Scientific Journals
Maritime University of Szczecin

Zeszyty Naukowe
Akademia Morska w Szczecinie

2010, 22(94) pp. 83–87

2010, 22(94) s. 83–87

Numerical analysis of propeller and rudder system working in off-design condition on LNG carrier

Numeryczna analiza opływu układu ster -pędnik pracującego w warunkach pozaprojektowych na gazowcu LNG

Jakub Handke, Tomasz Abramowski

West Pomeranian University of Technology, Faculty of Maritime Technology
Zachodniopomorski Uniwersytet Technologiczny, Wydział Techniki Morskiej
71-065 Szczecin, al. Piastów 41, e-mail: tomasz.abramowski@zut.edu.pl

Key words: cfd analysis, unsteady solution, screw propeller

Abstract

The paper presents results of unsteady numerical analysis of propeller-rudder system. For the calculations the Ansys Fluent solver was employed. The governing equations are RANSE. The results of computed forces, pathlines and pressure distribution on propeller and rudder are given as a function of rudder deflection angle. Initial investigations point to a number of very interesting effects, e.g. the formation on the rudder for small angle of deflection, the resulting force coincident with the direction of ship speed moving ahead. For that reason the resulting force may be considered as an additional component of the thrust force. Presented analysis may be employed during design of ships which are expected to attain good manoeuvring performance (e.g. LNG carriers) or ships which are designed with diesel-electric propulsion installation where there is a possibility to recover energy from inertia when slowing down.

Słowa kluczowe: analiza cfd, rozwiązanie niestacjonarne, pędnik śrubowy

Abstrakt

W artykule przedstawiono numeryczną analizę układu śruba-ster w sformułowaniu niestacjonarnym. Do obliczeń zastosowano system Ansys Fluent. Modelowanie przepływu odbywało się przy pomocy równań RANS. Zaprezentowano wyniki obliczeń sił, linii prądu oraz ciśnień na sterze i śrubie. Siły występujące w całym układzie śruba-ster zaprezentowano w funkcji kąta wychylenia steru. Wstępne wyniki tych badań wskazują na szereg interesujących efektów jak np. powstawanie na sterze dla małych kątów wychylenia wypadkowej siły zgodnej z kierunkiem ruchu statku, która w takim wypadku może być uważana jako dodatkowa składowa naporu. Zaprezentowana analiza może mieć zastosowanie w projektowaniu statków od których wymaga się dobrych właściwości manewrowych (np. gazowce LNG) lub statków wyposażonych w siłownię spalinowo-elektyczną, gdzie istnieje możliwość odzyskiwania energii podczas hamowania statku.

Introduction

LNG carriers are very technologically advanced ships from the design and operation point of view. Many research projects are carried out and the propulsion and manoeuvring performance are the very important studied characteristics. LNG carriers manoeuvre during significant part of their operation time and if we consider Polish gas terminal, manoeuvring characteristics may hold the key to the efficient operation. This is because of the restricted waterways in Danish straits and the approach to the terminal.

Presently, the application of dual-fuel-electric installation for propulsion systems on LNG carriers is wider and its use is reasonable when propeller characteristics can be determined also for off-design conditions. Efficient propeller with dual-fuel electric machinery may provide excellent propulsion characteristics for navigation in restricted routes, due to the availability of full propeller thrust at zero speed or vice versa the possibility of energy recovering during braking. Thus it is advisable to put some effort into more precise analysis of propeller-rudder system working in off-design condition.

Scope of analysis

The analysis of propeller-rudder system has been carried out for full-scale propeller with an advance coefficient of $J = 0.85$. The propeller's rotational velocity was 90 rpm and the ship speed $V = 7.65$ m/s. For the considered open water propeller it is the condition where torque coefficient becomes negative. The propeller is rotated by the flow as a turbine This often happens when ship is reducing rpm for slowing down. The situation is presented in figure 1. The propeller geometry is given in table 1.

