarsmen during the rowing stroke induces large variations in speed together with heave and pitch motions of the boat. It makes this flow quite uncommon in naval hydrodynamics. However, the flow around the blade is far more complex. The goal of this communication is to describe the numerical and experimental state of the research

* Corresponding author. Tel.: +33 2 40 37 16 48; fax: +33 2 40 57 25 23.
E-mail address: alban.leroyer@ec-nantes.fr

1877-7058 © 2014 The Authors. Published by Elsevier Ltd.
Selection and peer-review under responsibility of the Centre for Sports Engineering Research, Sheffield Hallam University.

carried out at the LHEEA Lab. (research Laboratory in Hydrodynamics, Energetics and Atmospheric Environment) of the École Centrale de Nantes to study the flow around rowing blades, with a special focus on the validation process of the numerical model. This work benefits from the growing power of the computational resources and the development of advanced numerical methods. CFD (Computational Fluid Dynamics) simulations are becoming increasingly popular and are more and more used for complex applications in naval hydrodynamics. In comparison with experimentations, the numerical approach is able to provide extra useful information, especially in full scale and for complex situations where physical experimentation reaches limits, although a particular attention must be paid to generating confident results.

During the last decade, free-surface capturing methods (volume of fluid, level-set formulation) have attracted an increasing interest due to their robustness and because no regridding is necessary. Moreover, the merging or breakup of the interface is handled in a natural way. Such approaches have been extensively validated for flows around hulls in calm water or with waves in Queutey and Visonneau (2007). For the majority of these cases, the evolution of the free surface remains quite limited. Here, the goal is to use and to validate such an approach for a far more violent test-case, namely the flow around a rowing blade. In this case, the blade moves with 6 DOFs close to the free surface, leading to an unsteady flow with a very complex shape of free surface varying in time and including breakup. It also involves fluid-structure interaction (FSI) through the flexibility of the shaft, considering the motion of the oar is imposed by the rower. In reality, the "boat-oar-rower" global system with the rower as a human actuator is even more complex, since the fluid force acting on the blade will be linked to the kinematics of the rower, since the latter activates a sensory motor control of his own motion in order to adapt it to the current position and external loads. For reference, see Rongère et al. (2011).

To conduct properly this ongoing research work, the experimental approach is used to provide a reference database. Thus, the article describes both aspects of the research carried out at the LHEEA Lab. to study the flow around rowing blades. First, the experimental approach is presented. Then, the numerical methodology will be briefly described prior to the different steps of the validation process we have adopted. To conclude, the next steps of this ongoing work will be revealed.

2. Experimental approach : in-situ measurements

The LHEEA Lab. has carried out experimental studies on rowing boats and oars using specific devices in the towing tank, as well as on-board measurements with an instrumented boat (see Barré and Kobus (2010)). On one hand, towing tank tests are focusing only on one part of the system (the boat or the oar); as a consequence the settings are necessarily simplified with respect to the real case. The main advantage is that the environment and the parameters are well controlled, with good accuracy and repeatability. This technique enables the propulsive device (the oar) to be separated from the motor (the oarsman) and from the boat, as it is done for propellers. On the other hand, on-board measurement works directly on the real system (boat-oar-rower). However, the effective achievement of the procedure involves technological constraints (i.e. weight of the instrumentation), environmental constraints (weather conditions) and human factor (availability of elite rowers).

The real phenomena get closer, but the price to pay is a larger number of parameters, which are sometimes not accurately controlled. Furthermore, only a limited number of measured quantities can be recorded. A loss of accuracy is then expected compared with towing tank experiments.

Measurements with instrumented boats allow the direct analysis of the rowing stroke and the boat behaviour. In order to validate CFD computations on rowing blades, specific sensors have been developed (more accurate than commercial sensors used by coaching staff) and procedures have been adjusted to obtain enough accuracy to reach the objective.

