Use of CFD to Study the Resistance of Sprint Master Canoe

Omar Yaakob[1,2], Yasser M. Ahmed[2,3], M. Rizal Othman[2]

[1]Marine Technology Center, Universiti Teknologi Malaysia, 81310, Skudai, Johor, Malaysia

[2]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, Skudai, Johor, Malaysia

[3]Faculty of Engineering, Alexandria University, Egypt

omar@fkm.utm.my, yasser@mail.fkm.utm.my

Keywords: Canoe; Sprint Master; CFD; resistance

Abstract. The resistance and the hydrodynamic characteristics of the Sprint Master canoe that is used in Universiti Teknologi Malaysia (UTM) have been studied at different speeds using Computational Fluid Dynamics (CFD) method. The resistance of the canoe at different speeds has been measured in UTM marine technology center (MTC). The finite volume code Ansys CFX has been used for carrying out the numerical simulations for the hull form of the canoe at the same experimental speeds. The Volume of Fluid method (VOF) has been used with Ansys CFX for capturing the free surface flow around the canoe hull at the same experimental speeds. The Shear Stress Transport (SST) turbulence model has been used in the RANSE code. The grid generator ICEM CFD has been used for building the grids for the RANSE code solver. The numerical results compare very well with the experimental results at the different speeds.

Introduction

There are many types of water sports in Malaysia such as swimming, jumping, scuba diving, yachting, canoeing and many more. Kayaks and canoes become an increasingly popular means of enjoying sport and leisure boating activities in Malaysia. Kayaks resemble canoes in that both are long, narrow, lightweight paddle boats which are pointed at both ends. Like canoes, kayaks have a hull, which is the hollow bottom shell of the boat. The main difference that separates the two is that canoes are propelled by single bladed paddle while kayaks are propelled using two bladed paddle. Generally, Kayak is a type of canoe and the word canoe can be used for describing both kayak and canoe [1]. There are two main types of canoes: C-boat (Canadian) and K-boat (Kayak) [2]. The first type is usually manufactured using many different materials, its length is around 5 m, and the paddler is usually kneeling while paddling the canoe. The other type, K-boat, is closed decked vessel, usually used by one person, and the paddler is usually sitting while paddling a kayak. Different materials are used for constructing canoes such as: wood, canvas, aluminum, fiberglass, Kevlar and Royalex [3].

The simulation of the free surface flow around a ship hull form based on Navier-Stokes Equations (NSE) is usually carried out by the interface-tracking method, e.g. a moving mesh method [4], and the interface-capturing method, e.g. the volume of fluid method (VOF) [5]. The mesh elements in the first method moves over an underlying fixed Eulerian grid for tracking a free surface flow pattern around ship hull. This mesh only covers the domain involving the water, where the free surface forms the upper boundary of the computational domain and is determined as part of the solution. This approach can be applied for moving boundary problems, but special treatments are required for simulating problems of large deformation such as breaking waves. In the interface-capturing method, both air and water are considered in the simulation and treated as two effective fluids. The numerical grids in this method are fixed in space, and the predication of the free surface location is achieved by solving an additional transport equation.

This study is complementary to previous research work [6], where the towing tank of the UTM marine lab was used for measuring the total resistance of the Sprint Master canoe at 3, 4, 5, 6 and 6.5 knots. The finite volume RANSE code Ansys CFX was used to simulate the viscous free surface flow around the canoe hull form of at the same speeds. However, the CFD results in the previous study were not agree well with the experimental results, therefore better mesh scheme and quality with more accurate numerical calculations have been conducted in this current study. The numerical results and the experimental results were compared very well.

Description of sprint master canoe

This type of canoe (Fig. 1) is usually used by UTM's canoeing club in all competitions; hence a special attention is given to this type of the canoe. The principal dimensions of the Sprint Master canoe are length = 4.4 m, breath = 0.54 m and draft 0.108 m.

Fig. 1 The hull form of Sprint Master canoe

Computational grid and boundary conditions

The computational domain extended as a function of ship length (*L*) for approximately *2.1L* in front of the canoe hull, *3.1L* behind the hull, *2L* to the side and *1.95L* under the keel of the canoe. The air layer extends *0.04L* above the still water surface. The grid generator of the RANSE code (ICEM CFD) was used for generating the unstructured tetrahedral grids that were required for Ansys CFX solver. Several computational grids were tested in this study to check the solution sensitivity, and the results of the final grid with 2,763,360 mesh elements (Fig. 2) are presented in this research paper.

