ISSN 0209-2069

ZESZYTY NAUKOWE NR 2 (74)
AKADEMII MORSKIEJ
W SZCZECINIE

EXPLO-SHIP 2004

Tomasz Abramowski

Numerical Wake Assessment of Maneuvering Ship

Key words: nominal wake, maneuvering

The article presents an attempt at numerical wake determination of a ship moving with drift. For the calculation of viscous flow a numerical method based on the solving of RANS equations has been applied. A bulk carrier hull has been chosen for the analysis. The results show complex features of the flow in the stern area. Distributions of wake factor are strongly non-uniform both in radial and circumferential directions and the non-uniformity gets stronger as drift angle increases.

Numeryczna analiza strumienia nadążającego manewrującego statku

Słowa kluczowe: nominalny strumień nadążający, manewrowanie

Zaprezentowano próbę numerycznej oceny strumienia nadążającego statku poruszającego się z dryfem. Obliczenia opływu lepkiego wykonano metodą opartą na rozwiązaniu równań RANS. Do analizy wybrano kadłub masowca. Obliczenia wskazują na skomplikowane parametry przepływu w rejonie rufy. Rozkłady współczynników strumienia nadążającego są silnie niejednorodne tak promieniowo, jak i obwodowo i niejednorodność ta wzrasta wraz ze wzrostem kąta dryfu.

Introduction

A lot of attention has been paid recently to the safety of sea traffic. Maneuvering simulators are an important part of these studies. They are presently a widely accepted tool for the determination of ship maneuvering features, while there are many areas where additional research could be purposeful. In thiswork interest is given to wake characteristics of a maneuvering ship. The *Fluent Inc.* solver has been applied for the analysis

Present methods for the determination of propeller-hull interaction during maneuvering are mainly based on model tests results. The authors like *Inoue et al*. [3] and *Kobyliński and Zolfaghari* [4] present the results of simulations with experimentally derived thrust and wake factors. Few attempts have been made to assess these factors numerically. *Simonsen* [8] carried out calculations for rudder in free stream, rudder behind a propeller and bare hull moving straight ahead without drift. *Le Thuy Hang* [6] presents an analysis of the propeller-rudder interaction with lifting surface theory applied to calculations. *Yosukuawa et al.* [9] have developed a method for calculations of forces acting on a ship and hull-propeller interaction coefficients. The method is almost purely theoretical, but wake factor is obtained experimentally. *El Moctar* [1] applies finite volume method to flow calculations of a ship hull. The results of hull forces are presented as a function of drift angles.

1. Applied numerical method

For the calculation of viscous flow around a ship with drift we have applied numerical method based on the solving of equations governing the case under consideration, i.e. RANS equations, having the following form:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho F_1-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)-\rho\left(\frac{\partial\overline{u'u'}}{\partial x}+\frac{\partial\overline{u'v'}}{\partial y}+\frac{\partial\overline{u'w'}}{\partial z}\right)$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho F_2-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)-\rho\left(\frac{\partial\overline{u'v'}}{\partial x}+\frac{\partial\overline{v'v'}}{\partial y}+\frac{\partial\overline{v'w'}}{\partial z}\right)$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho F_3-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)-\left(\frac{\partial\overline{u'w'}}{\partial x}+\frac{\partial\overline{v'w'}}{\partial y}+\frac{\partial\overline{w'w'}}{\partial z}\right) \quad (1)$$

In the above equations u, v, w are components of the mean velocity vector, P is the pressure, μ is the viscosity, u', v', w' are fluctuation parts of velocity vector, F_1, F_2, F_3 are volumetric forces.

Furthermore, the model must satisfy continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2}$$

For the modeling of the Reynolds stresses the *RNG k-ε* turbulence model has been applied. The placement of grid first point was established on the basis of non-dimensional parameter y^+, describing local Reynolds number. The y^+ may be determined according to the theory of flat-plate flow, e.g. *Schlichting* 7:

$$y^+ = 0{,}172\left(\frac{y}{L}\right)R_n^{\,0,9} \tag{3}$$

where: y is the distance from the wall, L is the body length.