2.1. Measuring the boat dynamics

A specific sensor was developed to measure the boat velocity. It is made of a small Schiltknecht ducted turbine fixed below the fin of the hull. This position ensures that the device is outside the boundary layer of the hull. The turbine with four low inertia blades is mounted on a sapphire-made bearing to reduce friction. This feature improves the latency time of the sensor. The turbine contains a Baumer magnetic sensor, which counts the passing of each blade. It has been calibrated fixed on the boat in the towing tank: the resolution is 0.7 mm of displacement per pulse with an accuracy of 2 cm/s.

2.2. Measurements for the oars

To measure the blade trajectory, the oarlocks are equipped with three accurate potentiometers to measure the oar orientation: the horizontal sweep angle, the immersion angle and the spin angle (rotation around its own axis).

Considering the dynamics of the oar, the following forces are involved: the action of the rower's hand on the handle, the reaction of the oarlock and the hydrodynamic, aerodynamic, inertia and gravity forces. We do not know where the resultant of the hydrodynamic force is applied. To retrieve this information, all the forces and their moments should be measured, which is an arduous task. Considering the main objective which is to validate the numerical simulation, it is enough to measure bending and torsion moments in a section of the outside shaft and to compute the numerical results at the same location. If experiment and simulation match well, the numerical simulations can be used to obtain all the components of the hydrodynamic forces. For the measurements of these moments, the oars have been equipped with three strain gauges: two for the bending in orthogonal directions and the third one for the torsion. The pure hydrodynamic contribution can be obtained by subtracting the inertia effects computed at the gauge positions. These dynamometric oars are calibrated in pair under static loads with a test bench specifically built for this procedure (see Fig. 1(a)). This test bench is also used to measure the linear and angular deflection stiffnesses (see Fig. 1(b)), which are two essential parameters for the CFD simulations taking into account FSI. Measures are made in a horizontal plane. The measurements are made as if a load F was imposed at the blade. The bending moment M at the gauges location can be computed using the arm lever formula. This known value is used to calibrate the gauges. The linear deflection d is then measured and thanks to the relation $M = k_d\, d$, the linear stiffness coefficient k_d (slope of the straight line) is deduced. The angular stiffness coefficient is found with the same protocol through the measurement of the angular deviation α.

(a) (b)

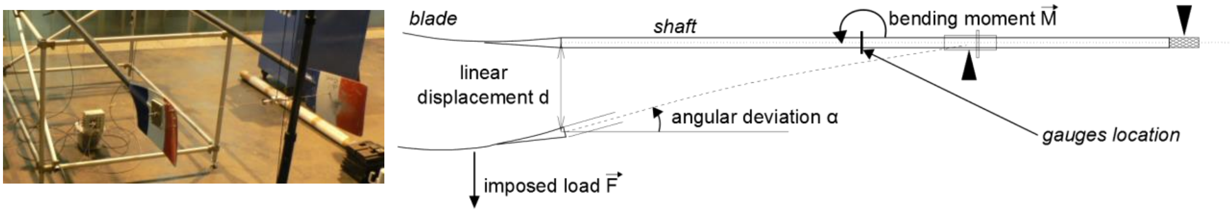

Fig. 1. (a) Test bench to calibrate the dynamometric oars and to measure the flexibility. (b) Linear and angular deviations due to the flexible shaft appearing in a rowing stroke.

2.3. Kinematic data to validate CFD simulations

The angles of the oars and the motion of the boat are transformed into input data in a geometric rowing model used for the simulation of the global system (see Rongère et al. (2011)). As an output, the model gives the motion of the blade with respect to the water frame of reference (see Fig. 2). All the details of the real oar are taken into account, in particular the shift between the three axes of rotation at the oarlock. Until now, the spin of the shaft is the only property which is not activated in the simulation since it is not crucial during the propulsive phase where the blade remains vertical.

2.4. Procedure to detect the contact of the blade with the water during the catch

The measure of the dynamic roll of the boat has not been integrated to the instrumented boat yet. Without this information, the position of the blade with respect to the water is not accurately known. A procedure to detect the instant where the blade is touching the water is thus required, to adjust the free surface location. Also, we have to assume that it remains valid during the whole drive phase, i.e. that the roll does not change, which is not totally true. The procedure of blade/water contact detection is mainly based on the analysis of the Z-bending and X-torsion moments: both are approximately null before the blade touches the water and increase suddenly when the contact occurs.

