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

HYDRODYNAMIC ANALYSIS REGARDING THE BALLAST-FREE CONCEPT

Vergil CHIȚAC[1]
Mihail PRICOP[2]
Valentin ONCICA[3]
Ionuț-Cristian SCURTU[4]
[1] Captain Professor Engineer, PhD, Naval Academy „Mircea cel Batran"
[2] Captain Professor Engineer, PhD, Naval Academy „Mircea cel Batran"
[3] Commander Lecturer Engineer, PhD, Naval Academy „Mircea cel Batran"
[4] Teacher Assistant, PhD candidate, Naval Academy „Mircea cel Batran"

***Abstract**: The ballast free concept is a real future possibility in ship design and environmental protection. Measurements from other sources reveal a speed decreasing with an amazing percent. Using new software from The Department of Naval Architecture, Shipping and Port Management this paper will simulate in Ansys FLUENT (CFD component)and AutoPower a classic ship versus the new ballast free concept. this paper, a part of the results and conclusions are presented.*
***Keywords**: Hydrodynamic, ballast, ship.*

1. INTRODUCTION

All ships carry ballast to stay stable when cargo is not present, and ballast is a great threat to environment. The ballast free concept will let the fresh or saltwater to flow through the ship hull through a set of pipes with low center of gravity in order to lower the whole ship center of gravity. This will be possible when the set of pipes will be filled with water so this type of ship will not carry ballast water from one port to another.

This concept is based on US patent #6,694,908/ 2004 and it was especially created to minimize the impact of the nonindigenous aquatic species transported by the ships arriving in the ballast condition to the environment. The first investigation of this new concept was in 2004 at the University of Michigan and the single purpose was to minimize risks from ballast water. In 2006 [1] was built the first scale model and in the same year they run the first model testing in towing tank experiments, the main purpose of this experiment was to optimize the location of the ballast-free trunk discharge.

Tests on the scale model in the towing tank and computer simulations indicate that this design reduces the power needed to propel the ship by 7.3%. We understand that this percent is very important to the future of shipping and we will analyze the concept regarding the reduction of the power needed to propel this model versus a classic shape ship.

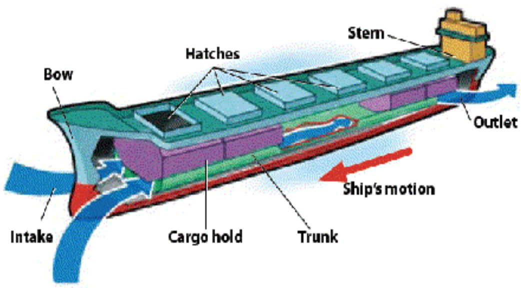

Fig.1. Ballast Free Concept [Source: 6]

2. SIMULATION INVESTIGATION

The Department of Naval Architecture, Shipping and Port Management has new licensed software like Ansys and Autoship to develop and simulate different projects and also for student use. The simulation investigation will use both mentioned software.

2.1. Ansys Investigation

The software used in this analysis is Ansys 13.0 FLUENT. Ansys FLUENT is a CFD software that contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems have served to broaden its reach. The main component used in this paper is ANSYS Fluent in which we analyzed a box shape 100X20X7m object moving through fluid with a constant speed(6m/s) and another similar box shape with a central hole with a variable diameter(0.5, 1, 1.5, and 2 m).

In the following experimental investigation over the ballast free concept. we will use ANSYS 13 FLUENT to develop a simple model with a box shape for a better and fast solution with no other interference. The main purpose of this study is to test this solution from the economically point of view. A ship propulsion needs more chemical energy from heavy fuel oil or from diesel fuel oil if the ship total resistance is higher, that is why a small difference in the variation of the total resistance will cause an increasing or decreasing of operating costs. The Ansys simulation is based on the transient dynamic equilibrium equation of interest, as follows for a linear structure:

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F^a\} \quad (1)$$

where: [M] = structural mass matrix, [C] = structural damping matrix, [K] = structural stiffness matrix, $\{u_{n+1}\}$= nodal acceleration vector, $\{u_n\}$ = nodal velocity vector, {u} = nodal displacement vector, {Fa} = applied load vector

There are two methods in the ANSYS program which can be employed for the solution of Equation 1: the central difference time integration method and the Newmark time integration method (including an improved algorithm called HHT). The central difference method is used for explicit transient analyses. The Newmark method and HHT method are used for implicit transient analyses.

