Scientific Journals
Maritime University of Szczecin

Zeszyty Naukowe
Akademia Morska w Szczecinie

2014, 40(112) pp. 27–30
ISSN 1733-8670

Fishing vessel hull design and towing resistance calculation by the CFD methods

Karol Sugalski

West Pomeranian University of Technology, Faculty of Marine Technology
71-065 Szczecin, al. Piastów 41, e-mail: karol.sugalski@zut.edu.pl

Key words: fishing vessel, stern trawler, resistance calculation, computational fluid dynamics

Abstract

Hull resistance and propulsion calculations are the basis of every ship design. In this paper, application of the computational fluid dynamics are presented together with the results of towing resistance of the model of stern trawler. The practical use of CFD in the preliminary vessels' design process were presented at the stern trawler vessel type.All calculations had been performed in the free CFD software called OpenFOAM. This is set of C++ programming language libraries, designed to solve Navier-Stokes equations.

Introduction

During preliminary design of a new vessel, with respect to the ship-owner needs, naval architect tries to find most optimal hull shape that fits all demands, for example, minimal hull resistance for a given velocity of the vessel. At the beginning of the design process, hull resistance and needed engine power are estimated by analytical equations, after that, towing tank tests should be performed but they are very expensive.

Nowadays, for hull shape, resistance and power calculation the CFD (Computational Fluid Dynamics) methods are widely used. Results are accurate and also confirmed by towing tank tests [1, 2, 3]. CFD methods are particularly useful for unusual shaped vessels, or for those which analytical equations for resistance are inaccurate.

In this paper, the practical use of CFD in the preliminary vessels' design process were presented at the stern trawler vessel type. Towing resistance of bare hull was calculated by using two phase viscous flow. Wave system generated by the moving hull was also obtained in the calculations.

All calculations were performed in the free CFD software called OpenFOAM. This is set of C++ programming language libraries, designed to solve Navier-Stokes equations. Mesh generators and postprocessor are included in the software package.

Governing equations and fluid to hull influence

Influence of fluid to the rigid wall can be calculated with the so called Navier-Stokes equations (2). They are also used to determine velocity field and pressure inside the volume of moving fluid. Combined with the continuity equation (1), the Navier-Stokes equations, among all other applications allow to calculate hull resistance of the moving vessel. They are presented below [4].

$$\frac{\partial V_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial V_i}{\partial t} + V_j \frac{\partial V_i}{\partial x_j} = F_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + v \frac{\partial^2 V_i}{\partial x_j \partial x_i} \tag{2}$$

where:

$i, j = 1, 2, 3$;

$V_{i,j}$ – components of main velocity vector;

F_i – components of mass force vector;

ρ – fluid density;

v – kinematics viscosity.

Navier-Stokes equations are the only choice for dynamics of Newtonian fluids but they can’t be solved in the turbulent case [4, 5]. When turbulences are important, modified form of N–S equations should be used. They are called Reynold

Averaged Navier Stokes Equations also known as RANSE method [6]. It is shown below (3):

$$\frac{\partial V_i}{\partial t} + \frac{\partial V_i V_j}{\partial x_j} = F_i + \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_i} \tag{3}$$

where τ_{ij} is a tensor of mean stresses (4):

$$\tau_{ij} = -P\delta_{ij} + \mu\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right) + \bar{\tau}_{ij} \tag{4}$$

where:
δ_{ij} – Kronecker delta;
P – pressure;
μ – dynamic viscosity;
$\bar{\tau}_{ij}$ – turbulent stress tensor.

Pressure and velocity field related to the hull was calculated with the RANSE method. Mentioned equations needs some kind of space that describes given experiment. This space is called the numerical grid or mesh. Values of pressure and velocity are calculated with the finite volume method. OpenFOAM is based on this method.

Concept of finite volume method is to discretize governing equations, written in the integral form, on the physical mesh describing given geometry. Values of variables are stored in the cell centers [7].

In the turbulent case, RANSE equations do not form a closed system. Adding additional equations to close the system. Those equations comes from the appropriate turbulence model.

Standard k-ε model [7] of turbulences was good enough to the towing resistance problem. In the flow separation region this model can be inaccurate but overall resistance values are close to the experimental data [1].