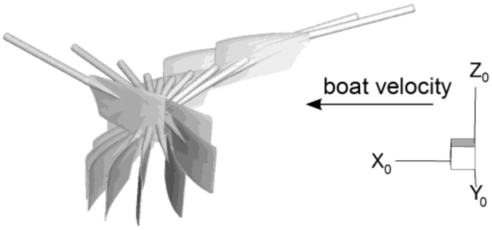

Fig. 2. Kinematics of the blade for the F001 experiment in the water frame of reference (with subscript 0).

3. Numerical Approach

The flow around a rowing blade presents two main features, unsteadiness and a very complex free surface deformation, which made it difficult to investigate for a long time. To investigate such a flow, only an experimental approach seemed possible around ten years ago. But nowadays, advances in computing power offer the possibility to investigate this flow using CFD. And the experimental investigations already available provide a useful and valuable reference database to validate CFD computations. In fact, CFD tools are no longer limited to simple physical problems and their field of applications has been significantly enlarged during the last years.

3.1. Description of the solver

The solver used for those simulations is ISIS-CFD, i.e. the core of the FINE™/Marine computing suite, which is developed by the DSPM team of the LHEEA Lab. ISIS-CFD is an incompressible unsteady RANS solver, based on the finite volume method to build the spatial discretisation of the transport equations. The flow equations are constructed face by face, which means that cells having an arbitrarily number of arbitrarily shaped faces can be accepted. As a result, it enables simulation of flows around complex geometries. Pressure-velocity coupling is obtained through a SIMPLE-like method. A Rhie & Chow reconstruction is used to form the pressure equation. In the case of turbulent flows, additional equations for modelled turbulent parameter are solved in a form similar to the momentum equation. An Arbitrary Lagrangian Eulerian (ALE) approach is used to deal with moving bodies and all DOFs, solved or imposed. Both rigid motion of the grid (translations and z_0-axis rotation) and analytical weighted deformation techniques (x_0-axis and y_0-axis rotations) are used to preserve a mesh fitted to the body during its motion. Where the grid is moving, the space conservation law must be satisfied. This is fulfilled using an appropriate computation of the grid displacement velocity flux through each face. We also benefit from the automatic grid refinement, described in more detail in Wackers et al. (2012), which enables in particular to refine the grid near the free surface when it is necessary.

3.2. Procedure for the oar flexibility

When the flexibility of the shaft is activated, the position of the blade is no longer fully imposed but depends on the fluid loads on the blade. Even if the position of the blade in the non-deformed configuration remains imposed, this FSI needs to be addressed with care. Since inertial effects of the oar are small compared to the fluid forces acting on the blade, the resolution of a static equilibrium of the structure is suitable. The linear and angular deflections are only considered in the X-Y plane, by taking into account the Z-axis moment M_z of the hydrodynamic load in the blade frame of reference (see Fig. 1(b)). In the other directions, loads are far weaker and deflection can be neglected. Then, the structural problem comes down to the equations $d_l = M_z / k_l$ and $d_a = M_z / k_a$, where d_l and d_a are respectively the linear and angular deviation with respect to the non-deformed configuration. k_l

and k_a denote respectively the linear and angular stiffness coefficient with respect to the Z-axis moment M_z evaluated at the gauge location. They are experimentally evaluated using the test bench described in Fig. 1(a). However, due to the added mass effect, a part of the fluid loads on the blade depends on the acceleration and may lead to divergence of the numerical coupling as shown by Söding (2001). To keep a stable fluid-structure coupling, the position of the blade is updated at each non-linear iteration through a quasi-Newton method using an accurate computation of the added mass. This procedure is explained with more details in Yvin et al. (2013).

