IJEAST

INTERNATIONAL JOURNAL OF ENGINEERING APPLIED SCIENCE AND TECHNOLOGY

VOLUME : 5 ISSUE : 12 Print / Issue Publication Date: 13-Jul-2021

ISSN : 2455-2143

DOI : 10.33564/IJEAST.2021.v05i12.003

Indexed In

Google SCHOLAR OPEN ACCESS doi Crossref

WWW.IJEAST.COM

editor@ijeast.com

International Journal of Engineering Applied Sciences and Technology, 2021
Vol. 5, Issue 12, ISSN No. 2455-2143, Pages 23-26
Published Online April 2021 in IJEAST (http://www.ijeast.com)

A COMPARISON STUDY ON THE RESISTANCE CALCULATION OF PLANNING BOAT

Do Quang Khai
Department of Shipbuilding
Vietnam Maritime University, Hai Phong. Viet Nam

Nguyen Thi Thu Quynh
Department of Shipbuilding
Vietnam Maritime University, Hai Phong, Viet Nam

Nguyen Manh Chien
Department of Shipbuilding
Vietnam Maritime University, Hai Phong, Viet Nam

***Abstract*— Calculation of the resistance and trim of high-speed boat at planning regime is a complex hydrodynamic problem. At the planning regime, with the aid of lifting force, the boat is lifted over the water surface with high trim angle. Many methods have been applied to calculate the resistance and the trim of the high-speed boat at this planning regime. This paper will study the two typical methods, the first one is empirical method by Savitsky [1] and the second one is the computational fluid dynamic method (CFD). The calculation results of two methods are compared with experimental results.**

Keywords— **High speed boat, CFD, Savitsky, planning regime, resistance**

I. INTRODUCTION

The high-speed boat has 3 operating regimes, characterized by the Froude (Fr_Δ) number as follows:

$$Fr_\Delta = \frac{V}{\sqrt{g\sqrt[3]{\Delta g/\gamma}}}$$

Displacement regime: $Fr_\Delta \leq 1$
Semi-displacement regime: $1 \leq Fr_\Delta \leq 3$
Planning regime: $Fr_\Delta \geq 3$

Where: V is volume displacement of the vessel (m^3)
Δ: weight displacement (kg)
γ: water density (N/m^3)
g: acceleration of gravity $g = 9.81\ m/s^2$

At the displacement regime, with $Fr_\Delta \leq 1$, the planning boats operate same as the displacement vessels, and the lifting force is small and the boats cannot be lifted over the water surface. As the speed of the boat increases, with the $Fr_\Delta \geq 1$, the boat operates at the semi-displacement regime. At this regime, the boat has high resistance and high trim angle. When $Fr_\Delta \geq 1$, the boat operates fully at planning regime. At planning regime, the boat operates more efficiently and needs less power to maintain the speed because the lifting force lifts the boat above the water surface, thus, the friction resistance decreases. However, at this state, it is more difficult to calculate the flow around the planning boat than the displacement vessel due to the contribution of hydrodynamic lift and large trim angle. There are few methods available to estimate the resistance of planning boat. They are emprical methods such as method by Savitsky [1], the numerical method by using high power computer (or computational fluid dynamic – CFD) and using experimental method by testing in towing tank. In this paper, the authors will using the first two methods: the emperical and the CFD method to calculate the resistance of the high speed boat at planning regime. The results will be compared with experimental one to assess the accuracy of each methods and give some guidelines for the designers in the selection of suitable method to calculate the resistance of planning boat.

II. CALCULATION METHODS

A. The Savitsky method

The Savitsky method [1] has been applied widely since 1964 to predict the power requirement for planning boat. This method requires the prismatic hull shape, so it will give more accurate result if the hull sections are more like the triangle. The advantage of this method is that it is very fast, the users can implement this method in Excel or even hand calculation. Moreover, it requires limited input data, only some basic dimensions of the planning boat, such as: speed, mass, deadrise angle, center of gravity. The outputs of the method are the total resistante, lift force, trim and sinkage and wetted surface area. Futher details of this method can be read in the Reference 1 [1]

B. The CFD method

Recently, with the high development of computational power, the computational fluid dynamic (CFD) method has been applied widely to solve the hydodynamic problems. Nowsaday, using CFD to calculate ship resistance has been the typical method in ship design office. However, the CFD method requires that the users have solid knowledge about hydrodynamics as well as solid experiments on using CFD software. Because the minor mistake can give totally wrong result. For the ship hydrodynamics problems, the Reynold Average Navier Stokes Equation (RANSE) CFD method is widely used among the other methods that require more computational resources such as LES or DNS.

