Conference or Workshop Item:

Alhinai, A. A. (2015) Estimating the Aerodynamic Performance of a High Speed Watercraft Using Computational Fluid Dynamics. In: USES 2015 - The University of Sheffield Engineering Symposium, 24 Jun 2015, The Octagon Centre, University of Sheffield.

10.15445/02012015.33

Estimating the Aerodynamic Performance of a High Speed Watercraft Using Computational Fluid Dynamics

A. A. Alhinai
Department of Mechanical Engineering, Faculty of Engineering, University of Sheffield.

Abstract

This paper presents a computational model to simulate the flow around a high speed watercraft operating at cruise conditions. The objective was to investigate the aerodynamic performance of the current design and asses the adequacy of Computational Fluid Dynamics (CFD) to conduct detailed design studies. The model developed was based on the physical conditions generally encountered in wind tunnel tests. The results show that the model is capable of capturing the general flow patterns with reasonable accuracy. However, accurate predictions of friction were not possible using this method. Nevertheless, the model proved to be useful in estimating the aerodynamic forces acting on the surface of the watercraft. In turn, specific design recommendations to improve the aerodynamic performance of the watercraft could be made using CFD.

Keywords Aerodynamics; CFD; COMSOL; Turbulence.

1. Introduction

In recent years Computational Fluid Dynamics (CFD) became a valuable tool in the design of watercraft. This technique has become possible through the rapid development of computers, which enable very detailed studies of the flow and resistance properties of the design to be made. Its advantage is that it is faster and cheaper than model-testing and the accuracy is continuously improving. For systematic variations and optimisation it has taken over much of the testing [1], while for very accurate predictions of the absolute values testing is still often considered necessary.

In the context of naval architecture there are two main types of CFD methods [1]:

- The simplest approach is based on potential flow theory where the viscosity is neglected. This enables calculations to be carried out rapidly without too much computational power. However, no quantities of friction can be obtained using this approach.
- In the second type, where viscosity is considered, some approximation of the fundamental equations of fluid mechanics is used.

In our analysis the viscous flow computations were carried out using the Reynolds Averaged Navier Stokes (RANS) equations. Larsson et al [1] investigated hull hydrodynamics using RANS, however, limited research was conducted to study the aerodynamics of high-speed watercraft. Therefore, the overall objective of this project was to develop a CFD model capable of predicting the aerodynamic forces acting on the watercraft with reasonable accuracy. The purpose of this research was to demonstrate to the watercraft manufacturer the potential savings that could be gained by replacing model-testing with CFD when considering systematic variation and optimisation studies to improve the aerodynamic performance.

2. Methodology

The simulations were carried out using the commercial software COMSOL Multiphysics. The turbulent flow interface with the K-Epsilon standard turbulence model was used to solve the RANS equations. The geometry and boundary conditions, shown in Figure 1, are similar to those found in wind tunnel testing. The value of inlet velocity was chosen based on the vehicles' maximum speed and parallel flow condition was prescribed at the outlet with no-slip condition prescribed for the remaining boundaries. Note the working fluid here is air and that the interaction between the water surface and the air is ignored here. The hydrodynamic resistance was calculated from field tests therefore our analysis focused on estimating the aerodynamic resistance. Following from the technical data of the watercraft and observations made from field tests, it was assumed that the watercraft was operating with maximum speed at cruise conditions with an angle of attack of 4 degrees.

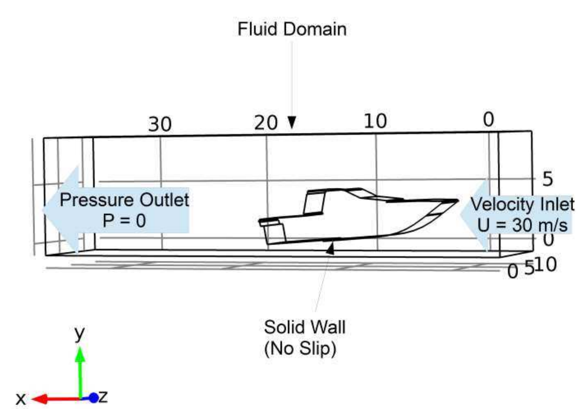

Figure 1. Schematic of the computational model showing the geometry and boundary conditions (dimensions in meters).