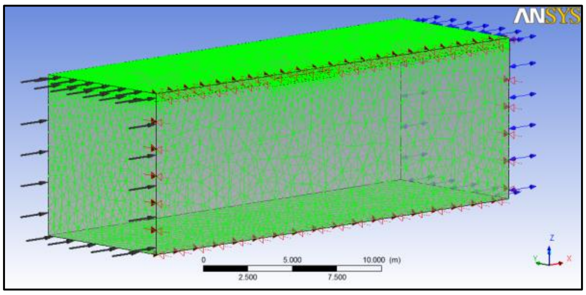

Fig. 2 Computational domain for the canoe meshed with 2,763,360 unstructured tetrahedral elements

The mesh was refined in the free surface region in order to get a sharp free surface. Furthermore, for predicting accurate values for R_P the mesh elements were refined on and near the ship hull surface in order to calculate accurate pressure forces acting on the hull form at the different speeds. The approximate value of y^+ in this study was 16.5. The no-slip boundary condition was imposed on the canoe hull. The static pressure at the outlet boundary was defined as a function of water volume fraction. In addition, the initial location of the free surface was imposed by defining the volume fraction functions of water and air at the inlet and outlet boundaries. Only the starboard side of the canoe hull form was considered with Ansys CFX due to the symmetric properties of the problem under consideration.

The flow was considered steady in Ansys CFX calculations, and the finite volume method is used for the discretization process. The SST turbulent model was used in this study. The high resolution numerical scheme was used for discretizing the advection terms. A linear interpolation scheme was used for interpolating the pressure, while the velocity was interpolated using a trilinear numerical scheme. The RMS criterion (root mean square) with a residual target value of 0.0001 was used for checking the convergence of the solutions.

Results and Comparisons

The numerical results of wave cut along the hull surface of the Sprint Master canoe predicted by the Ansys CFX at the different speeds are shown in Fig. 3. The comparisons between the results show that the height of the wave crest at the fore part of the canoe increased with the increase of the speed, which give a direct indication about the increase of the unit wave making resistance with increasing its speed. Furthermore, the wave trough along the canoe hull at approximately *-0.1L* to *0.4L* is increased with the increase of the speed, which will lead to reduce the pressure on the hull at the middle part. Finally, there are no appreciable changes in the wave height at the canoe stern for the different speeds.

The contours of the pressure distribution on Sprint Master canoe hull surface at the different speeds are shown in Fig. 4. The zones of high pressure at the bow and stern regions and the zone of low pressure at the middle part of the canoe hull form at the different speeds were detected well by Ansys CFX for the different speeds. The region of high pressure gradient at the canoe fore region zone augmented with the increase of the unit speed and the region of higher pressure at the aft part reduced with the increase of the speed. Furthermore, the value of high pressure surge with the speed, which will lead to the appearance of pronounced wave crest with this increase as shown in Fig. 3. The area of low pressure on the Sprint Master middle part enlarged with increasing the speed. The results of these variations in pressure distributions with speed will lead to a direct increase of pressure resistance with increasing the canoe speed.