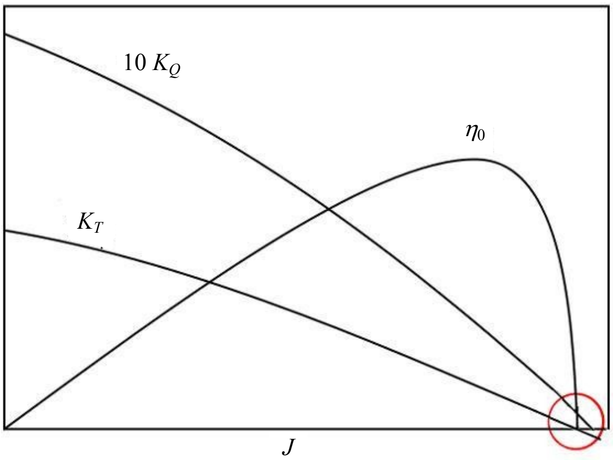

Fig. 1. For certain values of advance coefficient J torque coefficient becomes negative – the propeller works as a turbine
Rys. 1. Dla pewnych wartości współczynnika posuwu J współczynnik momentu zmienia znak na ujemny – śruba pracuje jak turbina

Table 1. The propeller geometry
Tabela 1. Geometria śruby

Pitch ratio P/D at $0.75r$ [–]	Diameter D [m]	Blade area ration $AE/A0$	Number of blades Z
0.739	5.9	0.578	4

It will be presented in the paper how the rudder influencing this characteristics. The entire system of rudder and propeller was modelled and the solution was unsteady for rotation of propeller and rudder. It was necessary to employ two rotation axis for two computational domains with non-matching sliding interface between them. The rudder was as well analysed in free stream – for the purpose of determination of its drag force.

The main research problem of carried analysis was to determine the propeller characteristics: thrust coefficient K_T, torque coefficient K_Q and propeller efficiency η_0. These are defined by the following formulae:

$$K_T = \frac{T}{\rho \cdot D^4 \cdot n^2} \tag{1}$$

$$K_Q = \frac{Q}{\rho \cdot D^5 \cdot n^2} \tag{2}$$

$$\eta_0 = \frac{K_T}{K_Q}\frac{J}{2\pi} \tag{3}$$

where: K_T – thrust coefficient [–], K_Q – torque coefficient [–], n – propeller's rotational speed [1/s], D – screw diameter [m], ρ – water density [kg/m^3], J – advance coefficient defined as:

$$J = \frac{V_A}{n \cdot D} \tag{4}$$

where: V_A – the resulting propeller's inflow [m/s].

Research method

The calculations were carried out using the finite volume method with unsteady formulation. The Ansys Fluent solver was applied. The flow is computed with turbulence modelling. The governing equations are the continuity equation for mass conservation. The continuity equation can be written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{5}$$

and momentum conservation in the form of RANS (Reynolds–Averaged–Navier–Stokes), for x, y, z directions:

$$\begin{gathered}
\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho F_x - \frac{\partial P}{\partial x} + \\
+ \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \rho\left(\frac{\partial \overline{u'u'}}{\partial x} + \frac{\partial \overline{u'v'}}{\partial y} + \frac{\partial \overline{u'w'}}{\partial z}\right) \\
\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho F_y - \frac{\partial P}{\partial y} + \\
+ \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) - \rho\left(\frac{\partial \overline{u'v'}}{\partial x} + \frac{\partial \overline{v'v'}}{\partial y} + \frac{\partial \overline{v'w'}}{\partial z}\right)
\end{gathered} \tag{6}$$

or in tensor notation:

$$\begin{gathered}
\rho\frac{\partial U_i}{\partial t} + \rho U_j\frac{\partial U_i}{\partial x_j} = \rho F_i - \frac{\partial P}{\partial x_i} + \\
+ \mu\left(\frac{\partial^2 U_i}{\partial x_j \partial x_j}\right) - \rho\left(\frac{\partial \overline{u_i' u_j'}}{\partial x_j}\right)
\end{gathered} \tag{7}$$

where: u, v, w – components of vector U of averaged velocities, P – pressure, μ – dynamic viscosity, u', v', w' – fluctuation parts of velocity vector, F_x, F_y, F_z – volumetric forces.

