The Computational Fluid Dynamics Study of Orientation Effects of Oar Blade

Alexandra Laurent,1 Annie Rouard,1 Vishveshwar R. Mantha,2,3 Daniel A. Marinho,3,4 Antonio J. Silva,2 and Abel I. Rouboa2,5,6

1University of Savoie; 2University of Trás-os-Montes-e-Alto Douro; 3Research Centre in Sports, Health and Human Development, Vila Real; 4University of Beira Interior; 5Research Centre and Technologies of Agro-Environment and Biological Sciences, Vila Real; 6University of Pennsylvania

The distribution of pressure coefficient formed when the fluid contacts with the kayak oar blade is not been studied extensively. The CFD technique was employed to calculate pressure coefficient distribution on the front and rear faces of oar blade resulting from the numerical resolution equations of the flow around the oar blade in the steady flow conditions (4 m/s) for three angular orientations of the oar (45°, 90°, 135°) with main flow. A three-dimensional (3D) geometric model of oar blade was modeled and the kappa-epsilon turbulence model was applied to compute the flow around the oar. The main results reported that, under steady state flow conditions, the drag coefficient \(C_d = 2.01\) for 4 m/s at 90° orientation has the similar evolution for the different oar blade orientation to the direction of the flow. This is valid when the orientation of the blade is perpendicular to the direction of the flow. Results indicated that the angle of oar strongly influenced the \(C_d\) with maximum values for 90° angle of the oar. Moreover, the distribution of the pressure is different for the internal and external edges depending upon oar angle. Finally, the difference of negative pressure coefficient \(C_p\) in the rear side and the positive \(C_p\) in the front side, contributes toward propulsive force. The results indicate that CFD can be considered an interesting new approach for pressure coefficient calculation on kayak oar blade. The CFD approach could be a useful tool to evaluate the effects of different blade designs on the oar forces and consequently on the boat propulsion contributing toward the design improvement in future oar models. The dependence of variation of pressure coefficient on the angular position of oar with respect to flow direction gives valuable dynamic information, which can be used during training for kayak competition.

Keywords: kayak oar blade, computational fluid dynamics, pressure coefficient, propulsion

Kayak rowing is a competitive sport, and the kayak speed is related to three components: (i) the strength and the coordination of the kayaker which determine the force transmission from the kayaker to the blade and to the boat (Aitken & Neal, 1992), (ii) the design of the boat which determines the drag (i.e., the force exerted by the water opposite to the boat displacement; Grare, 1985), (iii) the oar-blade system which transmits the force exerted by the kayaker (Mann & Kearney, 1980; Ackland, 2003). Different studies focused on the optimization of the race, concluding to the relationship between boat velocity and cycle parameters (frequency or length) related to kayaker’s power (Dolnik & Krasnopievtsen, 1981). A few studies were concerned with the kinematics of the blades for various paddling techniques (Plagenhoef, 1979; Issurin, 1980; Issurin et al., 1983; Kendal & Sanders, 1992). They concluded that the path of the blade must be parallel to the axis of the motion with a blade perpendicular to the boat during the propulsive phase. As the blade acts on the water, the blade did not remain fixed but presented slipping movement (Novakova, 1979; Plagenhoef, 1979; Mann & Kearney, 1980; Issurin, 1980; Issurin et al., 1983; Kendal & Sander, 1992; Di Puccio & Mattei, 2008). It was concluded that the flat blade allowed for increasing the boat velocity more quickly at the beginning of the race, whereas the curved blade would allow a gain of boat pace.
during the race (Wargnier, 1990). Moreover, Kendal & Sander (1992) observed that the velocity with a curved blade is related to greater forward attack and to a blade path close to the boat axis. Few studies concerned the force exerted by the blade, which appeared determinant of the boat velocity (Issurin, 1980; Issurin et al., 1983). A few studies are published on the hydrodynamic of the blade although the kayak factories developed different designs with variation of shape, curvature and width.