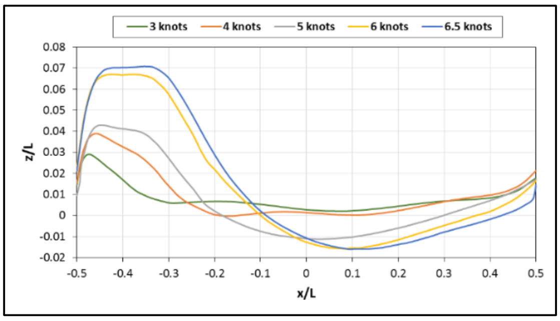

Fig. 3 Wave cut along the canoe hull at 3 knots

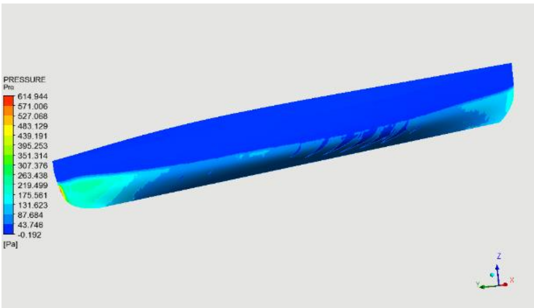

Fig. 4(a) Pressure contours on the canoe hull at 3 knots

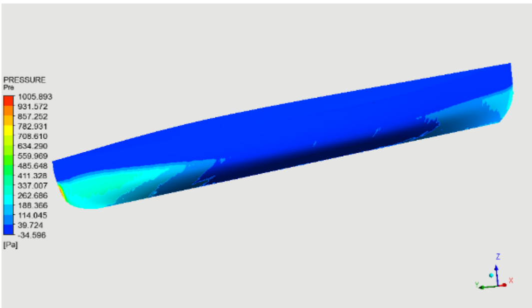

Fig. 4(b) Pressure contours on the canoe hull at 4 knots

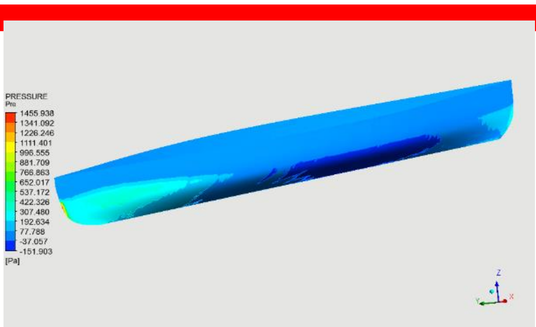

Fig. 4(c) Pressure contours on the canoe hull at 5 knots

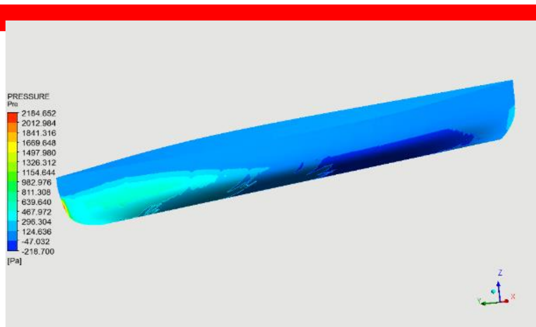

Fig. 4(d) Pressure contours on the canoe hull at 6 knots

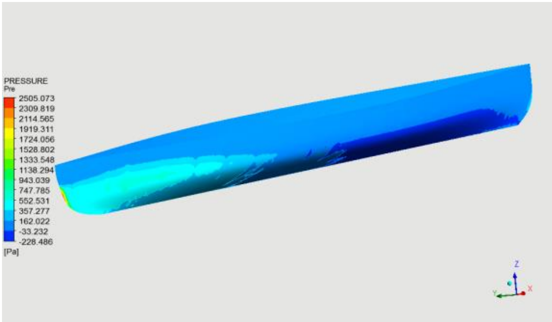

Fig. 4(e) Pressure contours on the canoe hull at 6 knots

The comparison between the predicted and the experimental values of the unit total resistance force (R_T), at different speeds can be seen in Fig. 5. In addition, Fig. 6 shows the comparison between the numerically predicted frictional resistance force (R_F) and that calculated by ITTC formula ($R_{F_{ITTC}} = 0.5\rho V^2 A C_F$). The value of C_F can be calculated from ITTC 57 friction line formula (Eq. (1)) [7].

$$C_F = \frac{0.075}{(\log_{10} R_e - 2)^2} \qquad (1)$$

The comparison between the calculated values of canoe pressure resistance (R_P) and the experimental values ($R_{P_{\exp}}$), (Eq. (2)) is presented in Fig. 7.