In the considered case wall functions have been applied, with $y^+ = 50$. Calculations were carried out for the Reynolds number of the model, keeping Froude number similarity. The scale factor was $\lambda = 30$.

The computational domain, presented in Fig. 1, was designed intentionally for the purpose of velocity direction changing at the inlet boundary. The outer boundary is a surface of revolution arising from the revolution of a trapezoid placed at the waterline.

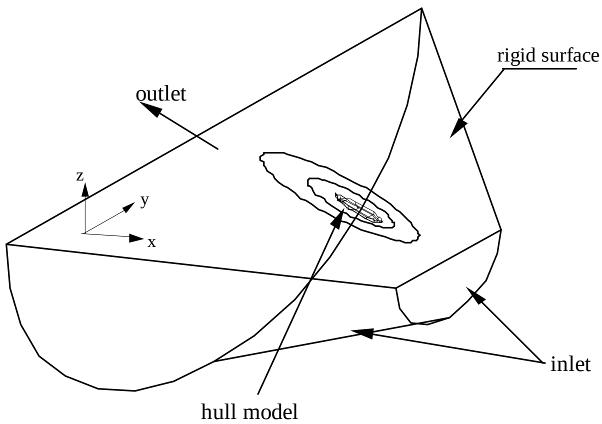


Fig. 1. The shape of computational domain
Rys. 1. Kształt dziedziny numerycznej

Four types of boundary conditions were placed at outer surfaces of computational domain:

1. Free surface was replaced with a rigid boundary and is treated as a plane of symmetry, for which conditions of zero normal velocity and zero gradient of other components must be satisfied. This assumption may be legitimate taking into account that Froude number is relatively low ($F_n = 0.16$).
2. The boundary condition on the hull is a no-slip condition with zero relative speed enforced.
3. At the inlet boundary condition all flow parameters must be specified: components of free stream velocity vector, kinetic energy of turbulence and its dissipation rate.
4. The outlet condition was set such that longitudinal gradients of velocity and pressure are equal to zero:

$$\frac{\partial(u,v,w,p)}{\partial x} = 0 \tag{4}$$

The arrangement of boundary conditions is presented in Fig. 1. The flow was computed for drift angles varying from 0° to 35°, with the 5° step.

2. Model parameters and numerical grid topology

The hull of a bulk carrier has been chosen for the analysis. The intention was to obtain relatively complex flow pattern in the stern area of a shape having a high block coefficient. Parameters of the ship hull together with flow numbers characteristics are presented in Table 1. The sketch of body lines is given in Fig. 2.

Table 1

Parameters of ship and model
Parametry statku i modelu

Length , *L*	185 m	6.167 m
Breadth, *B*	25.3 m	0.843 m
Draught, *T*	10.65 m	0.355 m
Speed , *V*	14 w	1.315 m/s
Froude number, *Fn*	0.16	0.16
Reynolds number, *Rn*	$1.4 \cdot 10^9$	$6.8 \cdot 10^6$

We have applied *Tribon Initial Design* [5] system for the modeling of hull surface and the Gambit system for grid generation. The hybrid grid with tetrahedral elements placed in the most part of domain and prisms near the hull have been applied. The grid is presented in Fig. 3 (view from the bottom and stern part). Total number of elements was 680 000.

