"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XV – 2012 – Issue 2
Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania

CALCULATION OF THE FLOW AROUND A PANAMAX TANKER

Marian RISTEA[1]
Ionel POPA[2]
[1]Lt J.G. (Navy), Lecturer, Eng, PhD, „Mircea cel Bătrân" Naval Academy, Constanta
[2]Commander senior lecturer dr. eng. ,Naval Academy "Mircea cel Batran"

Abstract: *The present study is devoted to the computation of a PANAMAX tanker in head wave with free heave and pitch motion. A RANS solver using finite-volume discretization and free-surface capturing approach is employed for the computation. The expected results refers to the drag force variation for a certain trim.*
Keywords: *tanker, hull, drag, force*

I. INTRODUCTION

As most cargo worldwide today is transported via ship, it is very important to design the ship hull forms such that they operate economically. To propel a ship, its engine has to provide enough power to overcome the hydrodynamic drag due to viscosity and wave generation. It is necessary to understand the complicated flow characteristics to design the hull forms with lower drag and higher propulsive efficiency. For better understanding of the flow around a modern commercial ship, it is of primary importance to produce reliable experiment data of practical hull forms.

The experiment data describing the local flow details are also invaluable in the field of computational fluid dynamics (CFD) for the validation of the developed physical and numerical modeling.

There have been some experimental data for the flows around ship models. The International Towing Tank Conference (hereafter, ITTC) summarized available benchmark database for CFD validation for resistance and propulsion of a ship (ITTC 1999; see also Longo and Stern 1996; Stern et al. 1998). For the cargo-container ship, Series 60 (Fry and Kim 1985; Toda et al. 1990, 1992; Longo et al. 1993; Suzuki et al. 1997) and Hamburg Test Case Bertram et al. 1994; Gietz and Kux 1995) are given. DTMB model 5415 is recommended for a combatant model Fry and Kim 1985; Ratcliffe 1998; Olivieri and Penna 1999; Longo and Stern 1999). For the full-form tanker, HSVA/Dyne tanker models (Knaack 1992; Denker et al. 1992; Dyne 1995) and Ryuko-Maru (Ogiwara 1994; Suzuki et al. 1998) are given.

Previously, two workshops (Larsson et al. 1991; Kodama 1994) were arranged for the computational analysis of flow around a ship, and HSVA/Dyne tanker models and a Series 60 model were chosen for the test cases. However, those data are often partial and not enough to understand the complicated flow phenomena. The hull forms used in those experiments are old-fashioned and quite different from the modern hull forms of ships today.

II. SOLVER

Computation has been performed with the ANSYS CFX solver available in the "Mircea cel Batran" Naval Academy. Turbulent flow is simulated by solving the incompressible Reynolds-averaged Navier-Stokes equations (RANS). The flow solver is based on finite volume method to build the spatial discretization of the transport equations. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure-equation. In the case of turbulent flows, additional transport equations for modeled variables are discretized and solved using the same principles. The gradients are computed with an approach based on Gauss's theorem. Non-orthogonal correction is applied to ensure a formal first order accuracy. Second order accurate result can be obtained on a nearly symmetric stencil. Inviscid flux is computed with a piecewise linear reconstruction associated with an upwinding stabilizing procedure which ensures a second order formal accuracy when flux limiter is not applied. Viscous flux are computed with a central difference scheme which guarantee a first order formal accuracy. We have to rely on mesh quality to obtain a second order discretization for the viscous term. Free-surface flow is simulated with a multi-phase flow approach. Incompressible and non-miscible flow phases are modeled through the use of conservation equations for each volume fraction of phase/fluid. Implicit scheme is applied for time discretization. Second order three-level time scheme is employed for time-accurate unsteady computation.

Velocity-pressure coupling is handled with a SIMPLE like approach. Ship free motion can be simulated with a 6 DOF module. Some degree of freedom can be fixed as well. An analytical weighting mesh deformation approach is employed when free-body motion is simulated. Several turbulence models ranging from one-equation model to Reynolds stress transport model are implemented in Ansys CFX. Most of the classical linear eddy-viscosity based closures like the Spalart-Allmaras one-equation model, the two-equation *k*-ω, SST model by Menter [5], for instance are implemented. Wall function is implemented for two-equation turbulence model.