The above system of equations is not closed and must be supplemented with the equations modelling turbulence. For this purpose the two-equation $k–\omega$ model has been employed. In this model k stands for kinetic energy of turbulence and ω is the specific dissipation rate. The $k–\omega$ model implemented in Ansys Fluent solver is the Wilcox's model. Transport equations for k and ω are the following:

$$\frac{\partial}{\partial t}(\rho_k)+\frac{\partial}{\partial x_i}(\rho_k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+G_k-Y_k+S_k$$

$$\frac{\partial}{\partial t}(\rho_\omega)+\frac{\partial}{\partial x_i}(\rho_\omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+S_\omega \tag{8}$$

where: G_k – represents the generation of turbulence kinetic energy due to mean velocity gradients, G_ω – represents the generation of specific dissipation rate ω, Γ_k and Γ_ω – represent the effective diffusivity of k and ω, Y_k and Y_ω – dissipation of k and ω due to turbulence, S_k and S_ω – are user-defined source terms.

Computational domain

The computational domain, presented in figure 2, has the form of a cylinder with the geometrical model of the propeller and rudder inside it. The total number of mesh elements was 7 500 000. Boundary layer region was discretized with prism, while the rest of the domain is filled with non-structural tetrahedral elements. The method of discretization is given in figure 3.

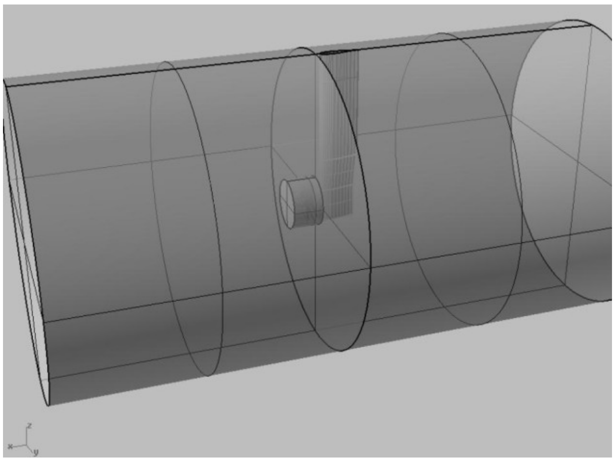

Fig. 2. Cylindrical computational domain with propeller and rudder
Rys. 2. Dziedzina obliczeniowa w postaci walca zawierająca modele geometryczne śruby i steru

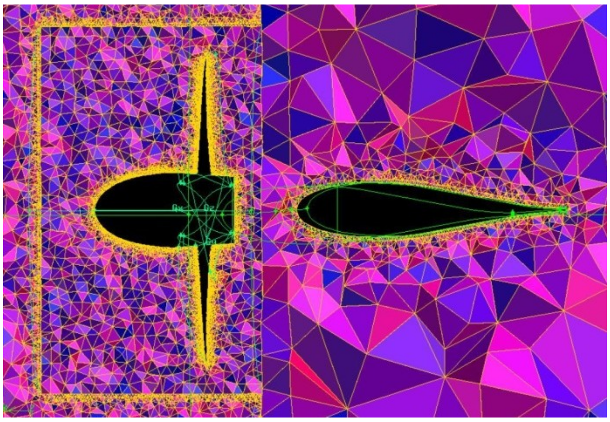

Fig. 3. Numerical grid for rudder and propeller
Rys. 3. Siatka numeryczna wokół śruby i steru