Due to lack of experimental data for the watercraft, it was necessary to run a test case with similar geometrical parameters to estimate the predictive capabilities of the computational model. To this end, we considered the flow over an Ahmed body which was extensively studied by Hucho [2]. The results in Table 1 show that our wind tunnel model is capable of predicting the drag coefficient (C_D) and pressure (C_p) coefficients, with reasonable accuracy.

Table 1. Comparison of the results from the test case for the Drag Coefficient and its components with experiment.

	C_p	C_f	C_D
Measurement [2]	0.23	0.055	0.285
Model	0.221	0.039	0.26
Error %	3.9	29	8.7

Prior to carrying out the full 3D simulation, a series of 2D studies were conducted in order to verify grid independence and to identify the appropriate turbulence model to be used. Following these studies, an unstructured grid (8284 elements in 2D and 47303 elements in 3D) was used to discretise the flow domain and the K-Epsilon standard turbulence model was selected to carry out the 3D simulations.

3. Results and Discussion

The comparison given in Table 1 demonstrates the models' limitations in predicting the friction coefficient. This was expected due to the limitations of the RANS equations and the turbulence model, mainly the use of the law of the wall to model the flow near the wall. This could be overcome by conducting a Large Eddy Simulation (LES) at the cost of higher computational effort. Despite the limitations of RANS and the K-Epsilon model, useful visualisation data of the flow field could be obtained for a quick assessment of new design ideas.

The flow field around the watercraft was visualised using the post-processing tools in COMSOL. Figure 2 shows the streamwise velocity field where high velocity regions are formed at the bow and the top of the cabin. Moving further downstream, a wake region develops on the deck behind the cabin and extends all the way to the stern. The results demonstrated that the simulation is capable of predicting the size of the wake region. This provides a useful design tool for sizing the cabin and the deck to minimize the drag in the wake region.

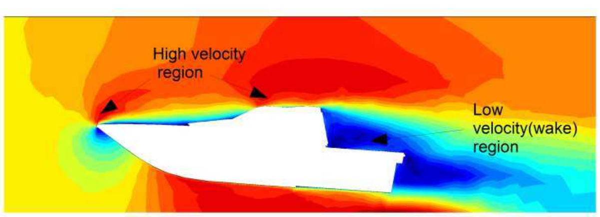

Figure 2. Streamwise velocity contour map showing the high and low velocity regions.

The pressure distribution over the watercraft surface was plotted to show the high and low pressure regions responsible for generating lift at the bow, as shown in Figure 3. Pressure surface plots were useful in identifying the areas where the design could be improved to enhance the overall aerodynamic performance of the watercraft.

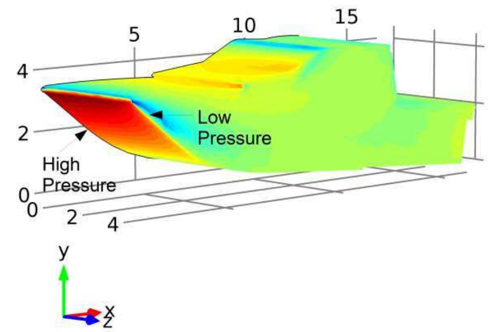

Figure 3. Pressure surface plot showing the high and low pressure zones over the bow (dimensions in meters).

4. Conclusions

A CFD model was developed to study the aerodynamics of a high speed watercraft. Comparison with experiments for the test case demonstrated that the model is capable of estimating the drag coefficient with reasonable accuracy. However, the results also show a major limitation in estimating the friction coefficient. Despite this limitation, useful flow visualisations could be obtained to assess new design concepts and their effects on the aerodynamic performance of the watercraft.

Acknowledgements

The author would like to thank United Engineering Services (UES Oman) for initiating this project and providing the CAD drawings.

References

1. Larsson L, Eliasson R, Orych M. Principles of Yacht Design.2013: p 326-335
2. Hucho W. Aerodynamics of Road Vehicles. 1998: p 106-211