ANALYSIS OF HYDRODYNAMIC PERFORMANCE OF A SHIP WITH PROPELLER USING OPENFOAM

ZHAI SHUCHENG[1], ZHENG CHAOSHENG[2], LIU DENGCHENG[3]
[1] *China Ship Scientific and Research Center, National Key Laboratory of Ship Vibration and Noise, Jiangsu Key Laboratory of Green Ship Technology, zsc_cssrc@163.com*
[2] *China Ship Scientific and Research Center, National Key Laboratory of Ship Vibration and Noise, Jiangsu Key Laboratory of Green Ship Technology, zsczsc2005@163.com*
[3] *China Ship Scientific and Research Center, National Key Laboratory of Ship Vibration and Noise, Jiangsu Key Laboratory of Green Ship Technology, edon_00@163.com*

Abstract
This paper analyses the resistance performance of an oil tank ship and the hydrodynamic characters of propeller behind this ship. The study was completed using an open source computational program, Open Field Operation and Manipulation (OpenFOAM), and the resistance character and propeller hydrodynamics performance were compared with experimental results. Reasonable results were produced such that the resistance, thrust, torque and efficiency trends were in good agreement with experimental data. It was found that OpenFOAM has good prediction of ship hydrodynamic performance.

Keywords: *hydrodynamic performance, propeller, OpenFOAM.*

This template provides a definition of the Microsoft Word styles for extended abstract submissions to the 13th OpenFOAM Workshop.

0 Introduction

The computational fluid dynamic (CFD) is more and more popular in ship and propeller design process. OpenFOAM is an established object-oriented library for Computational Continuum Mechanics, such as CFD [1], [2]. Many investigations have been done in ship and ocean engineering by this advanced tool package. Kim et al. [3] report a series of simulations on prediction of cavitation and performance of marine propellers. Simulations of propellers in open water utilizes the geometry of a single rotating frame of reference. Beaudoin et al [4] report a steady-state simulation of a Francis turbine, using the MFR model and two mixing plane interfaces. Zheng et al [5] gives the open water predictions of DTMB4119 propeller, which agree well with test data. Cha Jingjing et al[6] use interDyMFoam package with piston and flap type numerical wave makers to generate linear wave and finite amplitude waves. A damping zone function is added to the solver to absorb wave and be testified by several experiments. Li Yulong et al[7] simulated the ship motions coupled with tank sloshing in time domain based on OpenFOAM, and the simulation and experimental studied indicate that the numerical results of ship motion coupled with tank sloshing can clearly show the coupling effect of tank sloshing on the ship global motion. LV Lianli et al[8] established a numerical wave tanks to study the wave pressure field impacted on the ship. ZHOU Hu et al[9] investigated the difference between aerodynamics of two blades and three blades wind turbine under different wind conditions with different turbulence model and unstructured grid influences.
The aim for this paper is to successfully model a ship with propeller. The ship resistance and propeller thrust and torque are simulated. It will allow for good comparison between experimental results. Moreover, further analysis can be conducted in how the force is affected on ship and propeller in different conditions.

1 Model

1.1 CFD Methods
The incompressible Reynolds-Averaged Navier-Stocks equations (RANS) method is used to simulate the hydrodynamic performance of ship in this paper. See eq. (1) (2). With the RANS equations there is still a closure problem. This is solved using a turbulence model. The k-ω SST model is applied, which is using blending functions to be able to use the k-ω model near the wall and the k-ε in the free stream and to get a smooth transition between them [10]. This model is useful in ship and propeller simulation. The second order upwind scheme is used to discretize the equations and the pressure velocity coupling is solved by SIMPLE method. The Multiple Reference Frames (MRF) is used to simulate the propeller rotating. OpenFOAM version 2.3.0 is used in this paper.

$$\frac{}{x_i}\left(\overline{U_\iota}\overline{U_J} = -\frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_i} + \mu\frac{\partial^2 U_i}{x_j \ x_j} - \frac{\partial \overline{U_\iota U_J}}{x_j}\right) \quad (1)$$

$$\frac{\partial \overline{U_j}}{\partial x_j} = 0 \tag{2}$$

1.2 Ship Model

Oil tanker is one of the four main ship types in the shipping market. The present research model is an oil tanker with four blades propeller and rudder behind and its design speed is 14.5 kn. Main parameters of ship and propeller are shown in table 1.