Screw propeller and the rudder constitute together a flow device in the configuration of the rotor-stator. They are placed very close to each other thus the hydrodynamics effects on the one of the devices have substantial influence on the flow features of the other. Apart from a strong influence of the propeller on the rudder performance caused by vorticity of propeller downstream, there are as well opposite effects. The rudder affects the flow before a propeller despite it is located behind it. It is a device which on account of its geometry straighten propeller's slipstream and in consequence decreases induced, circumferential velocities. This affects the thrust force generated by the propeller, changing direction of thrust's components for individual screw blades. The rudder may as well change the advance coefficient J. From propeller efficiency definition (formula 3) it appears that the change of any of: J, K_T, K_Q carries with it modification of the efficiency. Although the geometry of the propeller do not fully match the real one due to unrealistic ending of the hub (Fig. 3), which was due to the authors' oversight, the overall results of the computed forces should at least illustrate the trends of hydrodynamics effects and the complication of the presented problems. Obviously, any further research will start from the geometry improvement.

Calculation results and conclusions

The conducted research demonstrates an important influence of the rudder on propeller performance. Results were studied on the basis of propeller thrust and torque plotted as a function of rudder angle. Moreover, the drag force of rudder in free stream is compared to its drag force when working behind the propeller.

For example the resulting thrust force of propeller computed together with the rudder is given in figure 4. It is clearly visible, that while for considered advance coefficient the thrust of open water propeller is negative, it becomes positive when placed in front of the rudder, even for zero angle of rudder deflection. When the angle of deflection increases the thrust increases as well and actually becomes several times greater when the rudder deflection achieves 40 degrees. This very strong effect comes mostly from slowing the flow by the rudder. It should be noted here that simulated case must be considered very theoretical and similar situation hardly takes place in real manoeuvring, when drift angle occurs.

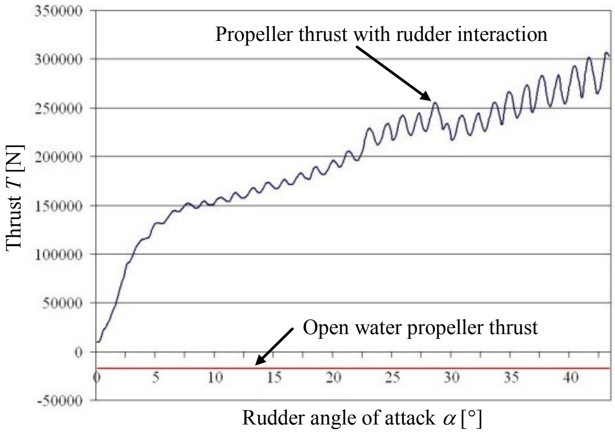

Fig. 4. Computed thrust as a function of rudder deflection angle
Rys. 4. Obliczona siła naporu pędnika w funkcji kąta natarcia steru

Similar effect can be observed for the torque. The situation presented in figures 4 and 5 might induce drawing a conclusion that presented influence of the rudder is so strong that changes propeller flow from ahead to almost bollard pull condition. The computed total force of propeller-rudder system is given in figure 6. For small angles of deflection it becomes initially greater but after a reaching maximum value it decreases. This is due to very large drag force acting on the rudder for greater angles of deflection. This force can be observed in figure 7 with the comparison of drag force on rudder in free stream (Fig. 8). The comparison is very interesting and this time the influence of propeller on rudder can be studied. For small angles of deflection rudder behind the propeller has negative values of drag and this means the thrust from rudder is generated. That effect corresponds well with figure 6, where a maximum of thrust force for rudder-propeller system for small deflection angles occurs.

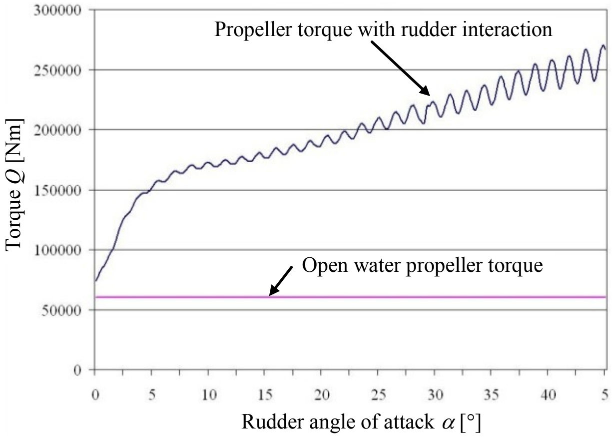

Fig. 5. Computed torque as a function of rudder deflection angle
Rys. 5. Obliczony moment pędnika w funkcji kąta natarcia steru