Kayak paddle can be compared with a swimmer’s hand, which has been investigated from hydrodynamic approach (Marinho et al., 2010). The kayaker’s force (or the swimmer’s force) is transmitted to the blade (or the swimmer’s hand), which propels the boat (or the body) through the water. The drag force is perpendicular to the direction of the movement of the hand or of the blade (Issurin, 1980; Issurin et al., 1983).

In swimming, based on kinematics or CFD, the drag force is strongly related to the orientation, the speed of the hand (Schleihaufl, 1974; Berger et al., 1997; Rouboa et al. 2006), the size of limbs (Rushall et al., 1994; Berger et al., 1995; Gardano & Dabichk, 2005; Marinho et al., 2008; Minetti et al., 2009), and the position of the fingers (Schleihaufl, 1974; Takagi et al., 2001; Minetti et al., 2009; Marinho et al., 2010). Moreover, the drag coefficient is constant for all flow speed and all sweep back angles (Bixler & Riewald, 2002; Silva et al., 2005; Rouboa et al., 2006; Alves et al., 2007; Marinho et al., 2010). More recently, using 2D-CFD analysis, Gourgoulis observed that the paddle increased the efficiency of the propulsion and the length of the stroke cycle (Gourgoulis et al., 2008).

In regard to (i) the new blade development, (ii) the lack of study on the hydrodynamic characteristics of the blades and (iii) the previous results obtained in swimming, the main goal of the current study is to evaluate the influence of the blade angle on the flow, in particular focusing on the local pressure coefficient using the CFD approach. We hypothesized that (i) the local pressure coefficient is not similar on all the surface of the paddle, with different values between the front and rear faces, (ii) the drag coefficient changes with the angle of the blade (45°, 90°, 135°), with a maximal value for the blade perpendicular to the boat. These propositions are examined by 3D CFD approach as previous studies underlined the greater precision with 3D approach as compared with 2D approach (Rouboa et al., 2006; Zaidi et al., 2008). Results indicated that irrespective of the oar angle, the three Cp lines of the frontal side of the oar presented the similar shape but presented minor variation in Cp values along the reference line. Greater Cp was observed on the front face of the blade than on the rear one, irrespective the orientation of the blade was (45°, 90° and 135°). The pathlines colored with average flow velocity starting from frontal face of oar blade show recirculation zone at rear face of oar blade and average velocity around the kayak oar with rear side view, the pressure and pressure coefficient on the front side of kayak oar are presented. The pressure and pressure coefficient have maximum values in the central region and gradually fall toward the both sides. The CFD approach could be a useful tool to evaluate the effects of different blade designs on the oar forces and consequently on the boat propulsion contributing toward the design improvement in future oar models. The dependence of variation of pressure coefficient on the angular position of oar with respect to flow direction gives valuable information, which can be used during training for kayak competition.

**Method**

The closure problem of the turbulent modeling was arrived at, by using k-ε model with appropriate wall functions (Bixler & Riewald, 2002; Rouboa et al., 2006) using ANSYS FLUENT commercial CFD software. The k-ε model, is extensively applied and validated in the varied industrial applications (Raiiesi et al., 2011). The k-ε model has been shown to be useful for free-shear layer flows with relatively small pressure gradients (Bardina et al., 1997). The present study is mainly focused on the variation of pressure coefficient due to angular orientation of oar blade with respect the flow direction excluding detailed flow characteristics (i.e., separation and reattachment of flow) including detailed turbulent effects. The system of equations for solving 3D, incompressible fluid flow in steady-state regimen is as follows:

\[
\frac{∂}{∂x_i}(\rho U_i) = 0 \tag{1}
\]

where \( i = 1, 2, 3 \).