Table 1 Main parameters of ship and propeller

Name		Symbol	Vale	Unit
Ship	Length of waterline	L_w	179	m
	Design draught	D_f	11	m
Propeller	Propeller diameter	D_p	7.29	m
	Pitch ratio	$(P/D_{PS})_{0.75R}$	0.835	-
	Blade area tatio	A_E/A_0	0.4	-
	Number of blades	Z	4	-
	Direction of rotation	-	Right	-
Scale		-	1:29	-

Fig 1 Oil tanker model with propeller

1.3 Mesh

The calculating domain is a rectangle with $7L_w \times 2L_w \times L_w$, The ship model position is L_w to inlet, L_w to side and $5 \times L_w$ to outlet. The boundary conditions are shown in figure 2. We use overlap model to simulate without consider the free water surface, because the Froude number of oil tanker is small, the wave resistance has little effect of propeller hydrodynamic performance, but it should be careful that some error will be put in the ship resistance. The mesh is generated by HEXPRESS software with unstructured hexahedron cell. In order to improve the spatial resolution in stern region using mesh density core, as shown in figure 3. Total mesh of resistance simulation case is about 2 million and 4 million for ship with propeller case. The personal computer with i7-3400@3.4Ghz CPU is usedto calculate and the time consume is about 10 CPU hours.

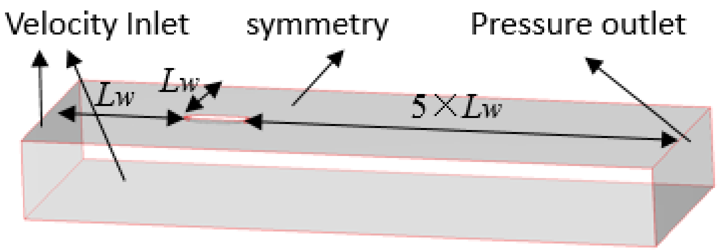

Fig 2 Calculating domain

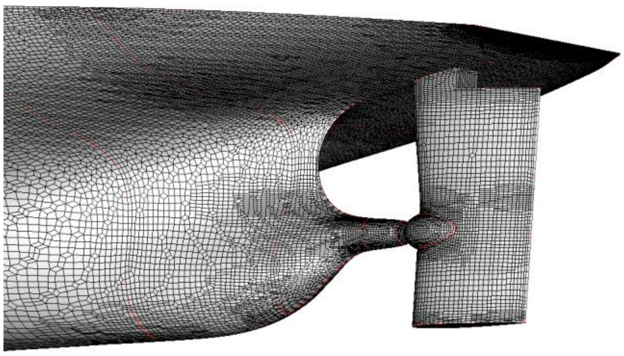

Fig 3 Mesh of the stern

2 Results and Analysis

Firstly, we calculate the ship resistance without propeller, and compare results with test data. C_{tm} is the resistance coefficient of bare hull defined as $C_{tm} = R/0.5\rho v^2 S$, where R is the resistance of ship, $\rho$is the water density, v is the upstream velocity, S is wet surface area. Here the commercial software Fluent simulate results are also shown in table 2. The simulation values are larger than experimental results. The error of OpenFOAM predictions is about 5.1%, a little larger than Fluent.

Table 2 Resistance of ship without propeller

V_s(kn)	F_n	Method	$C_{tm}(10^{-3})$	Δ (%)
10.5	0.129	Experiment	3.913	-
		Fluent	4.024	2.8%
		OpenFoam	3.645	-6.9%
14.5	0.178	Experiment	3.786	-
		Fluent	3.933	3.8%
		OpenFoam	3.980	5.1%

The ship self-propelled condition is a complex system, because the propeller is working in ship's wake flow field. The vortex structure generated by ship hull will flow into the propeller plane and affect the propeller hydrodynamic. In condition, the good simulation of rotating parts is also a difficult point. In this paper, the propeller rotating is simulated with MRF method, which is a quasi-steady method including centrifugal and Coriolis force in equation. The data transfer between moving region and static region is by using AMI type interface tool.

The propeller thrust and torque are calculated, as shown in Table 3. The OpenFOAM results are little better than Fluent results but both are higher than experimental results. The pressure coefficient C_p contours of the hull and propeller are shown in Figure 4 and Figure 5. The hull contour characters of pressure calculated by two softwares are almost the same. It should be noted that there are some differences at the pressure side and suction side of the propeller, which cause the thrust and torque with some differences. The oil tanker has large block coefficient and the stern geometry is plump, the water flows around these surfaces may separate when the upstream water velocity is specified. This complex separation flow is hard to simulate accurately, and leads to some simulation error.