4. Simulations, results and comparisons

From the experimental work described in section 2, a preliminary study has been launched to reproduce a real rowing stroke, including the FSI due to the flexible shaft. The only simplification is to keep the blade quasi-vertical as it can be seen on Fig. 2. The rotation around its own axis has been removed and only the stern pitch of the blade is imposed. That was done to avoid resorting to sliding grids for the moment and starting with a simpler configuration, knowing that this motion is not crucial for the propulsive phase. A moderate adaptive grid refinement around the free surface has been activated too. A coarse mesh contains initially 280,000 cells (before refinement around the free surface) to reach about 500,000 cells during the computation. The imposed movements come from two different experiments conducted by two different athletes (“B” and “F”) rowing at a rate of 18 strokes per minute.

The first results show the feasibility of such a configuration. Indeed, the evolution of the free surface is realistic (see Fig. 3). Besides, the Z-axis moment, which is the highest load during the drive and where the flexibility of the shaft acts, is quite well reproduced with respect to the experimental data (see Fig. 4). However, some discrepancies can be noted. The first one is an inflection appearing in the interval [40°, 70°] for the F001 case. This inflection appears later in the B002 case and clips the maximum value. The other disparity is the angular or temporal shift emerging after the maximum. The Y-axis bending moment (not represented here), a more minor component, follows nearly the same tendency but with larger differences in relative value. The origin of the differences is up to now not clearly identified.

Some differences can be explained by uncertainties or unmeasured data during the experiments. In particular, the roll angle was not measured. To try annihilating its effect, an averaged value (between port and starboard) has been taken for the immersion angle of the blade, considering the reasonable assumption that one blade is as immerged as the other. This procedure has enabled to get closer to the experimental results. In this test campaign, the boat pitch and heave were not measured, while they have an influence on the immersion of the blade. The setting of the oarlock angle is not known either, whereas it was shown on a series of computations that it has an effect on the stall observed. Moreover, the procedure to detect the contact of the blade with the water during the catch generates another uncertainty which affects the blade immersion: a small time difference of 0.05 s to detect the contact results in a gap of about 2 cm in the free surface location.

Numerically, dynamic stall and/or ventilation effects appearing during the stroke are possibly not correctly captured in the simulation. All the computations presented here were done with the Euler model because it was shown in Leroyer et al. (2010) that viscous effects have a weak influence. Other simulations with a turbulent model and with finer grids (up to 1.6 million cells) were tested but the results did not improved significantly. The meshes may be not fine enough to capture the whole physics of the flow, including more local phenomena.

(a)

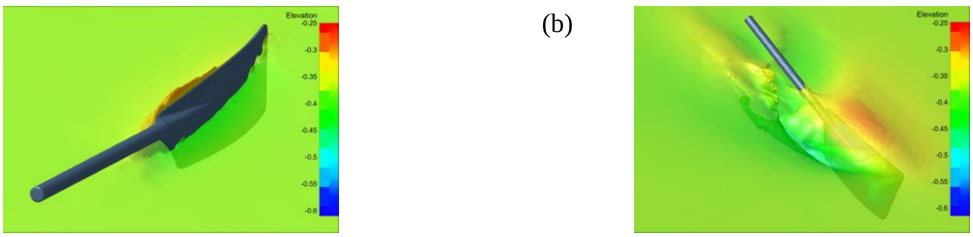

Fig. 3. Two views of the blade at (a) t=0.30s and (b) t=0.85s (F001 experiment). Display of the water elevation (in m).