Navier–Stokes (momentum) equations:

\[
\frac{∂}{∂x_j}(\rho U_i U_j) = -\frac{∂p}{∂x_i} + \frac{∂}{∂x_j}\left[\left(\frac{μ + μ_τ}{2}\right)\frac{∂U_i}{∂x_j}\right] + \frac{∂}{∂x_j}(\bar{U}_j U_i) - \frac{2}{3} \left(\bar{U}_i \frac{∂\bar{U}_i}{∂x_j}\right)
\]

(2)

where \( \bar{U}_i(t) = \bar{U}_i + \bar{u}_i \) is the component of instantaneous velocity in \( i \)-direction (m/s), \( \bar{U}_i \) is the component of time averaged mean velocity in \( i \)-direction (m/s), \( \bar{u}_i \) is the component of fluctuating velocity in \( i \)-direction (m/s), \( i, j \) are the direction vectors, \( ρ \) is average fluid density (kg/m³), \( μ \) is dynamic viscosity of fluid (kg/ms), \( μ_τ \) is turbulent viscosity of fluid (kg/ms), \( p \) is average pressure, \( k = \frac{1}{2}(\bar{u}_i \bar{u}_i) \) the turbulent kinetic energy per unit mass (m²/s²); \( ε \) is a dissipation rate of the turbulent kinetic energy per mass unit (m²/s³), \( δ_τ = 1 \) if \( i = j \) and \( δ_τ = 0 \) if \( i ≠ j \), the turbulent model constants \( σ_k, σ_ε, C_1, C_2 \) are shown in Table 1.

<table>
<thead>
<tr>
<th>( C_μ )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( σ_k )</th>
<th>( σ_ε )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

**Table 1** **Modified k-ε model constant**
The equations of turbulence model:

\[
\frac{\partial k}{\partial x_i} (\rho U_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu}{\sigma_e} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon \quad (3)
\]

\[
\frac{\partial \varepsilon}{\partial x_i} (\rho U_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu}{\sigma_e} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \frac{\varepsilon}{k} - C_2 \rho \varepsilon^2 \quad (4)
\]

The production of kinetic energy term is given as:

\[
P_k = -\rho \bar{u}_i \bar{u}_j \frac{\partial \bar{U}_j}{\partial x_i} = \mu_t S^2 \quad (5)
\]

where \( S \) the modulus of the mean rate-of-strain tensor is given by

\[
S \equiv \sqrt{2S_{ij}S_{ij}} \quad (6)
\]

where the mean strain rate \( S_{ij} \) is defined by

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (7)
\]

The turbulent viscosity is calculated by the following relation:

\[
\mu_t = \rho C_{\mu} k^2 \quad (8)
\]

To limit numerical dissipation, particularly when the geometry is complex inducing an unstructured grid, second-order discretization schemes are used (Figure 1). In generic terms, the convergence of the calculation is checked by the value of the residuals of the various flow parameters. The convergence criteria in ANSYS FLUENT was set at \( 10^{-6} \). This criterion is assumed sufficient to ensure the convergence of the solution for the current study. The boundary layer was created with aspect ratio algorithm with 4 rows of boundary layer grid cells with transition pattern of 1:1, with first row of grid at 0.01 mm immediately preceding wall grid with last percentage of 50% maintaining internal continuity. The first cell was 0.01 mm away from the blade producing mean y+ value was 1.09 and the tetrahedral grids had maximum skewness of 0.74 and overall average skewness of 0.38. Appropriate number of tetrahedral grids cells in simulation model was arrived, which was an outcome of grid independence test carried out at the beginning of actual simulations. It was found that the difference in solutions for the drag coefficients for subsequent refinement in tetrahedral grid were less than 1%, when tested at an angle of 90° to flow direction.

**Oar Blade Geometric Model**

For this first study on CFD applied on oar motion in water, the most common blade was chosen (Macon model), which presented symmetrical form in relation to the oar axis (Figure 2).

The blade used in the numeric simulation was first modeled using a CAD commercial Software (Solidworks Inc, Concord, USA) to compute the 3D geometrical model (Figure 3). The oar blade front face surface area is 0.12758742 m², length of blade is 0.72 m, width is 0.2 m.