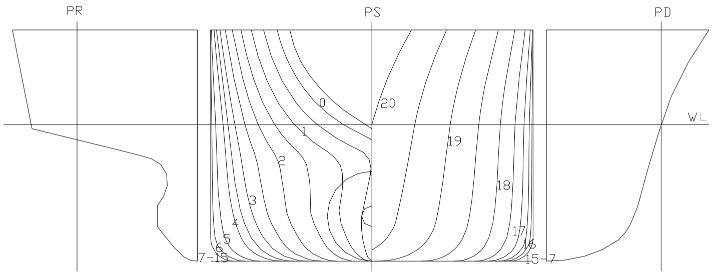


Fig. 2. Bulk carrier body lines
Rys. 2. Kształt kadłuba analizowanego masowca

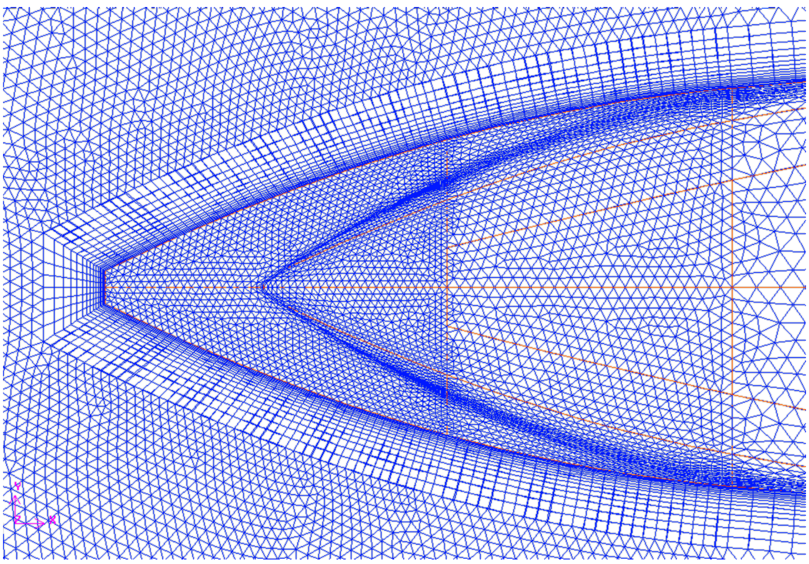


Fig. 3. View of the numerical grid in the stern area
Rys. 3. Siatka numeryczna w rejonie rufy. Widok od strony dna

3. The results

The results show complex features of the flow in the stern area. Streamlines are presented in Fig. 4 and 5 in the view from the stern. Formations of strong vortices are present, even for the straight ahead course. Distributions of nominal wake factor W_N are calculated according to Taylor method:

$$W_N = 1 - \frac{V_{AX}}{V_X} \tag{5}$$

In the above equation V_{AX} is the axial component of the velocity in the screw plane and V_X is the axial component of ship's speed.

Wake is strongly non-uniform both in radial and circumferential directions and the non-uniformity is stronger as drift angle increases. The flow straightening effect of the hull causes strong non-uniformity of the wake, which becomes visible in regions where backflow occurs. Mean values of wake factor, presented in Fig. 6, were calculated on the basis of its radial distribution:

$$W_N(r) = \frac{1}{360}\int_0^{360} W_N(\varphi)d\varphi \tag{6}$$

In the above equation r is assumed to be the radius of screw and φ is circumferential position. Hence, the mean value of wake factor was determined as:

$$W_N = \frac{1}{0{,}48}\int_{r=0,2}^{r=1} rW_N(r)dr \tag{7}$$

Unfortunately, we did not have any experimental results of wake and streamlines for a reliable verification process. We have attempted to assess the results for straight ahead course, related to viscous part of the total resistance. According to Froude's hypothesis the hull resistance can be divided between its viscous and residual parts:

$$C_T = C_R + C_V \tag{8}$$

where:

C_T is the total resistance coefficient,
C_R is the residual resistance coefficient,
C_V is viscous resistance coefficient.

The viscous coefficient is a function of Reynolds number and the shape of a ship. It can be expressed as:

$$C_V = (1+k_0)\, C_{F0} \tag{9}$$

where: k_0 is the form factor, C_{F0} is the frictional resistance coefficient. The frictional resistance coefficient and the form factor k_0 can be estimated by means of the *ITTC* friction line. The numerical algorithm calculates hydrodynamic forces by integration of normal pressure stresses and the frictional shear stresses over the hull surface. Hence, the total resistance coefficient can be expressed as a sum:

$$C_x = C_{px} + C_{fx} \tag{10}$$

where:

C_{px} is the resistance coefficient from the pressure,

C_{fx} is the frictional resistance coefficient.