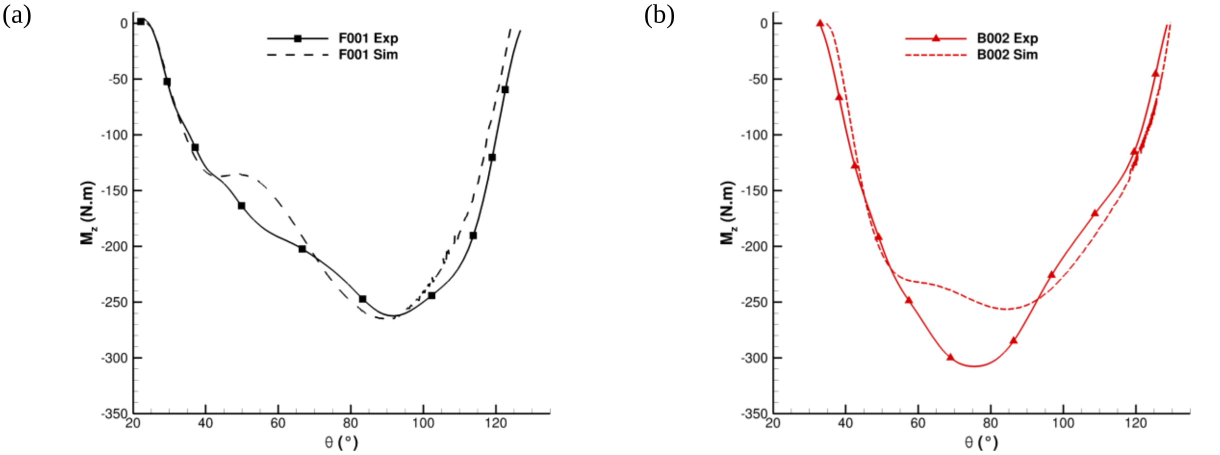

Fig. 4. M_z moment (N.m) at the gauges location during the propulsion for the cases (a) F001 and (b) B002, in function of the sweep angle *θ*, between the boat axis and the shaft.

5. Conclusion

This article aims at reporting progress about the approach, both experimental and numerical, pursued at LHEEA to gain knowledge about the flow around rowing blades. In particular, it shows the complementarity of these two approaches for this kind of research work. A study involving a real measured kinematic of a real oar has been started. It has reinforced the potential of the numerical methods which are used, especially the adaptive grid refinement to limit the diffusion of the interface, and the procedure to stabilize numerically the interaction between the flexible shaft and the fluid. However, experimental uncertainties, hard to estimate and resulting from unmeasured parameters (such as the roll, pitch and heave of the boat), make the validation task for this case difficult. Running computations on much finer grids will be one of the next steps of this validation process. The aim is to get the supposed physical behavior which is not captured with the coarser meshes used so far. The final goal is to better understand the physics of this complex flow and to improve the simplified models which are used in the simulator of the global system boat-oar-rower to predict performance.

Acknowledgements

This work was granted access to the HPC resources of CINES and IDRIS under the allocation 2B0129 made by GENCI (Grand Equipement National de Calcul Intensif). This study is supported by a scholarship of the “Région Pays de la Loire” (project ANOPACy).

References

Barré, S., Kobus, J.-M., 2010. Comparison between common models of forces on oar blades and forces measured by towing tank tests. Proceedings of the Institution of Mechanical Engineers, Part P: Journal of Sports Engineering and Technology 224, 37–50.

Leroyer, A., Barré, S., Kobus, J.-M., Visonneau, M., 2010. Influence of free surface, unsteadiness and viscous effects on oar blade hydrodynamic loads. Journal of Sports Sciences 28, 1287–1298.

Queutey, P., Visonneau, M., 2007. An interface capturing method for free-surface hydrodynamic flows. Computers & fluids 36, 1481–1510.

Rongère, F., Khalil, W., Kobus, J.-M., 2011. Dynamic modeling and simulation of rowing with a robotics formalism, in: Methods and Models in Automation and Robotics (MMAR), 2011 16th International Conference On. pp. 260–265.

Söding, H., 2001. How to integrate free motions of solids in fluids, in: 4th Numerical Towing Tank Symposium, Hamburg.

Wackers, J., Deng, G.B., Leroyer, A., Queutey, P., Visonneau, M., 2012. Adaptive grid refinement for hydrodynamic flows. Computers & Fluids 55, 85–100.

Yvin, C., Leroyer, A., Visonneau, M., 2013. Co-simulation in fluid-structure interaction with rigid bodies, in: Numerical Towing Tank Symposium 2013 (NuTTS’13).