---

**Figure 1** — The computational mesh model showing boundary condition along with orientation of oar blade for individual case under study.
Figure 2 — The blade reference model showing location of lines studied.

Figure 3 — Computational geometrical model from SOLIDWORKS software.
The 3D model is positioned in a rectangular geometrical domain and is meshed in GAMBIT (ANSYS FLUENT Inc, Sacramento, California) commercial meshing software. The boundary conditions were applied, the wall boundary condition of the oar surface, inlet and outlet surface at side surfaces and the symmetric boundary condition on the remaining exterior surfaces. Finally, the mesh model was exported to FLUENT software to simulate the flows around the oar.

The flow conditions were considered as following:

• for this first approach of CFD in kayak, the flow was considered as steady state with a horizontal velocity, \( U \) of 4 m/s, which corresponds to a Reynolds number (\( Re = UL/\nu \), with L denoting the blade’s length of 0.72 m and \( \nu \) the fluid kinematic viscosity of water). As an initial condition, the water was assumed to be at rest.

• The pressure was set equal to zero for all the sides of the computational domain.

• The fluid was considered incompressible with density of \( \rho = 996.6 \) kg m\(^{-3} \) and dynamic viscosity of water is \( 8.90 \times 10^{-4} \) Pa s at 25 °C.

• The action of the gravity force was set at \( g = 9.81 \) m s\(^{-2} \) with the assumption of turbulence intensity of 1% at the inlet.

The oar was positioned stationary in the central part of the domain, the water flowed perpendicular to the oar.

To simulate the different phases of the stroke, the flow was modeled for three different positions of the oar: 45°, which corresponded to the catch phase of the water; 90°, the major propulsive phase, which corresponds to the compressive phase (power phase); and 135°, which corresponds to the end of the aquatic phase (Begon, 2006).

The evaluation of the distribution of the local pressure on the oar was carried out along three imaginary lines along the length of oar blade, each on the front and rear sides of the oar blade. The location of these three lines were taken judiciously so as to reflect the average of pressure variation in nearby of three region i.e., internal, middle, external respectively (Figure 2) as per the location of the individual lines, which entailed the detailed analysis of pressure variation. This analysis can contribute in more individual lines, which entailed the detailed analysis of external respectively (Figure 2) as per the location of the variation in nearby of three region i.e., internal, middle, external.

The FLOWTRAN simulation results allowed to get the \( C_p \) for each x-coordinate of each line (internal, middle, external), for each side (front versus rear) and for each oar angle (45°, 90°, 135°). The study of variation of \( C_p \) along the respective lines was carried out as \( C_p \) was most important component contributing toward total drag coefficient and the propulsion obtained from the oar emerges to be essentially based on this drag component. The drag coefficient is obtained by the following Equations (9):

\[
C_D = \frac{F_{\text{drag}}}{\frac{1}{2} \rho V^2 \cdot A_{\text{projected}}} \tag{9}
\]

where \( F_{\text{drag}} \) is drag force (about 2143 N), \( C_D \) is drag coefficient, \( \rho \) the average fluid density, \( V \) is average fluid velocity, \( A_{\text{projected}} \) is the average projected cross-section area (about 0.133508 m\(^2 \) in the flow direction, all in S.I. units. The \( t \) test was applied to test for the difference in means for the different lines, the different sides, and the different angles (\( p < .05 \)) (Ockerman & Goldsman, 1999).