The Newmark method uses finite difference expansions in the time interval Δt, in which it is assumed that

$$\{\dot{u}_{n+1}\} = \{\dot{u}_n\} + [(1-\delta)\{\ddot{u}_{n+1}\}]\Delta t \quad (2)$$

$$\{u_{n+1}\} = \{u_n\} + \{\dot{u}_n\}\Delta t + \left[\left(\frac{1}{2}-\alpha\right)\{\ddot{u}_n\} + \alpha\{\ddot{u}_{n+1}\}\right]\Delta t^2 \quad (3)$$

where: α, δ = Newmark integration parameters; Δt = t_{n+1} – tn; {un} = nodal displacement vector at time t_n; { n} = nodal velocity vector at time t_n; { n} = nodal acceleration vector at time t_n; {un + 1} = nodal displacement vector at time t_{n+1}; { n + 1} = nodal velocity vector at time t_{n+1}; { n + 1} = nodal acceleration vector at time t_{n+1}.

The solution for the displacement at time tn + 1 is obtained by first rearranging equation 2 and equation 3 in Fluent solver.

Fluent requiers a domain for fluid flow around the created geometry with minimal specifications in order to minimize interference from the walls. This step is shown in fig. 2.

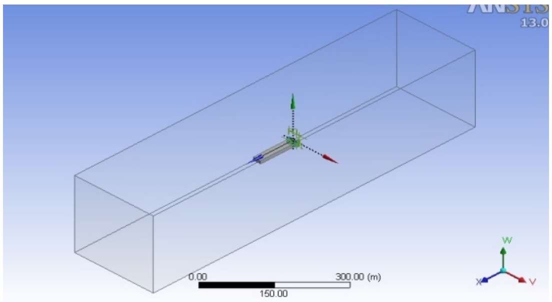

Fig.2. Building the geometry and the model enclosure

The process for generating a mesh of nodes and elements consists of three general steps:
1. Set the element attributes.
2. Set mesh controls (optional). ANSYS offers a large number of mesh controls from which you can choose as needs dictate.
3. Meshing the model.

The following mesh-generation topics are available:

1. Free or Mapped Mesh 2. Setting Element Attributes 3. Mesh Controls 4. Controls Used for Free and Mapped Meshing 5. Meshing Your Solid Model 6. Changing the Mesh 7. Meshing Hints 8. Using CPCYC and MSHCOPY Commands

We used the free or mapped mesh. Before meshing the model, and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it. A mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this type of mesh, you must build the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh.

Both domain and geometry will pass through meshing step where smaller cells will be automatically created by FLUENT software if the flow speed variation is higher . This step is shown in fig. 3.

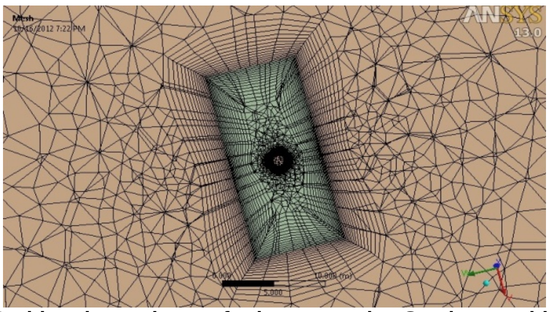

Fig.3. Meshing the enclosure for better results. Section meshing view

Resistance results from Ansys fluent simulations using the K-Omega turbulence model are shown fig. 9-10 and in the table 1 in results section.

2.2. Autoship Investigation

Autoship is a full package software designed especially for ship building industry for a fast analysis. This software includes Model Maker which allows users to develop new tridimensional ship architecture, also Autopower package which is used for total resistance calculus.

The first step of the investigation was to find a ship with known dimensions and total resistance, we draw the tridimensional ship and we tested Autopower capabilities, fig.4-5.