III. DESCRIPTION OF TEST CASE

The test case chosen for the present study is based on an older simulation carried on a TRANSAS LCHS simulator, where for this type of vessel, were determined the loading conditions in several ballast situations. The present case is characterized by a stern draft of 7,56 meters and bow draft of 3,03 meters, as shown in the following table:

Table I. Parameters for the simulation

Determined value	Load case						
	I	II	III	IV	V	VI	VII
Medium draft [m]	2,96	3,29	4,31	4,77	5,29	6,19	6,6
Stern draft [m]	5,57	5,22	6,40	6,32	7,56	5,99	7,45
Displacement [t]	16500	19692	24875,3	27896,4	30958	37274,1	39963,8
Wet surface [m^2]	5748,44	5874,99	6025,06	6128,89	6163,35	6385,79	6443,36
Block coefficient	0,750	0,764	0,776	0,7871	0,7876	0,8105	0,815

For the considered case, we used several velocities of the current, starting from 10 knots and finalizing with 25 knots.
All calculations described in this paper were conducted for the unappended hull form.

IV. COMPUTATIONAL DOMAIN

The computational domain was defined for the full scale model, with boundaries at $4 \cdot L_{PP}$ at the stern and aside, and with the inlet boundary at $0.5 \cdot L_{PP}$.

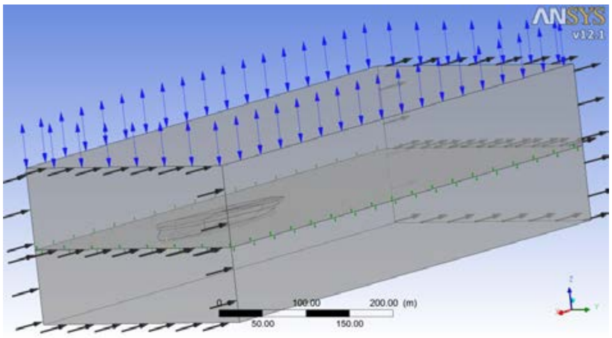

Figure 1. The computational domain

The results presented in this paper were all obtained on structured grids with H-O topology with some extra grid clustering close to the ship hull.

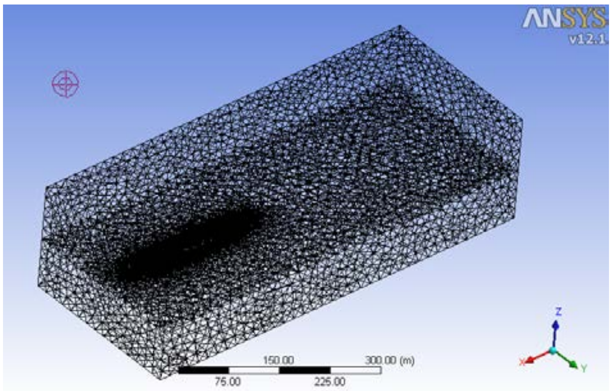

Figure 2. Mesh structure

Table II. Mesh information

Domain	Nodes	Elements
Air	38341	206321
Water	91131	499955
All Domains	129472	706276

At the ship surface the no-slip condition is applied directly and the normal pressure derivative is assumed to be zero. The undamped eddy viscosity, the variable in Menter’s one-equation model, vanishes at a no-slip wall. With the present formulation of the *k*−ω model (Kok and Spekreijse, 2000), all the turbulent quantities are zero at a solid wall.

IV. NUMERICAL CONVERGENCE

In the present calculations we have adopted as convergence criterion the reduction of the maximum difference between consecutive iterations of the three velocity components and of the pressure to 10^{-4}.

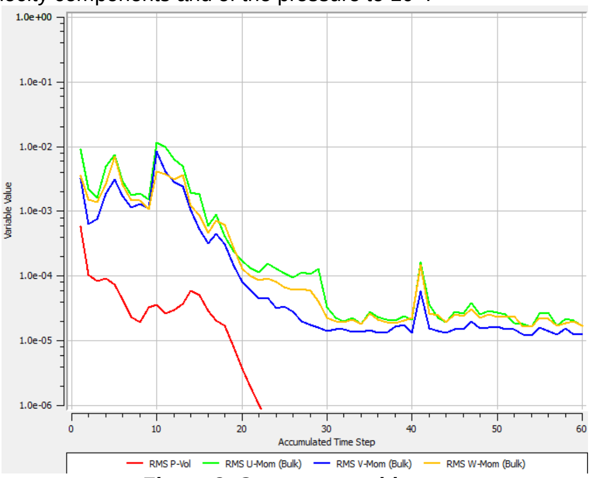

Figure 3. Convergence history

V. RESULTS

After performing the calculations, there was determined the profiles for pressure and force along the Ox axis and also the velocity profile on the waterplane.