**Results**

Results indicated that irrespective of the oar angle, the three lines of the front side of the oar presented the similar shape of \( C_p \) curve along the x-axis (Figure 4). For the three different orientations (45°, 90°, and 135°) the \( C_p \) curve could be divided into three parts. The first part between 0 and 0.3x, which corresponded to the upper part of the blade (between the end of the handle and the beginning of the blade). For this first part, \( C_p \) is different for the three different lines with a decrease for the middle line, an increase and then a decrease for the external and internal lines of drag coefficient. A constant drag coefficient characterized the second portion of the curve between 0.3 at 0.5x for the three lines (the middle part of the blade). The third part of the curve located between 0.5–0.8x (the end of the blade) presented similar \( C_p \) for the three lines with an increase from 0.5 to 0.7x and decreases from 0.7 to 0.8x. Thus for all lines and all angles (45°, 90°, 135°), superior and unsteady \( C_p \) values were observed at the extremities of the blade, with stable lower \( C_p \) for the central part of the blade. These results clearly indicated a non-uniformity of \( C_p \) on the front face of the paddle especially at the extremities of the oar.

Although there is similarity of the \( C_p \) curves observed for the three oar angles, different values were observed for different angles of orientation with the main flow. The comparison of the stable portion showed greater \( C_p \) for an oar angle of 90° compared with 45° and 135° angles. To evaluate these differences, we compared the \( C_p \) of the three lines during the steady portion of the curve (0.3–0.5x) for the front side. Results of the \( t \) test indicated significant differences between the \( C_p \) at 45° and 90° (\( p = .00532 \)), 90° and 135° (\( p = .0012 \)) and not between 45° and 135° (\( p = .875 \)). In other words, the oar force is greater, when the oar is perpendicular to the boat with no significant differences among the three lines indicating an homogeneous distribution of \( C_p \) values, on the blade (Figure 5b). For the 45° blade angle, the \( C_p \) was higher for the external line of the blade than for the middle and interior ones (Figure 5a). Opposite results were observed for the 135° blade angle with higher \( C_p \) for the interior line than for the middle and exterior lines (Figure 5c).

The front side, presented minor variation of \( C_p \) values whatever the orientation of the oar, and whatever the line of the oar (Figure 6). \( C_p \) is present on the entire front side of the blade when the pressure appeared to vary only on few areas of the rear side with differences from one line to another. We can also observe that for both sides, the
pressure on areas appeared to vary similar whatever the orientation of the blade.

Greater $C_p$ was observed on the front face of the blade than on the rear one (Figure 7), whatever the orientation of the blade was ($45^\circ$, $90^\circ$ and $135^\circ$).

The pathlines plot colored with average flow velocity starting from frontal face of oar blade showing recirculation zone at rear face of oar blade and contour plots of average velocity around the kayak oar with rear side view, the pressure and pressure coefficient on the front side of kayak oar are presented in the Figure 8. The pressure and pressure coefficient have maximum values in the central region and gradually fall toward the both sides.

**Discussion**

For the first CFD study on kayak, the main goal was to evaluate the influence of blade angle on the flow. We hypothesized that (i) the pressure and the local pressure coefficient were not uniform regarding the surface of the
blade (internal, middle and external portions) and the side of the blade (front and rear), and (ii) the local pressure coefficient changed with the orientation of the blade with a maximum at 90° orientation angle of the blade.

Results pointed out that even if the fluid flow exerts different pressures on the blade, similar patterns of the local pressure coefficient on the x-axis were observed for the three blade angles (45°, 90°, 135°) and for the three lines (internal, middle, external) of the blade with higher values at the extremities and lower stable values for the central portion for the front side. As observed for the fin fish (Lauder et al., 2007; Zhu et al., 2008), the pressure is not uniform on all the surface of the blade reflecting different flow motions. The greater $C_p$ at the handle-blade junction is like the greater $C_p$ observed at the fin-body junction of the fish (Zhu et al., 2008). At this junction, water could not circulate around the handle-blade system (or body-fin for the fish) and, consequently, the pressure increased. The greater $C_p$ at the free extremity could be due to the smaller rounded surface at the end of the blade. A gradual increase of the local pressure coefficient was observed (Zhang et al., 2006) from the fin base to the fin tip similar to the observed $C_p$ increase from the top to the tip of the blade in present study. The stability and the lower $C_p$ in the central portion of the blade could be due to the greater area and the uniform design of this portion.