Hull geometry of the stern trawler

Towing resistance was calculated for the modified version of the B410 stern trawler. Simplified general arrangement is shown in figure 1. Three-dimensional model of the hull was created from the lines plan. The model shown in figure 2, was used to create numerical grid.

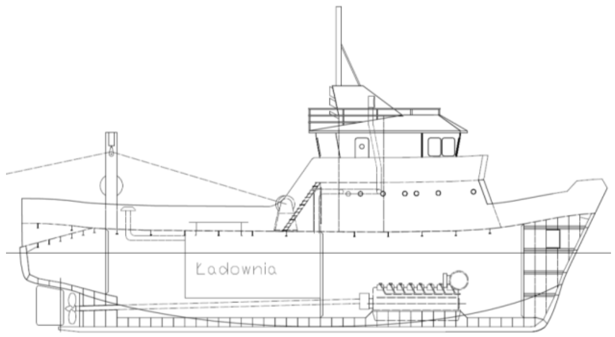

Fig. 1. Side view of B410 vessel

The hull dimensions are shown in table 1.

Table 1. Dimensions of hull type B410

LOA [m]	LWL [m]	*B* [m]	*T* [m]	*D* [T]	Cb
25.6	24.46	7.14	3.25	362.68	0.623

The geometry of the hull should be simple in non-essential regions. This may reduce the level of grid cells and save computation time.

Computational model of the stern trawler hull

To describe the phenomenon under study, the computational models exists, namely simplicity which are designed to present what happens in nature. For hydrodynamic analyzes were used two phase flow models. Newtonian model of fluid was chosen. Such fluid has a particular properties [4]:

- components of stress tensor are linear function of strain speed tensor components;
- isotropic, fluid behave uniformly in all directions.

All fluids had constant density and were viscous. The grid was created in the Cartesian coordinate system. The hull geometry still remained. Scale factor was $\lambda = 25$.

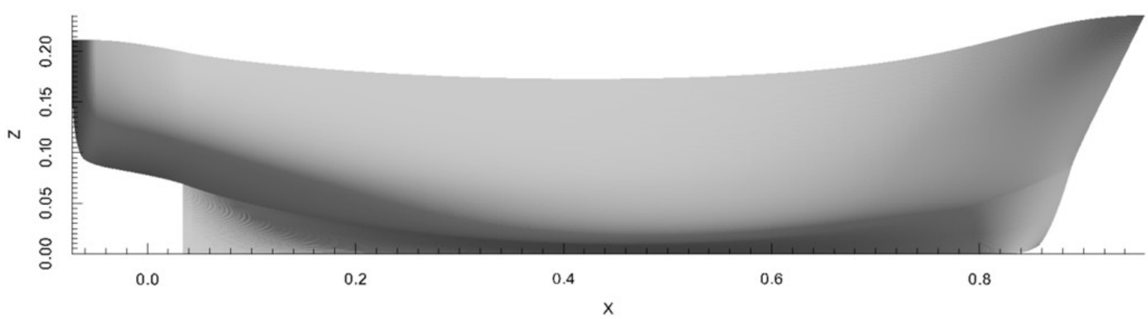

Fig. 2. Geometry of hull analysis. Model scale

Appropriate boundary conditions should be used to solve the Navier-Stokes equations. Those conditions describes how the equations behaves on the borders of the mesh. To avoid influence of boundary conditions to the solution, borders of the grid should be far away from the geometry. The boundary conditions used in this paper are most common and simple: rigid wall, symmetry plane, inlet and outlet. Mathematical explanation of above boundary condition can be found in [8]. In OpenFOAM, boundary conditions have to be defined for all computed variables. Details of the implemented boundary conditions can be found in [9].

Numerical grid is created from so called computational cells. The cells can have many different shapes [5]. In OpenFOAM all cells are of hexahedron type. In the vicinity of geometry, cells are twisted and folded, by a meshing program, to fit the shape of analyzed object.

Quality of the analyzed shape depends on the number of cells. More cells give also better and more accurate solution of governing equations.

Bare hull with no appendages was used for simulation. Three velocity cases were chosen. They are shown in table 3. The hull was cut in the symmetry plane to speed up calculations. In the domain two fluids were modeled: water and air. Water level was set to the draught of the vessel in the model scale. Properties of the computational model shown in table 2. The boundary conditions of the simulation shown in table 4.