Neglecting the free surface effect implies that residual coefficient C_R in (8) can be assumed to be equal to zero.

The pressure coefficient C_{px} expresses the form resistance of the hull, and it is possible to use it for the calculation of the form factor k_0 and C_{fx} can be compared with the *ITTC* formula. Hence the expression (8) can be written as:

$$C_T = k_0 C_{F0} + C_{F0} \tag{11}$$

By comparison of (10) with (11) the k_0 can written as:

$$k_0 = \frac{C_{px}}{C_{fx}} \tag{12}$$

The results of calculations presented above are shown in Table 2.

Table 2

Computed values of resistance coefficients and ITTC values

Obliczone wartości współczynników oporu porównane z wartościami uzyskanymi zgodnie z zaleceniami ITTC

	C_{Fx}	C_{px}	C_x	$1+k_0$
Calculations	$2.98\cdot10^{-3}$	$0.745\cdot10^{-3}$	$3.73\cdot10^{-3}$	1.25
ITTC	$3.21\cdot10^{-3}$	–	–	1.137

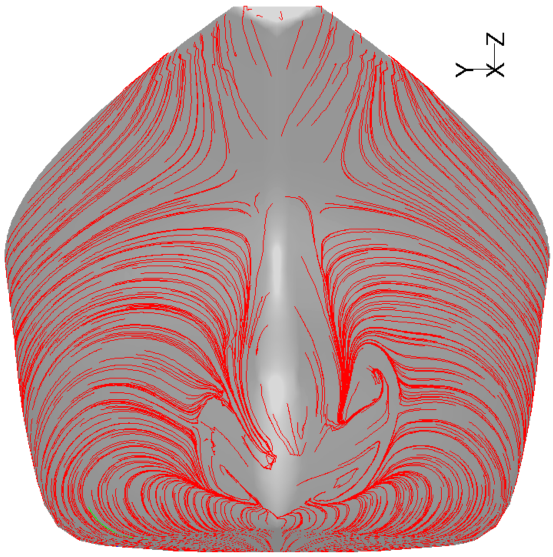


Fig. 4. Streamlines on the hull surface. Drift angle $\beta = 0°$. The view from the stern
Rys. 4. Linie prądu na powierzchni kadłuba w widoku od strony rufy. Kąt dryfu $\beta = 0°$

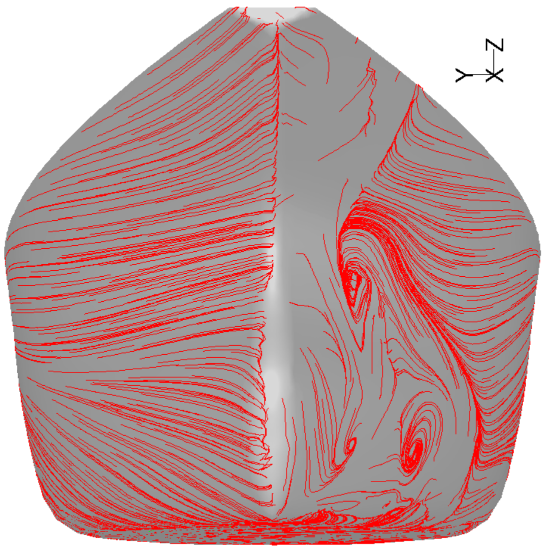


Fig. 5. Streamlines on the hull surface. Drift angle $\beta = 35°$. The view from the stern
Rys. 5. Linie prądu na powierzchni kadłuba w widoku od strony rufy. Kąt dryfu $\beta = 35°$