Despite the same $C_p$ patterns for the three lines and the three angles, results showed higher values for the blade angle of 90° and lower ones for 45° and 135°. The fluid did not exert the same force at 45°, 90° and 135°. These results were similar to those observed in swimming, with higher $C_p$ for an attack hand angle of 90° either quantified from a kinematic approach (Schleihau, 1974) or from CFD (Bixler & Riewald, 2002; Rouboa et al., 2006; Marinho et al., 2010). In addition, for 90° $C_p$ is not significantly different for all the portions of the blade (internal, middle and external). For this angle, the blade is perpendicular to the fluid flow, so the fluid hits this zone uniformly. In this position, the fluid cannot go around the blade, and consequently no vortex and no lift force can be created. In this case, the only propulsive force is the drag force applied uniformly on all the surface of the blade.

The nonsignificant differences between $C_p$ values for 45° and 135° indicated that the fluid globally exerted the
Figure 8 — For velocity of 4 m/s and 90 degree angle of oar with respect to the flow direction the (a) pathlines colored with average velocity starting from frontal face of oar blade, (b) contours of average velocity around the kayak oar blade, (c) contours of average pressure on kayak oar blade, (d) contours of average pressure coefficient of the kayak oar blade. (The online PDF version of this article contains this figure in color.)
same pressure at the beginning and at the end of the oar motion in the water. Despite the similarities of \(C_p\) values, the distribution of the \(C_p\) is not the same for \(45^\circ\) and \(135^\circ\), with superior \(C_p\) for the external edge compared with the internal one for \(45^\circ\), and converse by for \(135^\circ\). These differences between the internal and external edges could reflect differences in lift forces for these angles and/or differences in vortex productions. Similar results have been observed on the fish locomotion (Liao, 2007). For example, a counter-clockwise vortex is visible at the head when the fish-body presented an angle of \(100^\circ\) with the direction of the fluid. The vortex moved to the middle of the body when the fish was at \(180^\circ\), i.e., in the same direction as the fluid flow. It was concluded (Zhang et al., 2006) that the pressure is quite small at the leading edge area to reach its maximum at the trailing edge area. Thus, the repartition of \(C_p\) values changed with the orientation of the oar, as did the undulation of the fish.

Finally big differences in \(C_p\) values were observed between the rear and front sides of the oar. For the three blade angles, negative \(C_p\) was observed for the rear side and positive \(C_p\) for the front one. Moreover, the rear side presented some areas without any noticeable drag contributions. Similar results were observed for the swimmer, for which the boundary layer separation resulted in formation of a low-pressure region behind the body (Naemi et al., 2009) and also on the fish. It was noticed (Dubois et al., 1974) that the pressure was positive in front of the fish and became negative at the back portion. It was concluded (Hirata, 2001) that pressure is negative at the opposite side of the propulsive side when the fish is swimming. The negative pressure can be explained from the Bernoulli principle: the water accelerated to pass around the fish, resulting in a decrease of the pressure (Dubois et al., 1974). The pressure difference between the front and rear sides created a force oriented from the high to the low pressures areas, i.e., from the front to the rear sides contributing to propel the boat.

The present study represented the application of CFD approach of the flow around the oar in kayaking. Results indicated that the angle of the oar strongly influenced the \(C_p\) with maximum values for a \(90^\circ\) angle of the oar. Moreover, the distribution of the pressures is different for the internal and external edges depending of the oar angle. Finally, the difference of negative \(C_p\) in the rear side and the positive ones in the front side contributed to create a propulsive force. The CFD approach could be a useful tool to evaluate the effects of different blade designs on the oar forces and consequently on the boat propulsion contributing toward development of new oar design.

References


Lauder, G.V., Anderson, E.J., Tangorra, I., & Madden, P.G.A. (2007). Fish biorobotics: Kinematics and hydrodynamics...