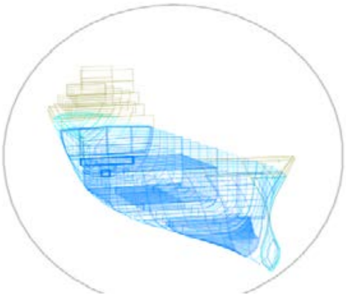

Fig.4. Building test model for known ship

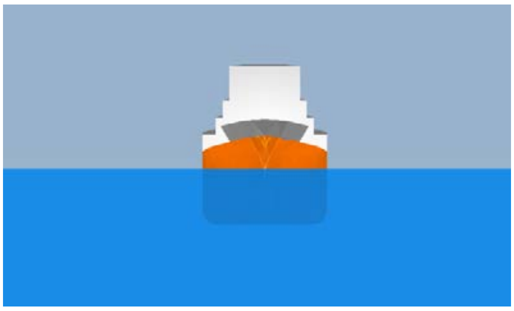

Fig.5. Test model for known ship

The desired design for the ballast free ship is realized in the figure no.6 and the analysis from the Autohydro package revealed the new hydrostatic curves, which are not part of our aim.

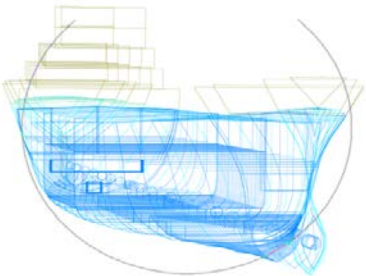

Fig. 6. Ballast free ship design

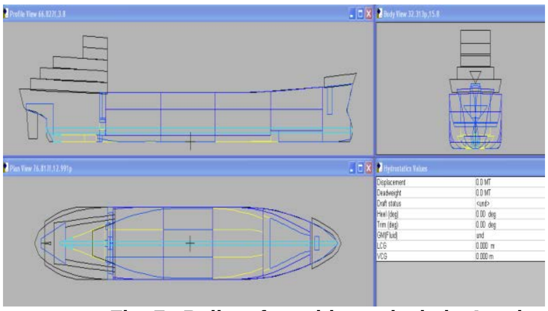

Fig. 7. Ballast free ship analysis in Autohydro

After importing the geometry in the Autopower software it was possible to determine new values for the ballast free ship.

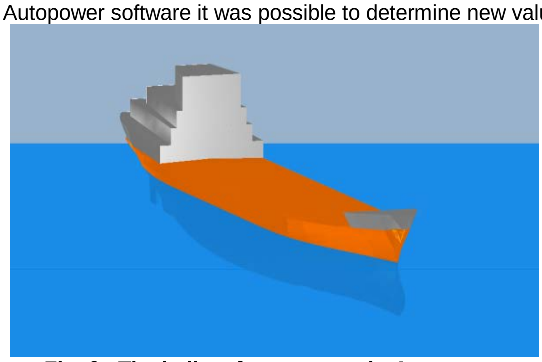

Fig. 8. The ballast free concept in Autopower

3. RESULTS

The results from the Fluent analysis show us a maximum decrease of the total resistance with 1,78% for a 2 m diameter opening on the first analyzed object. Ansys velocity distribution on XOZ plane and pressure distribution on XOZ plane are shown below.