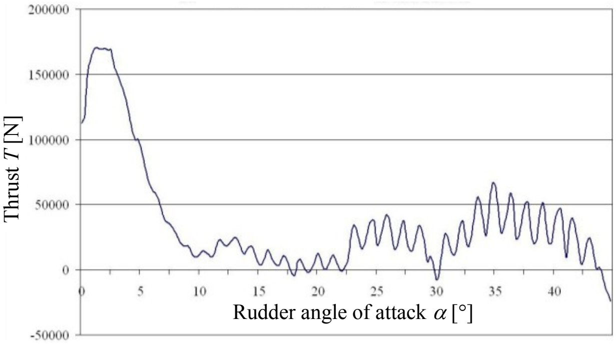

Fig. 6. Computed total force of rudder and propeller system
Rys. 6. Całkowita siła układu pędnik-ster

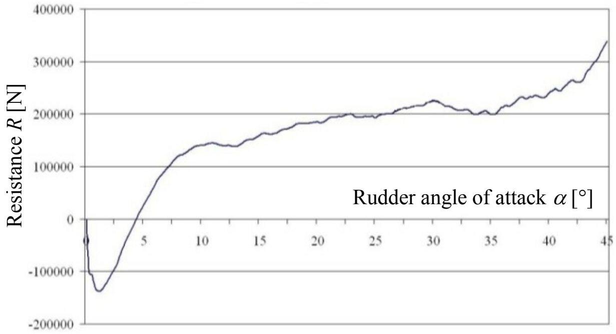

Fig. 7. Drag force of rudder behind the propeller
Rys. 7. Siła oporu steru za śrubą

