Fisheries Research 161 (2015) 400–407

Contents lists available at ScienceDirect

Fisheries Research

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

Design of trawl otter boards using computational fluid dynamics

Yuki Takahashi[a,*], Yasuzumi Fujimori[b], Fuxiang Hu[c], Xiaoli Shen[c], Nobuo Kimura[b]

[a] Graduate School of Fisheries Sciences, Hokkaido University, Hakodate, Hokkaido 041-8611, Japan
[b] Faculty of Fisheries Sciences, Hokkaido University, Hakodate, Hokkaido 041-8611, Japan
[c] Tokyo University of Marine Science and Technology, Minato, Tokyo 108-8477, Japan

ARTICLE INFO

Article history:
Received 19 November 2013
Received in revised form 7 August 2014
Accepted 11 August 2014
Handling Editor Dr. P. He
Available online 9 September 2014

Keywords:
CFD analysis
Otter board
Hydrodynamics characteristics
Flow visualization

ABSTRACT

Computational fluid dynamics (CFD) analysis was conducted for a biplane-type otter board model to validate the use of CFD analysis in designing otter boards. In the base mode (aspect ratio 3.0, camber ratio 15%, gap-chord ratio 0.91, and stagger angle 30°), CFD analysis results show a maximum lift coefficient value of 1.65 for angle of attack 17.5°, which is in agreement with results of the flume tank experiment (C_{Lmax} = 1.64 when α = 20°) under the same conditions. In addition, the streamline and separation zones around the otter board were visualized and compared to the results from past tank experiment studies. CFD analysis results showed patterns similar to the experimental results, and the details of flow with a trailing vortex were confirmed. We concluded that CFD analysis is useful in the designing and development of otter boards.

1. Introduction

The otter board is a vital component for the efficient operation of an otter trawl system. It requires high lift and low drag forces for net spreading, because of which the otter board's hydrodynamic characteristics are of critical importance. Some studies have investigated the hydrodynamic characteristics of otter boards and trawl systems by conducting flume tank experiments.

Crewe (1964) investigated the hydrodynamic characteristics of the flat-plate-type otter board from several aspects. Patterson and Watts (1986) tested low-aspect-ratio airfoils for otter boards using a model experiment. Park et al. (1993a,b, 1994a,b, 1996) studied the characteristics of flat-plate-type and cambered otter boards by measuring the hydrodynamic forces and conducting a visualization experiment. Recently, Fukuda et al. (1997, 1999) and Fukuda (1999) developed a biplane-type otter board, which has high lift force and stability and is widely used in Japanese fisheries. Sea-trials have also been carried out in the development processes of the otter board and the trawl system. Yamazaki et al. (2007) proposed a new design for the otter board and compared it with the conventional otter board through sea-trials. A mathematical model of the interaction between the otter board and trawl net was proposed and undertaken in a full-scale experiment by Prat et al. (2008). In addition, smart fisheries practices are currently required because of increasing fuel costs and decreasing fish populations. Sala et al. (2009) investigated the performance and impact of an existing and new design otter boards on the seabed through model tests and full-scale sea trials. Ivanovic et al. (2011) simulated the physical impact of trawl components on the seabed using the finite element method, and then compared those findings with results from sea trials.

Generally, model experiments and sea-trials have been conducted over the years to improve the hydrodynamic performance of the otter board. However, these methods are time-intensive and costly, furthermore, are difficult to execute when examining multiple combinations of design factors and how each combination impacts hydrodynamic performance. As a result, the design of the otter board has relied on empirical methods and experiences of designers.

Computational fluid dynamics (CFD) analysis—a numerical method for solving the equation of motions for a fluid—is widely used in the field of fluid dynamics. With advancements in computer technology, the precision of this method has been improved markedly. Moreover, its initial cost is lower than the cost of a tank experiment and does not require special facilities, such as a flume tank.

In ocean engineering, CFD analysis has been widely used for the development and investigation of ship design. For instance, Yelland et al. (1998) investigated wind stress on the upper structure of a ship by using CFD analysis. Ship hull forms were also optimized by Percival et al. (2001) with CFD analysis. Sadat-Hosseini et al. (2013)

* Corresponding author. Tel.: +81 138 40 8848.
E-mail addresses: yuuki87710@ec.hokudai.ac.jp, yuki.takahashi0710@gmail.com (Y. Takahashi).

http://dx.doi.org/10.1016/j.fishres.2014.08.011
0165-7836/

conducted CFD verification of the added resistance and motion of a bare hull. However, few CFD analytical studies have been conducted in fisheries engineering. The "net plane" (Patursson et al., 2010) and an aquaculture system (Rasmussen and McLean, 2004; Helsley and Kim, 2005; Fredriksson et al., 2008) have been analyzed by CFD. Concerning the otter board, the shape has been optimized with CFD analysis by Jonsson et al. (2013). However, the calculated otter board has been assumed two-dimensional. Furthermore their results have not been verified by flume tank tests or full-scale sea trials.

In this study, we conduct CFD analysis for the otter board to validate its suitability for otter board design. Lift and drag forces were estimated and compared with results from flume tank experiments. In addition, we visualized the streamline and the separation zones around the otter board, and compared these results with those from past visualization experiments. In this paper, we describe our methods and results obtained, and discusses suitability of the CFD method for designing otter boards.