Table 2. Characteristics of the hull analysis model

Flow type	Turbulent transient
Simulation time [s]	10
Air density [kg/m^3]	1
Water density [kg/m^3]	999
Kinematic viscosity of air[m^2/s]	$1.48 \cdot 10^{-5}$
Kinematic viscosity of water [m^2/s]	$1.13896 \cdot 10^{-6}$

Table 3. Velocities used in the hull analysis

Real scale velocity [kn]	Model scale velocity [m/s]	Froude number	Reynold's number
8	0.823	0.265	706981.1
11	1.132	0.3653	972421.1
15	1.5433	0.4981	1325739.9

Numerical grid

The upper part of the grid form a simple box, the lower part is a quarter of cylinder. The purpose of such shape is to limit cell number. The dimension of this domain can be found in table 5. The perspective view of the domain is illustrated in figure 3.

Table 4. Boundary conditions for hull analysis

Boundary name	Velocity field U	Pressure P	Alpha field	Turbulent kinetic energy k	Dissipation rate ε
Inlet	fixedValue	buoyantPressure	fixedValue	fixedValue	fixedValue
Outlet	zeroGradient	zeroGradient	zeroGradient	zeroGradient	zeroGradient
Bottom	symmetryPlane	symmetryPlane	symmetryPlane	symmetryPlane	symmetryPlane
Atmosphere	slip	slip	slip	slip	slip
Side wall	pressureInletOutletVelocity	totalPressure	inletOutlet	inletOutlet	inletOutlet
Symmetry Plane	symmetryPlane	symmetryPlane	symmetryPlane	symmetryPlane	symmetryPlane
Hull	fixedValue	buoyantPressure	kqRWallFunction	epsilonWallFunction	zeroGradient

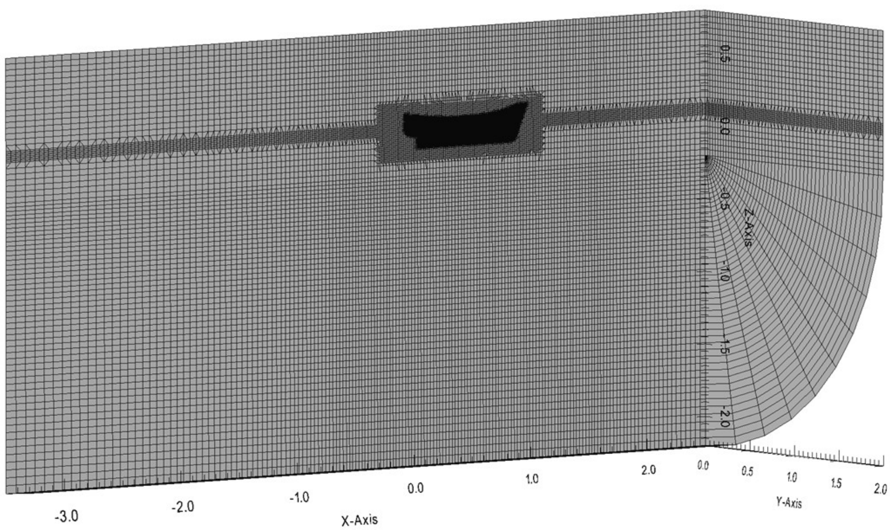

Fig. 3. Overall view of hull analysis domain

Table 5. Dimensions of the hull analysis domain

Length of domain on x-axis [m]	Length of domain on y-axis [m]	Length of domain on z-axis [m]	Cell number
6	2	3	1 245 506

Grid refinement visible in figure 3, in the free surface region, allows to reproduce waves with good accuracy.

Results of numerical computations

The results of towing resistance calculations are shown in table 6. Resistance values are averaged over time $t \in (3, 10)$ of the simulation.

Table 6. Computed resistance of the bare hull

Velocity [m/s]	Total resistance of bare hull [N]
0.823	1.03
1.132	2.729
1.5433	18.879

Calculated waves system shown in figures 4, 5 and 6.