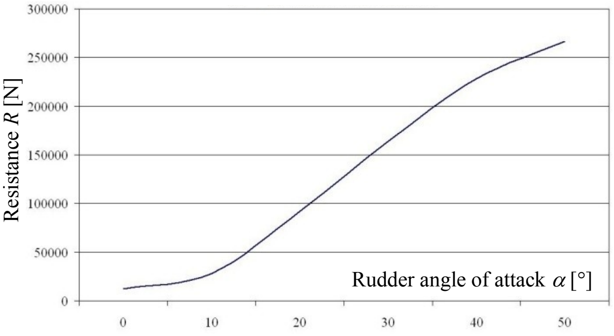

Fig. 8. Drag force of rudder in free stream
Rys. 8. Siła oporu steru swobodnego

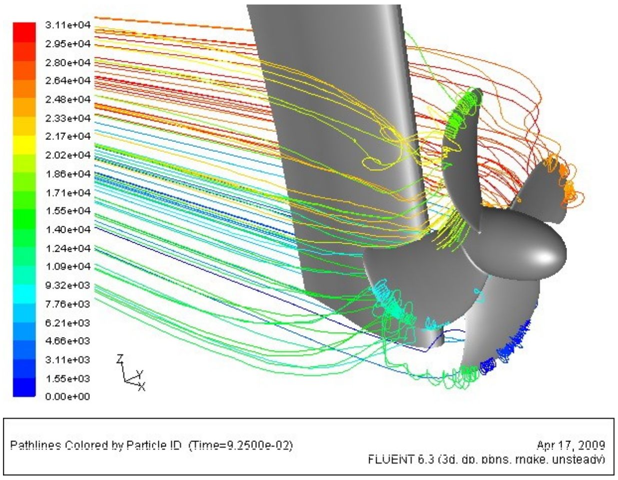

Fig. 9. Computed pathlines
Rys. 9. Obliczone linie prądu

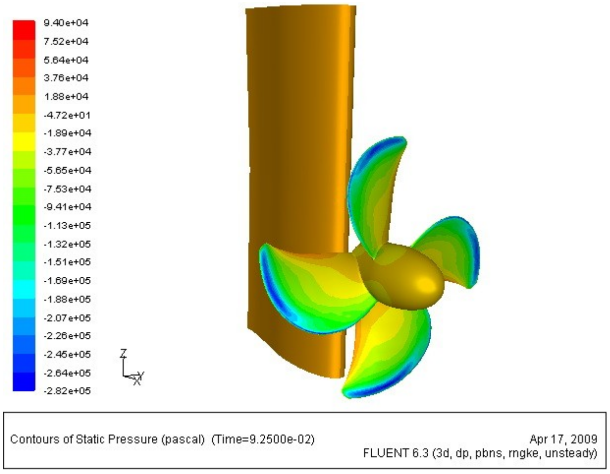

Fig. 10. Pressure on propeller blades
Rys. 10. Rozkład ciśnienia na skrzydłach śruby

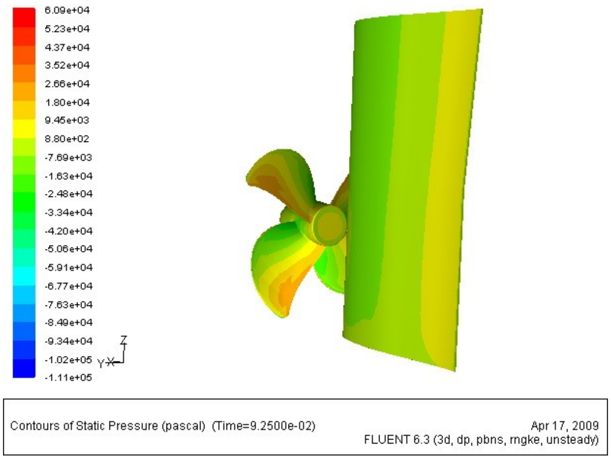

Fig. 11. Pressure on propeller blades and rudder surface
Rys. 11. Rozkład ciśnienia na skrzydłach śruby i powierzchni steru

From the results of analysis it can be found that due to very strong interactions between rudder and propeller their performance should be investigated with the use of models capable to catch all presented effects. What is more, the modelling of a rudder or a propeller separately may lead to doubtful results, especially when high accuracy is needed for the purpose of manoeuvring performance analysis.

It is worth to observe that for the calculated rudder-propeller configuration, for the small angles of deflection (and this can happen in service), forces on the rudder have resultant direction coincident with the direction of ship movement. This effect is caused by the flow with non-zero angle of attack which comes from the water rotation due to propeller. This resulting force has a higher value than a component from viscosity so that the total force can be considered as a part of the overall thrust force.

Next research should consists of analysis for different geometry configurations and as well the drift angle effects should be taken into account. If the inflow velocity will be assumed very low like during ship crabbing the flow characteristics under such conditions can be investigated.

Presented analysis can be applied for more precise investigations for ships which are expected to reach high manoeuvring performance or those where diesel-electric propulsion systems are to be installed.

References

1. ABRAMOWSKI T.: Prediction of Propeller Forces during Ship Manoeuvring. Journal of Theoretical and Applied Mechanics, 2005, 43,157–178.
2. ABRAMOWSKI T.: Application of artificial neural networks for determination of propeller's crash-ahead, crash-back and backing performance. Ship Technology Research, 2001, 48, 4, 154–160.
3. BERTRAMV.: Practical Ship Hydrodynamics. Butterworth–Heinemann, 2000.
4. EL MOCTAR O.M.: Selected Topics of CFD for Ship Manoeuvring. INSEAN, Rome, 2001.
5. NATARAJAN S.: Computational Modelling of Rudder Cavitation and Propeller / Rudder Interaction. The University of Texas at Austin, 2003.
6. SIMONSEN C.D.: Rudder, Propeller and Hull Interaction by RANS (PhD thesis). Technical University of Danmark–Lyngby, 2000.

Recenzent:
prof. dr hab. inż. Jan Szantyr
Politechnika Gdańska