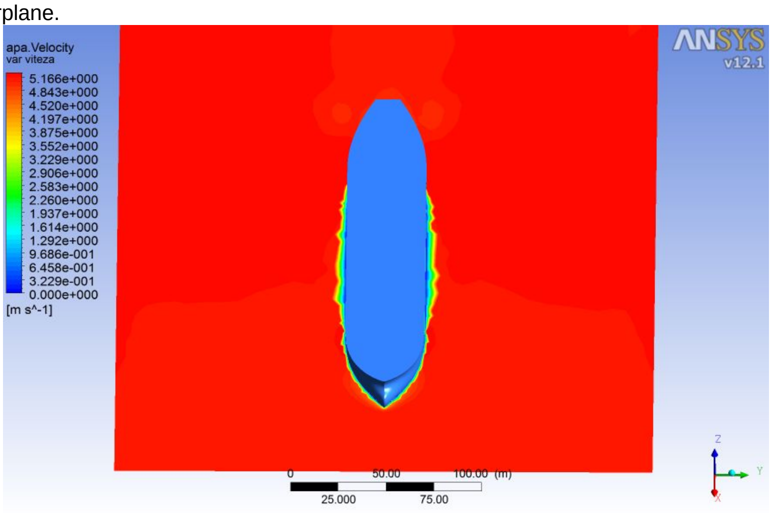

Figure 4. Velocity variation on the free surface plane

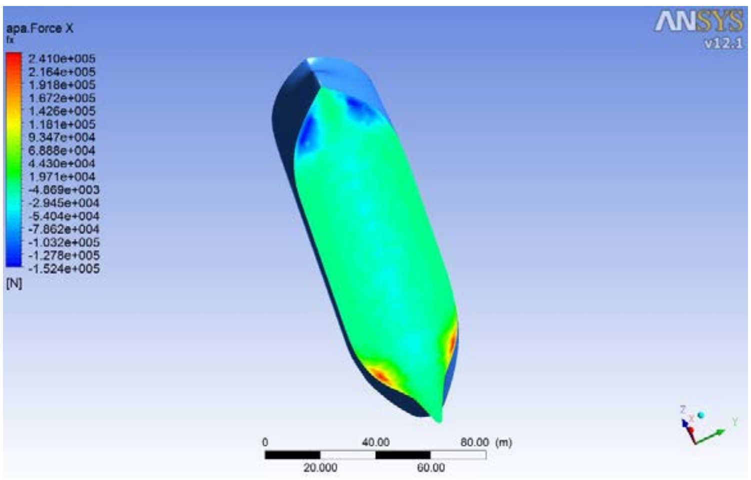

Figure 5. The variation of the drag force along the ship's hull

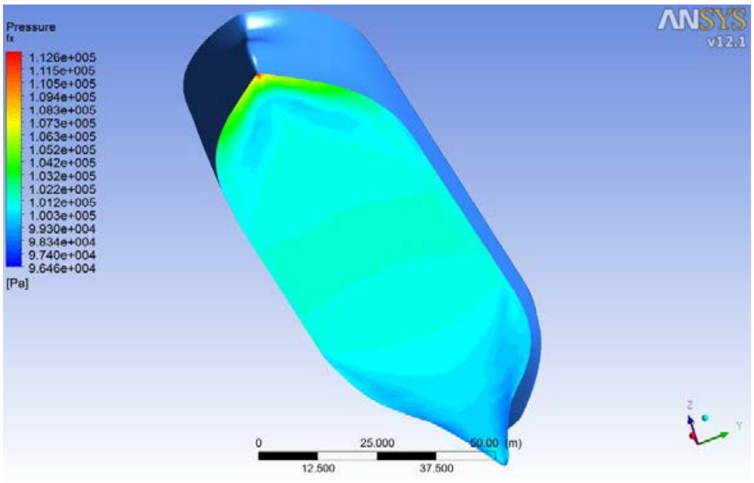

Figure 6. Pressure variation on the ship's hull

VI. CONCLUZIONS

In this paper we focus on the value of the drag force at a certain value of the trim angle and on the variation of pressure along the ship's hull. It can be noticed that the maximum values for the force on Ox axis are established in an unusual location, due to the fact that the draft from astern is higher than the one from the bow.