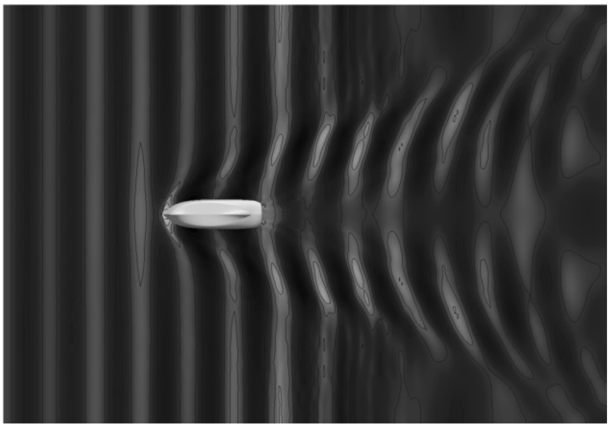

Fig. 4. Wave pattern for velocity 0.823 m/s

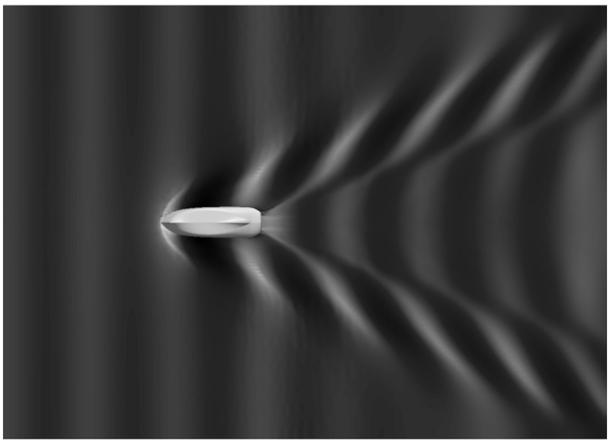

Fig. 5. Wave pattern for velocity 1.132 m/s

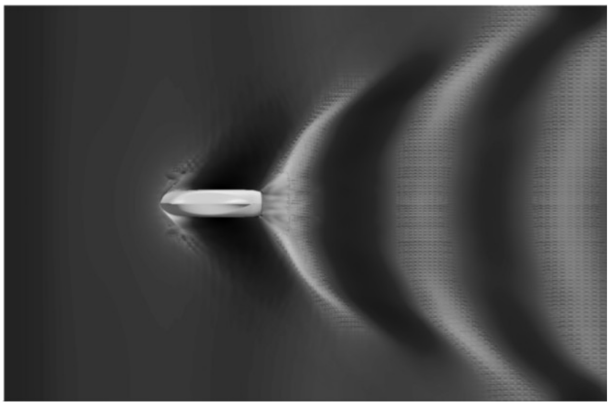

Fig. 6. Wave pattern for velocity 1.5433 m/s

Conclusions

The towing resistance and waves system of the model of stern trawler were calculated. Values of resistance can be used to determine engine power needed for contracted speed of the vessel. In figures 5 and 6 change in the wave system can be seen. This phenomenon occurs for high Froude number. This may be proof of good accuracy of the selected methods of calculation.

CFD can be useful tool for preliminary design of ships. Many variants of the hull can be tested before creating physical model for the towing tank experiment.

References

1. SZELANGIEWICZ T., ABRAMOWSKI T.: Numerical analysis of influence of ship hull form modification on ship resistance and propulsion characteristics. Polish Maritime Research 16, 4, 2009.
2. Numerical analysis of influence of ship hull form modification on ship resistance and propulsion characteristics. Part II. Polish Maritime Research 17, 1, 2010.
3. SZELANGIEWICZ T., ABRAMOWSKI T., ŻELAZNY K.: Numerical analysis of effect of asymmetric stern of ship on its screw propeller efficiency. Polish Maritime Research 17, 4, 2010.
4. GRYBOŚ R.: Podstawy Mechaniki Płynów. Wydawnictwo Naukowe PWN, Warszawa 1998.
5. BLAZEK J.: Computational Fluid Dynamics: Principles and applications. Second Edition. Elsevier, New York 2005.
6. OERTEL H.: Prandtl's Essentials of Fluid Mechanics. Springer, New York 2004.
7. FERZIGER J.H., PERIĆ M.: Computational Methods for Fluid Dynamics. Springer, Berlin 2002.
8. ANDERSON J.D. JR.: Compoutational Fluid Dynamics. McGraw-Hill Inc., New York.
9. ESI-OpenCFD. www.openfoam.org [online] ESI-Open-CFD, 2014. http://www.openfoam.org/docs/cpp/.