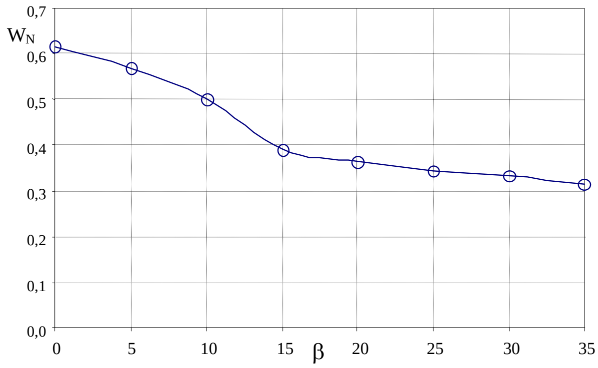


Fig. 6. The nominal wake factor plotted vs. drift angle

Rys. 6. Nominalny współczynnik strumienia nadążającego w funkcji kąta dryfu

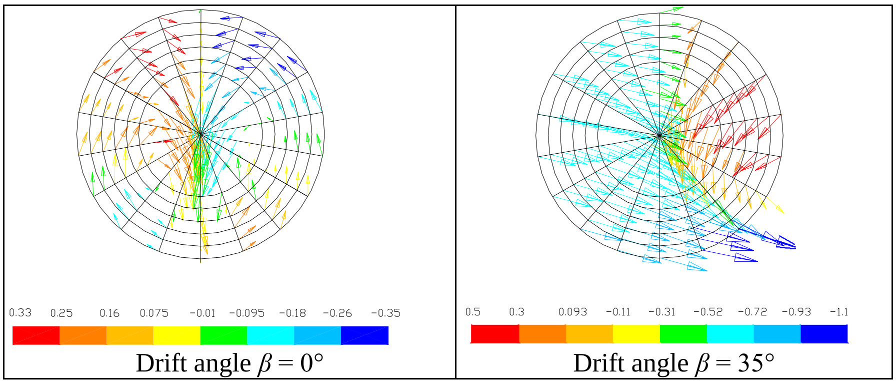


Fig. 7. Transverse velocities in propeller plane; V_Y [m/s]

Rys. 7. Prędkości poprzeczne w płaszczyźnie kręgu śrubowego; V_Y [m/s]

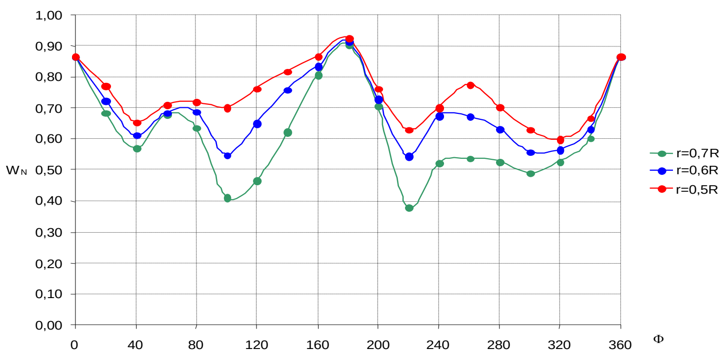


Fig. 8. Circumferential distribution of wake factor. Drift angle $\beta = 0°$
Rys. 8. Obwodowy rozkład strumienia nadążającego. Kąt dryfu $\beta = 0°$