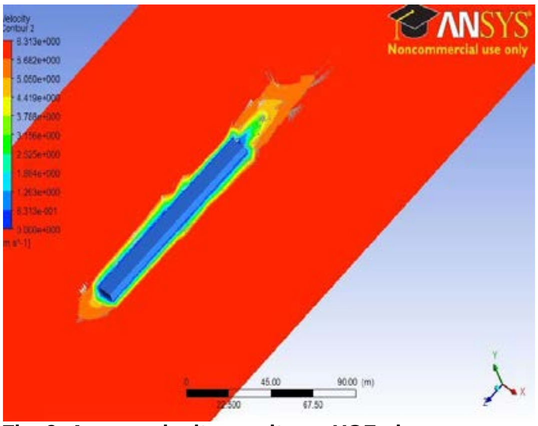

Fig. 9. Ansys velocity results on XOZ plane

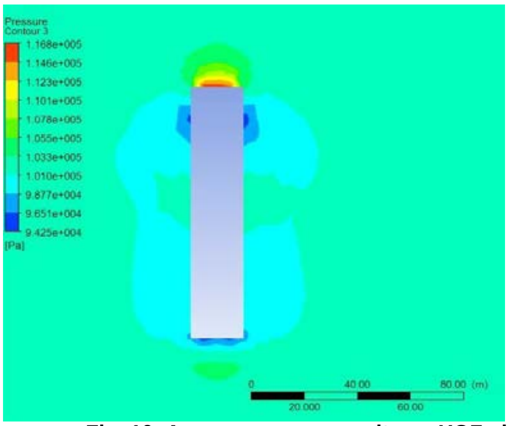

Fig. 10. Ansys pressure results on XOZ plane

.**Table. 1**. Ansys Fluent results for 6 m/s

Object	Pressure Resistance [KN]	Viscous Resistance [KN]	Total Resistance [KN]	Decrease percent Decreasing percent [%]
Full box	2398.342	7.032	2405.374	-
Box with 0.5 m diameter hole	2395.788	6.891	2402.679	0.12
Box with 1 m diameter hole	2387.927	6.752	2394.679	0.45
Box with 1.5 m diameter hole	2374.869	6.622	2381.491	0.99
Box with 2 m diameter hole	2356.482	6.487	2362.969	1,78

Ansys results regarding speed from 4m/s to 10m/s are shown in graphics below

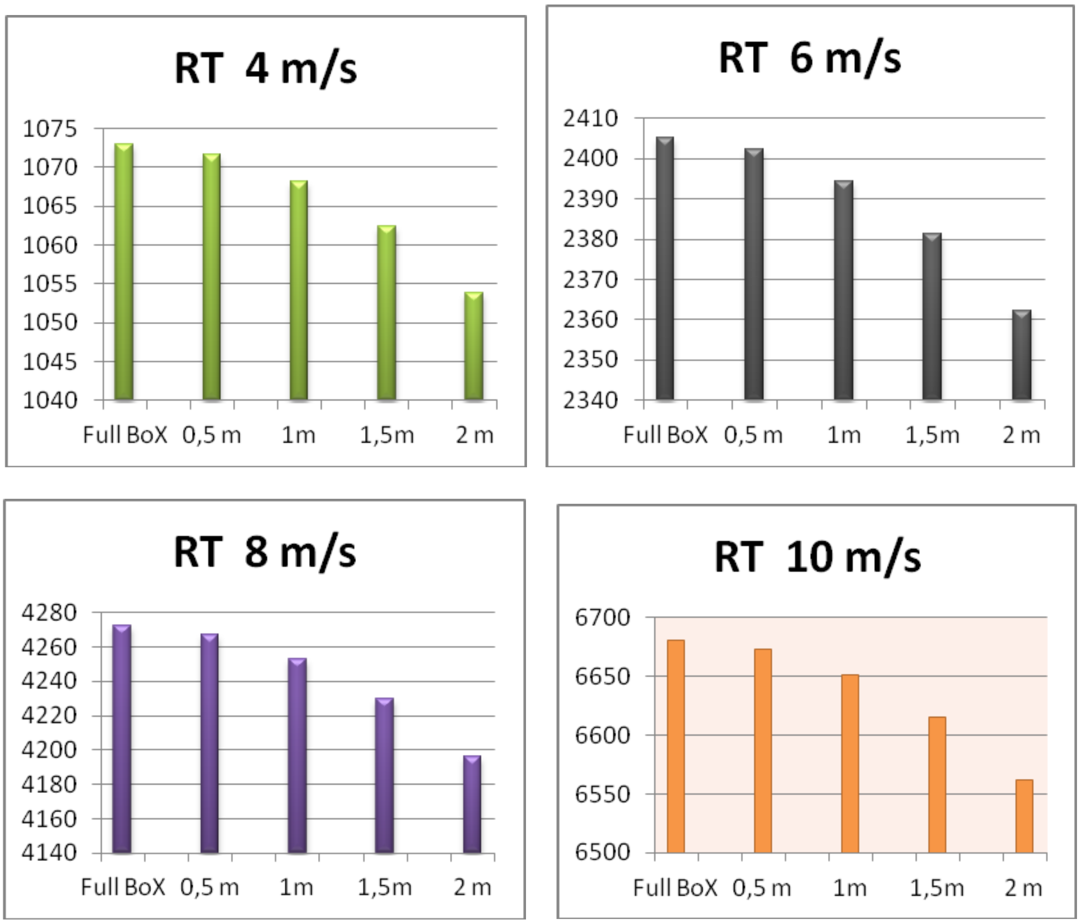

Fig .11. Ansys compared results for different speeds

AUTOSHIP investigation revealed a dropping with 3% of the total resistance value. The graphic below is trying to plot the small difference.

