FLOW MODELLING AROUND PROPELLER FOR A DEEP DRAFTED VESSEL IN VERY SHALLOW WATER

SYED MOHAMAD NAJMI BIN SYED TALIB

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Engineering (Marine Technology)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

JANUARY 2014

To my beloved father and mother

ACKNOWLEDGEMENT

First, I would like to express my heartiest gratitude to Allah s.w.t for his bless in the completion of this thesis. No such thing is done without permission from Allah. Million thanks too dedicated to my parent for their huge support and keeping me updates with the study until completion and encouraging me both physically and spiritually.

Not to forget very much appreciation to a couple of individuals who extensively assisted me in completing this project, especially to my main supervisor, Dr. Agoes Priyanto for giving me loads of useful information, advice, comment and guidance through completing this thesis. Appreciation also dedicated to my co-supervisor, Dr Yasser, who sacrifices his precious time in guiding me towards mastering the field of Computational Fluid Dynamics. Finally, my second co-supervisor, Prof. Dr. Adi Maimun who fed me his piece of mind as well as expert recommendation and suggestion towards an accurate method in craving my thesis.

Thanks also to the Ministry of higher Education (MOHE), for providing financial support for this research, coded by R.J130000.7824.4F049. A condusive place for me to conduct this research, which is the Marine Technology Center (MTC), at Universiti Teknologi Malaysia.

Finally, a lot of thanks to all my fellow friends, those who have, in one way or another, assisting me weather direct or indirect towards the completion of this thesis.

ABSTRAK

Kod bendalir dinamik berkomputer (CFD) semakin mendapat sambutan kerana ianya merupakan satu medium yang efektif bagi memahami ciri-ciri aliran air, antaranya perolakan air di sekeliling kipas kapal. Justeru, tesis ini mempersembahkan model berangka bagi ciri-ciri aliran air pada bahagian buritan kapal LNG membabitkan kesan daripada kipas kapal dan kedalaman air yang cetek. Simulasi dibuat berpandukan model kipas kapal jenis B5-75 bergaris pusat (*D*) 7.7m yang telah direka dan diuji di MARIN, Netherland. Perisian ANSYS Fluent versi ke 12 digunakan bagi menyelesaikan persamaan RANS, manakala ICEM CFD digunakan untuk menjana grid isipadu serta permukaan yang direka. Grid yang dijana pada kipas kapal adalah jenis grid struktur tetra berselerak pada kawasan aliran air berdasarkan permukaan 3D tidak termampat persamaan Navier-Stokes. Dua jenis model perolakan aliran digunakan dalam perisian ANSYS Fluent, iaitu model biasa k-epsilon (k-ε) untuk simulasi malar manakala pembawa daya rincih (SST) k-omega (k-ω) bagi simulasi tidak malar. Bagi perincian ruang simulasi, kipas kapal ditempatkan dalam dua silinder; silinder luar dan dalam yang masing-masing berdiameter sekata. Dua jenis ruang simulasi digunakan iaitu ruang statik (stator) dan ruang dinamik (rotor). Bagi ruang stator, jarak dari tempat air masuk ke bilah kapal adalah 2*D*, manakala jarak dari bilah ke air keluar adalah 6*D*. Diameter keliling adalah 3.6*D*. Bagi ruang rotor, jarak dari air masuk ke bilah adalah 0.2*D* manakala jarak aliran keluar berada dalam julat antara 0.4 dan 0.7*D*, serta diameter keliling sebanyak 1.4*D*. Simulasi air bergolak mengambil kira kedua-dua pendekatan rotor-stator, iaitu rujukan posisi pelbagai (MRF) serta kaedah grid gelincir (SD). Bandingan dilakukan melalui eksperimen dari jurnal-jurnal yang telah diterbitkan, serta kajian terperinci berkenaan kaedah kebergantungan terhadap simulasi berangka dan parameter berkomputer telah dilaksanakan. Prestasi kipas kapal bagi kes simulasi umumnya diramal dengan perbezaan kecil berbanding ekperimen di air lepas, kira-kira 10%, mungkin disebabkan strategi penjanaan grid , resolusi grid serta kualiti grid. Simulasi jaga dibuat terhadap kehadiran kemudi kapal yang diletakkan selepas kipas kapal dimana ianya menyebabkan kecekapan kipas kapal meningkat dan terus-menerus meningkat apabila kemudi diputar pada sudut -7^0 and -20^0. Kemudi bertindak menghapuskan pusaran air yang terhasil dari kipas kapal yang secara tidak langsung meningkatkan tujahan serta torknya. Seperti yang dijangka, berlaku prebezaan dari segi pengamatan halaju antara simulasi kipas kapal di air lepas dengan interaksi antara badan kapal dan kipas kapal. Kesan daripada kipas kapal dan kemudi terhadap butiran kelajuan air di sekitar buritan kapal LNG telah dikenalpasti dengan jelas. Dengan keutamaan pada kedalaman air paling cetek (h/T = 1.1), butiran halaju ekstrem tertumpu pada bahagian buritan kapal yang ditenggelami air serta bahagian dasar laut.

ABSTRACT

Computational fluid dynamics (CFD) codes, are recently used as efficient tools to understand flow characteristics such as wake development around propeller. This thesis presents numerical modelling of flow characteristics in the stern region for a deep drafted LNG carrier with the effect of propeller and rudder in shallow water. The modelling was conducted based on the B5-75 type propeller, with a diameter (*D*) of 7.7m, which was designed at MARIN in the Netherlands. The ANSYS Fluent version 12 software was used to solve the Reynold Averaged Navier Stokes (RANS) equations, and ICEM CFD as a mesh generator. The propeller was meshed using tetra unstructured mesh in a flow field based on 3D incompressible Navier-stokes solver. Two turbulent models were applied in the ANSYS Fluent; which are the standard k-epsilon (k-ε) model for the steady simulation and transient shear stress transport (SST) k-omega (k-ω) for the unsteady simulation. For the computational domain, the propeller blades were mounted on two finite long constant radius cylinders. The two types of cylinder domains, were developed; stator domain and rotor domain. For the stator domain, the inlet flow was 2*D* from blade, the outlet flow at 6*D* and the outer boundary was 3.6*D*. The upstream for the rotor domain was maintained at 0.2*D* but the downstream was extended between 0.4*D* and 0.7*D*, and the outer boundary at 1.4*D*. The turbulent model was simulated in the rotor domain by using the stator-rotor approaches such as the multiple reference frame (MRF) and the sliding mesh (SM) method. Comparisons with the published experiments were presented, and the dependence of the numerical solutions on the computational parameters was studied extensively. The thrust and torque of the propeller were generally predicted with a small error when it was compared with the published experiments. The difference in performance of propeller in the open water test is about 10 percent, likely due to mesh strategy as well as mesh resolution and quality. The performance of the propeller was also studied. It was found that the rudder placed in front of propeller increased the efficiency of the propeller and produced greater thrust increments when the rudder was deflected to -7^0 and -20^0. The presence of the rudder which acts by cancelling the trailing vortices from the tip of propeller slipstream leads to increase of thrust and torque of propeller. There was, as expected, a difference in the velocity concentration between propeller only and propeller-hull interaction. The effects of propeller and rudder on the velocity profiles in the region for the LNG carrier in shallow water are clearly identified. Especially in very shallow water, (h/T = 1.1), the extreme velocity profile is concentrated in vicinity of top part of the stern and seabed regions.