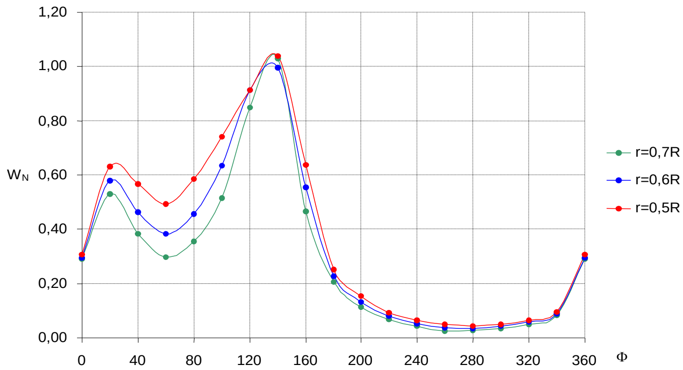
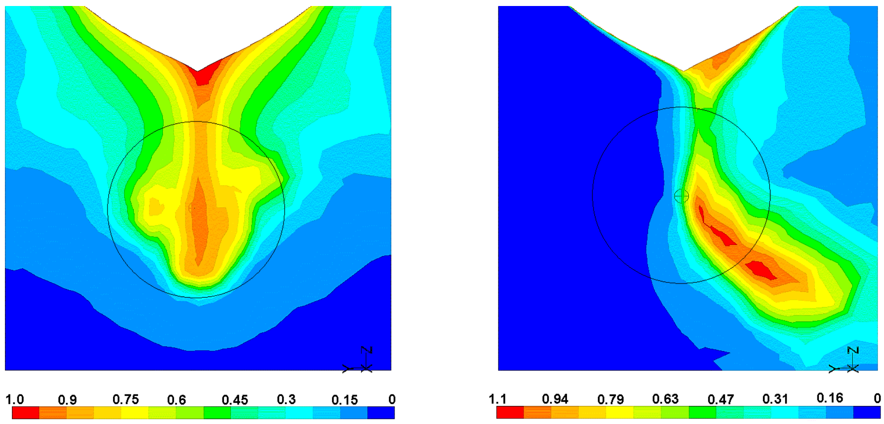


Fig. 9. Circumferential distribution of wake factor. Drift angle $\beta = 35°$
Rys. 9. Obwodowy rozkład strumienia nadążającego. Kąt dryfu $\beta = 35°$



Drift angle $\beta = 0°$ Drift angle $\beta = 35°$

Fig. 10. Wakes distributions in propeller plane
Rys. 10. Rozkłady strumienia nadążającego w płaszczyźnie kręgu śrubowego

Conclusions

The case of ship moving with drift causes complicated flow patterns in the stern region. Carrying out such calculations requires the application of most advanced numerical techniques, taking into consideration real hydrodynamics effects. The RANS viscous method has been applied and the results are at least qualitative for mean and integral values. Inaccuracies may come from an imperfection of the applied turbulence model and from the errors of numerical method. The results obtained for streamlines are especially interesting and complex vortex formations can be observed. The assumption neglecting free surface effects seems to be legitimate in the considered range of the Froude number and for the purpose of research, i.e. the investigation of wake. If the analysis aimed atthe calculation of hull forces, more accuracywould be advisable. The results of flow calculation can be applied in the algorithm for the determination of screw propeller forces in maneuvering conditions.

Literature

1. El Moctar O.M., *Numerical computations of flow forces in ship maneuvering*, Ship Technology Research, Vol. 48, 2001.
2. Fluent INC.: Fluent 5 User's Guide, 1998.
3. Inoue S., Hirano M., Kijima K., Takashina J., *A Practical Calculation Method of Ship Maneuvering Motion*, Int. Ship. Progress, vol. 28, 1981.
4. Kobyliński L., Zolfaghari G., *Prediction of maneuvering characteristics in ship design*, 12th Int. Conf. on Hydrodynamics in Ship Design, Szklarska Poręba, 17-19 September 1997.
5. Kockums Computer Systems Ltd., *Tribon Init. Design User Guide*, 1998.
6. Le Thuy Hang, *Calculation of the influence of propeller operation on the hydrodynamic characteristics of the rudder*, Polish Maritime Research, no 2(28), vol. 8, 2001.
7. Schlichting H., *Boundary Layer Theory*, McGraw-Hill, New York, 1968.
8. Simonsen C.D., *Rudder, Propeller and Hull Interaction by RANS*, PhD thesis, T.U. of Denmark, 2000.
9. Yasukawa H., Yoshimura Y., Nakatake K., *Hydrodynamic Forces on a Ship with Constant Rudder Angle*, Int. Conf. On Marine Simulation and Ship Maneuverability, Rotterdam 1996.

Wpłynęło do redakcji w lutym 2004 r.

Recenzenci

prof. dr hab. inż. Tadeusz Szelangiewicz
dr inż. Zbigniew Szozda

Adres Autora

dr inż. Tomasz Abramowski
Politechnika Szczecińska, Wydział Techniki Morskiej
e-mail: tomasz.abramowski@ps.pl