The determined values for the drag force are presented in the next table:

Table III. The drag force

Speed [knots]	Drag Force (-Fx) [KN]
10	**396**
11	**486**
12	**570**
13	**668**
14	**781**
15	**890**
20	**1579**
25	**2480**

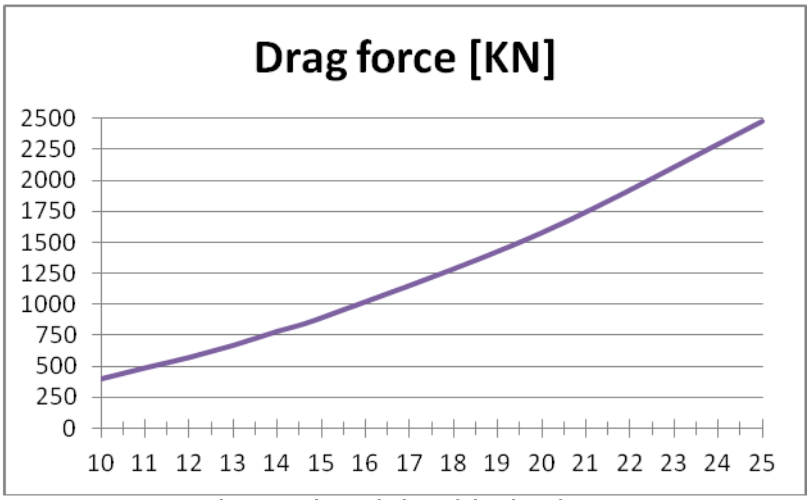

Figure 7. The variation of the drag force

VII. REFERENCES

1. Dacles-Mariani J., Zilliac G.G., Chow J.S., Bradshaw P., ”Numerical/experimental study of a wing tip vortex in the near field”, AIAA Journal, Vol. 33, September 1995, pp. 1561-1568.

2. Eça L., Hoekstra M.,Windt J., ”Practical Grid Generation Tools with Applications to Ship Hydrodynamics”, 8*th* International Conference in Grid Generation in Computational Field Simulations, June 2002, Hawaii, USA.

3. Eça L., Hoekstra M., ”An Evaluation of Verification Procedures for CFD Applications”, 24*th* Symposium on naval Hydrodynamics, July 2002, Fukuoka, Japan.

4. Eça L., Hoekstra M., ”A Verification Exercise for Two 2-D Steady Incompressible Turbulent Flows”, 4*th* *E*uropean Congress on Computational Methods In Applied Sciences And Engineering, ECCOMAS 2004, July 2004, Finland.

5. Eça L., Hoekstra M., ”On the influence of grid topology on the accuracy of ship viscous flow calculations”, 5*th* Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, 2005, Osaka, Japan.

6. Kok J.C., ”Resolving the Dependence on Free-stream values for the k – ω Turbulence Model”, NLR-TP-99295, July 1999, National Aerospace Laboratory, NLR, The Netherlands.

7. Kok J.C, Spekreijse S.P., ”Efficient and Accurate Implementation of the k – ω Turbulence Model in the NLR multi-block Navier-Stokes system”, NLR-TP-2000-144, May 2000, National Aerospace Laboratory, NLR, The Netherlands.

8. Menter F.R., ”Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications”, AIAA Journal, Vol.32, August 1994, pp. 1598-1605. Menter F.R., ”Eddy Viscosity Transport Equations and Their Relation to the k− ω Model”, Journal of Fluids Engineering, Vol. 119, December 1997, pp. 876-884.

9. Roache P.J., ”Verification and Validation in Computational Science and Engineering”, Hermosa Publishers,1998.

10. Toxopeus S.L. ”Validation of calculations of the viscous flow around a ship in oblique motion”, The First MARIN-NMRI Workshop, October 2004, pp. 91–99.

11. Calculation of the Flow around the KVLCC2M Tanker