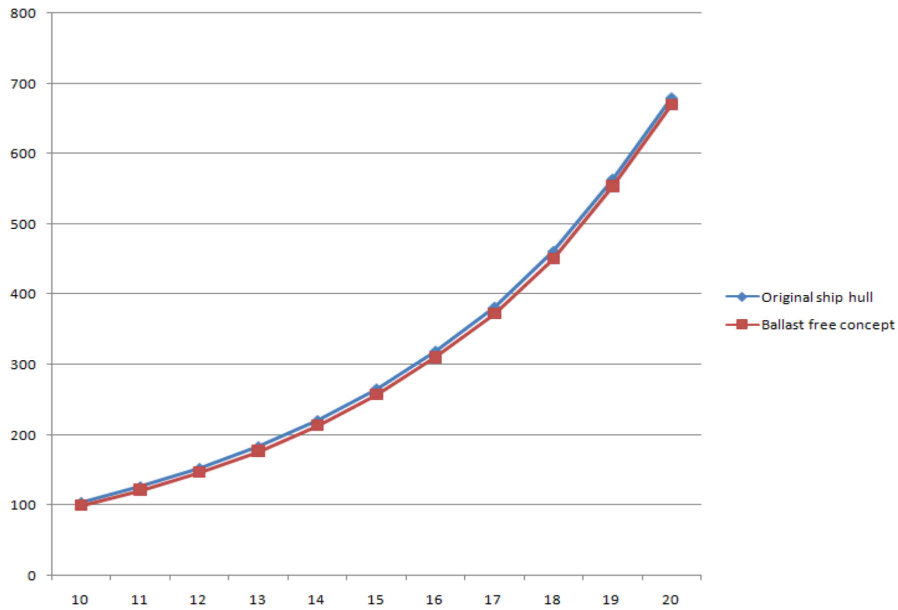

Fig.12. Resistance versus speed for original shape(blue) and for the studied concept(red)

Table 2. Results from Autopower

Original ship hull Total Resistance, Rt (kN)				Ballast free concept Total Resistance, Rt (kN)		
Speed (kt)	Fn	Holtrop		Speed (kt)	Fn	Holtrop
10,00	0,15	103,93		10,00	0,15	99,70
11,00	0,16	126,28		11,00	0,16	121,24
12,00	0,17	152,37		12,00	0,17	146,50
13,00	0,19	183,30		13,00	0,19	176,60
14,00	0,20	220,39		14,00	0,20	212,87
15,00	0,22	265,34		15,00	0,22	257,08
16,00	0,23	318,88		16,00	0,23	309,95
17,00	0,25	381,39		17,00	0,25	371,87
18,00	0,26	460,65		18,00	0,26	450,80
19,00	0,28	563,42		19,00	0,28	553,65
20,00	0,29	678,87		20,00	0,29	669,35

4. CONCLUSION

This concept was not designed for lowering the costs of ships running in ballast condition, the main purpose was to minimize the ecological impact of translocated species, and the results obtained confirm this theory. A closer look at the impact of the openings over the ship's fuel economy was needed in order to show the differences and the possibilities of this concept.

The ship motion will cause the fluid flow through the ship's system of pipes that is why the pressure on the bow will become smaller and will cause a maximum 3% total resistance reduction according to this paper that is a smaller percent then the initial from other sources.

5. REFERENCES

1. IMO 2004 “International Convention for the Control and Management of Ships' Ballast Water & Sediments,” Diplomatic Conference, February, London.
2. Kotinis, M. and Parsons, M. G., “Hydrodynamic Optimization of the Ballast-Free Ship Concept”
3. Mathur, S.R. and Murthy, J.Y. 1997 “A Pressure-Based Method for Unstructured Meshes,” Numerical Heat Transfer, 31, pp. 195-215.
4. Parsons, M. G., “Ballast-free ships: Investigations at the University of Michigan use local water for ballast,” Great Lakes Seaway Review, 35-2: 19 & 21, January-March, 2007.
5. Ansys website www.ansys.com
6. http://machinedesign.com/article/scanning-for-ideas-ballast-free-ship-would-keep-lakes-and-waterways-clean-0619