2. Materials and method

2.1. Details of the otter board

An overview and the parameters of the biplane-type otter board used in this study (Fukuda et al., 1997, 1999; Fukuda, 1999) are shown in Fig. 1, and the corresponding parameter values are given in Table 1. The biplane-type otter board consists of two cambered plates (Fig. 1): the front plate is known as the "fore wing" and the back plate is called the "rear wing." We modified the gap-chord ratio (h/c) and the stagger angle (Θ) from those in the base model. Here, the gap-chord ratios are set at 0.75, 0.91, and 1.07 and the stagger angles at 20°, 30°, and 40°. The angle of attack of the flow (α) was set between 0° and 50°.

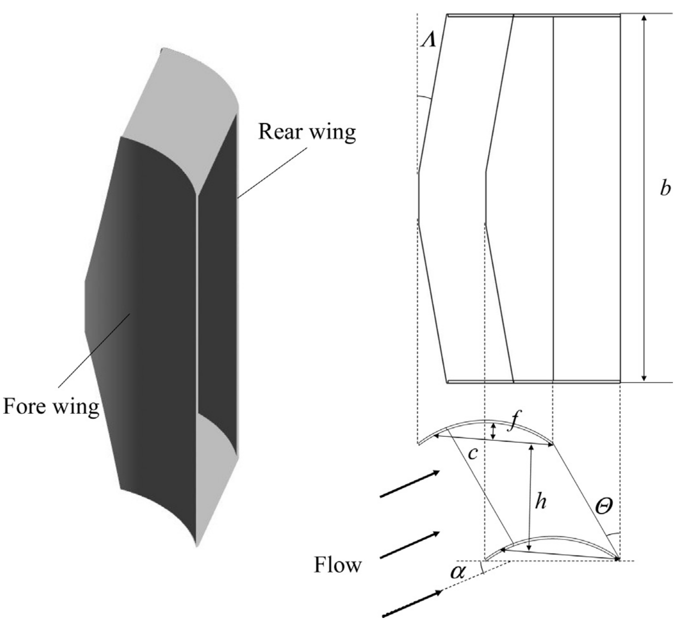

Fig. 1. Overview and design parameters of modeled biplane otter board, c: mean chord length, b: wing span, f: maximum camber of the mean chord, Λ: sweepback angle, h: gap-chord length, Θ: stagger angle.

2.2. CFD analysis

The numerical CFD code ANSYS Fluent 14.0 (ANSYS Inc, 2011) was adopted for CFD analysis. In the calculations, the finite-volume method was used to solve the Reynolds-averaged Navier–Stokes equations. Closure was provided using the realizable k–ε turbulence model, and Enhanced Wall Treatment was employed for the wall function.

The calculation domain of the analysis is shown in Fig. 2. Because the biplane-type otter board is symmetrical in shape, it was possible to use just half the model for calculations. For the base model, both the half model and the full model were calculated. Results show no difference between the two methods. The domain extended 0.5 m upstream and 1.0 m downstream, with a width of 0.5 m and a height of 0.5 m.

The computational grid around the otter board is shown in Fig. 3. On the computational grid, we selected a mixed-element mesh with prisms in the boundary layer and tetrahedrons in the far field. The maximum element size surface on the otter board is set as 2.0×10^{-3} m, and far field is 5.0×10^{-2} m. The prism mesh has an initial height of 4.0×10^{-5} m ($y^{+} \approx 1$), and extending 15 layers from surface. The number of elements totaled approximately 1.2×10^{6} with the node elements totaling approximately 4.5×10^{5}.

The study's boundary details are shown in Fig. 4. Water was modeled as fresh water with a density of 998.2 kg m^{-3} and viscosity of 0.001003 kg m^{-1} s^{-1}. The current velocity at the inlet boundary was set at a uniform flow of 1.0 ms^{-1} in the x direction, turbulence intensity was 1%, and turbulent viscosity ratio was 5. The pressure outlet condition used was downstream of the domain. The otter board walls were assumed to be smooth and with no-slip conditions ($u, v, w = 0.0$ ms^{-1}). The bottom of the computational domain was set to be symmetrical. Smooth walls with free-slip conditions were also assumed for the other walls. The simulation time needed for one calculation was around 50 min.

Following computation to a steady state, the lift and drag coefficients (C_L and C_D, respectively) were calculated using the following equations:

$$C_L = \frac{L}{0.5\rho S V^2} \quad (1)$$

$$C_D = \frac{D}{0.5\rho S V^2} \quad (2)$$

where L and D are the estimated lift and drag forces, respectively, ρ is the fluid density, S is the total area of the fore and rear wings, and V is the free-stream inflow velocity.

Table 1
Base model parameter of the biplane otter board.

Parameter	Base model	Modified models			
		Model 1	Model 2	Model 3	Model 4
Mean chord length: c (cm)	9.3	9.3	9.3	9.3	9.3
Aspect ratio: b/c	3.0	3.0	3.0	3.0	3.0
Camber ratio: f/c	0.15	0.15	0.15	0.15	0.15
Sweepback angle: Λ(°)	10	10	10	10	10
Gap-chord ratio: h/c	0.91	0.75	1.07	0.91	0.91
Stagger angle: Θ(°)	30	30	30	20	40