Water interference effect on ship due to square shaped object shielding

Ionuț Cristian SCURTU, Cătălin CLINCI, Adrian POPA
Naval Academy “Mircea cel Batran”, Constanța, Romania
E-mail: ionut.scurtu@anmb.ro

Abstract. This scientific paper presents performed numerical 2D Forces resulted from pressure variation due to interference with a square shaped object are analyzed by computer fluid dynamics module ANSYS. Using the 2d geometry we created a three-dimensional mesh around the ship with water speed $v=ct$ and variable shielding are used for analysis. Interaction is computed on 2D model for distances between objects $0.2L…0.9L$, where $L$ is the length of the ship. Various positions of the square shaped object will determine different case as presented in the fluid flow simulation in a restricted domain simulating open sea operation. The pressure for each case can be easily found in CFD-Post and interference is presented for ship operation in open sea.

1. Introduction

Water speed $v=ct$ and current variable incidence angle are used for analysis in Workbench parametric simulation. Interaction is computed on 2D model for distances between objects $0.2L…0.9L$, where $L$ is the length of the ship. The main goal is to determine the numerical water interference effect on ship performed by computational tools provide appropriate approaches for calculations of the complex 2D presented model characteristics. The main value of the paper is that the obtained results with the realized numerical flow simulation can be compared with the experimental data measures in ship operation near shielding objects.
2. Mathematical Model

The performed numerical simulation is governed by the continuity, Navier-Stokes (momentum conservation) and energy equations. But the flow is turbulent, that’s why it was used Reynolds Average version of their equations. Therefore it was Standard $k$-$\varepsilon$ Model turbulence model. In the beginning, it has six variables to solve for: 3 components of velocity, pressure, temperature, and kinematic eddy viscosity. The equations are:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]  
(1)

\[
\rho \left( \frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j} (U_i U_j) \right) = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2\mu S_{ij} - \rho u_i u_j \right)
\]  
(2)

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \frac{\mu + \mu_t}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_{\varepsilon}
\]  
(3)

Where (1) is Continuity Equation,  
(2) is Reynolds Averaged Navier-Stokes equation,  
(3) is conservation of energy equation,

Using the approach from [1] the given above equation are converted to algebraic equations. It then solves for our six variables ay each of the cell centers of our mesh. This means that if we have 3,000,000 cells, CFX is going to solve 4.8 million equations to solve the problem.

3. Standard $k$-$\varepsilon$ Model theory used in calculation by Ansys Software.

Two-equation turbulence models allow the determination of both, a turbulent length and time scale by solving two separate transport equations. The standard $k$-$\varepsilon$ model in ANSYS Fluent falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Fast calculation and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

The turbulence kinetic energy, $k$, and its rate of dissipation, $\varepsilon$, are obtained from the following transport equations:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]

and

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \varepsilon \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_2 \rho \frac{\varepsilon^2}{k} + S_{\varepsilon}
\]

In these equations, $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as described in Modeling Turbulent Production in the $k$-$\varepsilon$ Models.
Various positions of the square shaped object will determine different case as presented in the fluid flow simulation in a restricted domain simulating open sea operation in a 2D domain 12L X 10B where L is ship length and B is ship breadth. S1 and S2 dimensions are used to determine square object position relative to ship bow.

The pressure for each case can be easily found in CFD-Post and interference is presented for ship operation in open sea as presented below.
Boundaries not used are set as no friction surfaces for a simulation reflecting interference in open sea. Differences in pressure contour on each side can be easily computed in lateral force.

Figure 9. Bow pressures
Figure 11. Pressure contours 6 m

Figure 10. Pressure contours 5 m
Figure 12. Pressure contours 7 m

Figure 13. Pressure contours 8 m
Figure 14. Pressure contours 10 m

Figure 15. Pressure contours 12 m
Pressure contours 14 m
The geometry and the domain around the ship in ANSYS and also the chosen boundary conditions according to ship in open sea operation. Creating an accurate mesh involves a grid generation using appropriate shape cells; here the cell for the mesh is triangular. Essentially, it consists of converting the grid into a format which can be understood by the CFX solver in order to approximate the equations of the fluid mechanics in each cell.

4. Numerical results

Lateral force computed after the simulation is done, CFX give the following values about $S_1$ and $S_2$ coefficients: $S_1=5,6,7,8$ m and $S_2=1,2,4$ m. Presented results from simulation are showing less interference for higher $s_2$ distance for any $s_1$ value.
5. Conclusions

The main conclusion that can be made onto this stage of the study is that observing the result for lateral force shown in figure 18. For more precision it have to be made numerical experiments and it have to be included the technique mesh refinement on a HPC. The numerical simulation method for 2D viscous flow was formulated using ship interference with a square shaped object. We explicitly reduced number of design point in order to obtain a fast calculation procedure. Forces resulted from pressure variation due to interference with a square shaped object are analyzed and presented by computer fluid dynamics module ANSYS. Water speed $v=ct$ and current variable incidence angle are used for analysis. Interaction is computed on 2D model for distances between objects 0.2L…0.9 L, where L is the length of the ship. Various positions of the square shaped object will determine different case as presented in the fluid flow simulation in a restricted domain simulating open sea operation. The pressure for each case is presented in CFD-Post and interference is presented for ship operation in calm water.

References
[1] www.ansys.com
[2] Domnisoru L, Găvan E, Popovici O – Analiza structurilor navale prin metoda elementului finit, Editura Didactica si Pedagogica, Bucuresti 2005, ISBN 973 – 30 – 1075 – 8
[3] Wei Tong, ed. (2010). Wind Power Generation. WIT Press. ISBN 978-1-84564-205-1.
[4] Călimănescu I., Stan L. C., Computer fluid dynamics (CFD), Atom 2016, Conference Paper.
[5] Stan L. Călimănescu I., C.A New innovative, 2016, Conference Paper.
[6] http://web.mit.edu/xeviroca/www/rocaMeshForSim.pdf
[7] https://www.padtec.com/blog/wp-content/ANSYS-Meshing_Blog.pdf
[8] https://www.ozeninc.com/wp-content/uploads/2014/11/MESHING_WORKSHOP_2014.pdf