$$R_{P_{\exp}} = R_{T_{\exp}} - R_{F_{ITTC}} \qquad (2)$$

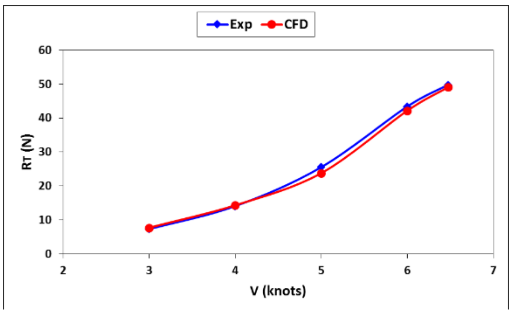

Fig. 5 Comparison between Ansys CFX results and the experimental results of R_T

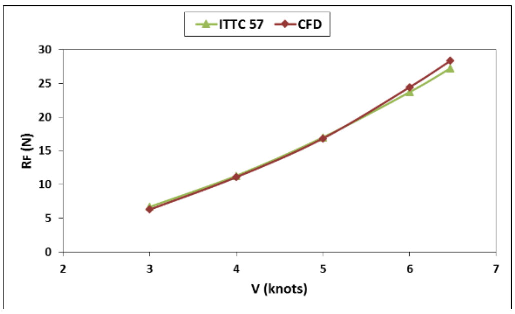

Fig. 6 Comparison between Ansys CFX results and Hughes results of R_F

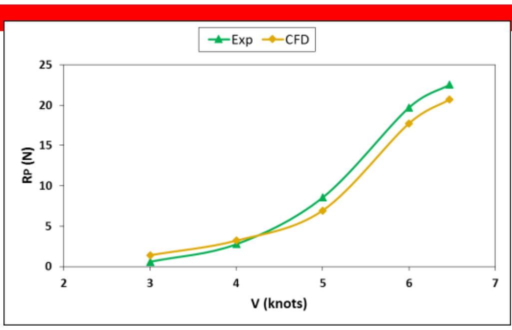

Fig. 7 Comparison between Ansys CFX results and the experimental results of R_P

The results in Fig. 5 show very good agreement between the experimental and CFD results at the various speeds. The comparison between the frictional and the pressure forces in Figs. 6 and 7 shows the higher ability of Ansys CFX on predicting very accurate results for both forces athigher and lower speeds.

Conclusion

The total resistance of the Sprint Master canoe has been measured in the UTM marine lab at different speeds. Furthermore, the turbulent free surface flow around the hull of the canoe at the same speeds has been simulated numerically using the finite volume RANSE code Ansys CFX to get more understanding for the hydrodynamic characteristics of the Sprint Master canoe. The wave profiles along the hull of the canoe have been detected at the different speeds, which show the potential effect of the hull form and speed on the produced wave pattern. Finally, the CFD results show higher ability for predicting very precise results for the Sprint Master canoe R_T and its components (R_F and R_P) at the different speeds, which can be used as a base for altering the unit hull form to improve its hydrodynamic characteristics.

References

[1] S. Rizzetta, Canoe and Kayak Building the Light and Easy Way: How to Build Tough, Super-Safe Boats in Kevlar, Carbon, or Fiberglass, Camden, ME: International Marine, 2009.

[2] M. Rizal Othman, "Hydrodynamic Design of One Person Canoe", B.Sc. Thesis, Universiti Teknologi Malaysia, Johor Bahru, Malaysia, 2007.

[3] Canoe types available at web site (accessed in 17 August 2014): http://canoe.ca/AllAboutCanoes/canoe_types.html

[4] T. Li, and J. Matusiak, "Simulation of modern surface ships with a wetted transom in a viscous flow", Proceedings 11th International Offshore and Polar Engineering Conference, vol. 4, pp. 570–576, 2001.

[5] Y. Ahmed, M. A. Fonfach, and C. G. Soares, "Numerical Simulation for the Flow Around the Hull of the DTMB 5415 at Different Speeds", International Review of Mechanical Engineering Journal, vol. 4, no. 7, pp. 957-964, 2010.

[6] Omar Yaakob, Y. Ahmed, M. Rizal Othman, "Studying the Hydrodynamic Characteristics of Sprint Master canoe", International Conference on Applications and Design in Mechanical Engineering 2012 (ICADME), Malaysia, 2012.

[7] A. F. Molland, S. R. Turnock, and D. A. Hudson, Ship Resistance and Propulsion: Practical Estimation of Ship Propulsive Power, Cambridge: Cambridge University Press, 2011.