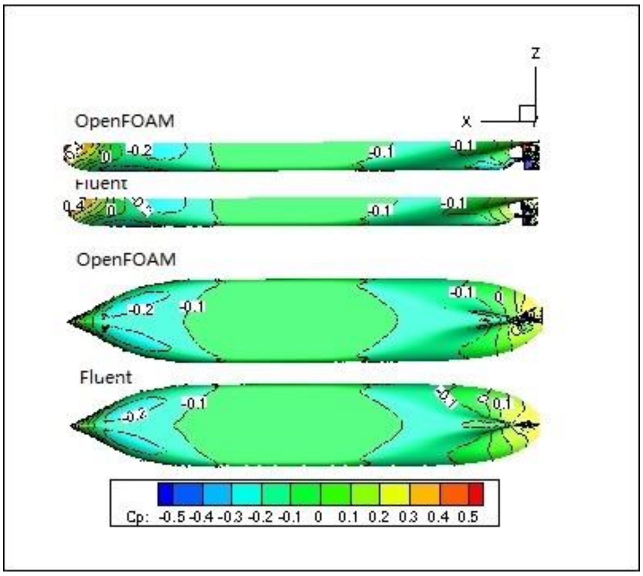

Fig 4 Pressure coefficient on ship hull

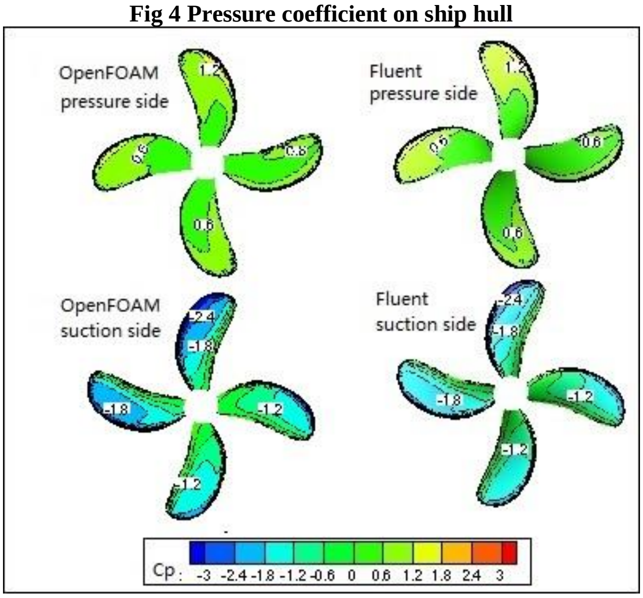

Fig 5 Pressure coefficient on propeller

Table 3 Hydrodynamics of propeller behind ship

Vm(m/s)	N_m(rps)	Method	T_m(N)	Δ (%)	Q_m(N.m)	Δ (%)
1.385	6.664	Experiment	27.005	-	0.906	-
		Fluent	29.204	8.1%	0.989	9.2%
		OpenFOAM	27.449	1.6%	0.914	8.4%

3 Conclusions

The hydrodynamic performances of hull and propeller behind it are simulated by OpenFOAM in this paper. The hydrodynamic of ship model and propeller performance are calculated successfully. The errors between simulation and test data are acceptable. In further the turbulence model and some solution methods will be investigated to improve the simulation accuracy.

Acknowledgements

The authors thank the Jiangsu Key Laboratory of Green Ship Technology to provide test data. The colleagues of CFD group of National Key Laboratory of Ship Vibration and Noise are acknowledged for open source code study.

Reference

[1] Joris Degroote, Patrick Segers and Jan Vierendeels . CFD Challenge: Solutions Using an Open-Source Finite Volume Solver, OpenFOAM. ASME 2012 Summer Bioengineering Conference. pp. 139-140. 2012.

[2] P. Berg, G. Janiga and D. Thévenin. CFD Challenge: Solutions Using the Commercial Solver Fluent and the Open-Source Solver OpenFOAM. ASME 2012 Summer Bioengineering Conference. pp. 115-116. 2012.

[3] Kim, S.-E., Scroeder, S., and Jasak, H.: "A Multi-Phase CFD Framework for Predict-ing Performance of Marine Propulsors", InThe Thirteenth International Symposium on Transport Phenomena and Dynamics of Ro-tating Machinery (ISROMAC-13), Honolulu,Hawaii, April 2010.

[4] Page, M., Beaudoin, M., and Giroux, A.-M.: "Steady-state Capabilities for Hydroturbines with OpenFOAM", Int. J. of Fluid Machinery and Systems, 4(1):160–170, Jan-Mar 2011.

[5] ZHENG Chaosheng, ZHANG ZHirong, Prediction Method for Open-Water Perfoamance of Propeller Based on OpenFOAM.Chanese Journal of Ship Research, Vol.7 No.3, Jun.2012.

[6] CHA Jingjing, WAN Decheng. Numerical wave generation and absorption based on OpenFOAM. The Ocean Engineering, Vol. 29 No.3 Aug.2011, pp.1-12.

[7] LI Yulong, ZHU Renchuan, MIAO Guoping, FAN Jun, LU Zhimei. Simulation of ship motions coupled with tank closhing in time domain based on OpenFOAM. Journal of Ship Mechanics, Vol. 16 No.7 Jul. 2012. pp : 751-762.

[8] LV lianli, SUN Hequan, WANG Jiguang, HUANG Pengfei. Ocean Wave Impact on Ship Hydrodynamic Pressure Field Based on OpenFOAM. Ship Electronic Engineering. Vol.34, No.10, pp: 135-147.

[9] ZHOU Hu, WAN Decheng. Numerical simulation of the unsteady flow around wind turbines with different blades numbers. Chinese Journal of Hydrodynamics. Vol. 29, No.4, July, 2012, pp: 445-452.

[10] Davidsson L., an Introduction to Turbulence Models. Göteborg, Sweden, Chalmers University of Technology, Department of Thermo and Fluid Dynamics. November 2003.