Ship resistance calculation for the displacement vessel can be solved quite accurately by RANSE CFD method, with the differences less than 2% comparing with experimental result [2] [3]. However, to calculate the resistance for planning boat, it is much more difficult due to high trim angle, leading to the high deforming mesh, and negative cells; the solver may be crashed during calculation. Thus, in this paper, the authors will present the method to dealing with this problem of high deforming mesh. It is the combination between Savitsky method and CFD method. The solver that the authors use in this calculation is Numeca Fine Marine. The boat model is model C1 in Naples Series [4] developed by Università degli Studi di Napoli Federico II. The experimental result of resistance calculation is also available and is provided by Università degli Studi di Napoli Federico II

III. CALCULATION STEPS

As mentioned above, the boat model that authors select to calculate is model C1. The basic dimensions of the boat are present in Table 1. The 3D shape and boat section of this model are shown in Figure 1 and 2

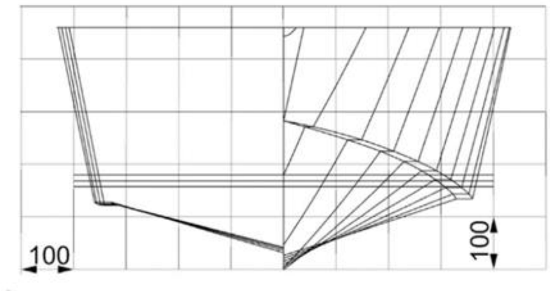

Fig. 1. Sections of the Naples C1 hull

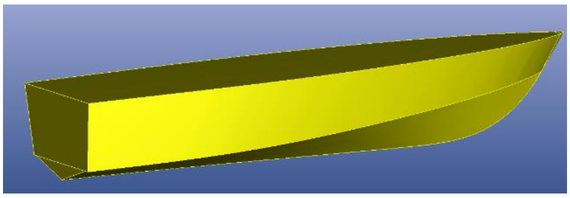

Fig. 2. 3D visualization of Naples C1 hull

Table -1 Basic dimension of Naples C1 hull

	Value	Unit
Water line length L_{WL}	2.4	m
Breadth B_{WL}	0.743	m
Depth D	0.46	m
Draft d	0.1677	m
Wetted surface area Sw	1.7	m^2
Longitudinal center of gravity	0.943	m
Vertical center of gravity	0.193	m
Speed	7	m/s
Fr_{Δ}	3.24	

For the Savitsky method, we just need to input some basic parameters of the boat such as ship speed, ship weight, center of gravity, water length. Then we will have the result of ship resistance after few second

Besides, the CFD method requires much more efforts including: meshing, setup calculation parameters, performing calculation then post-processing the result. First step for the CFD is creating the domain around the boat then meshing this domain. The dimension of the fluid domain around the boat is presented in the Figure 3 below

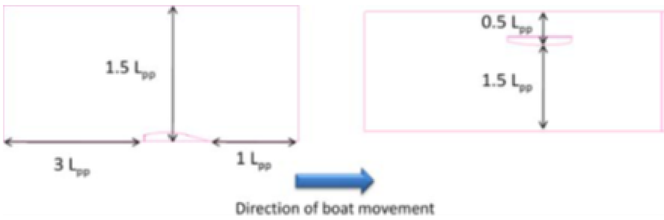

Fig. 3. Dimension of the fluid domain [3]

Theoretically, the larger fluid domain, the better calculation result. However, increasing fluid domain too large will lead to large number of cells for meshing and high calculation time. So, the fluid domain size as Figure 3 is sufficient, based on the recommendation from Numeca. The number of cells is around 2.2 million cells. Some special areas are refined with much more cells such as the bottom, the chines. The mesh of the boat is shown in Figure 4. Figure 5 presents the mesh at free surface.

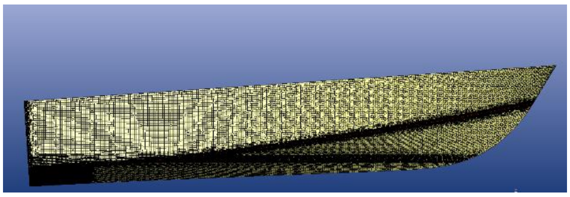

Fig. 4. Mesh of the boat

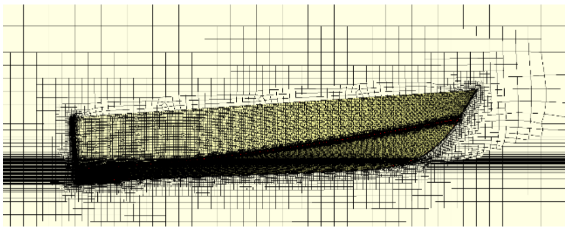

Fig. 5. Mesh of the free surface

The viscous layers for solving the boundary layer have 13 layers for the boat hull with the thickness of 0.242 mm, to ensure the Y+ around 30. The wall function is applied to solve the boundary layer. The calculation parameters to perform CFD calculation are as follows:

- For the water phase, the dynamic viscosity ν=0.001138 Pa.s ; water density : 999.1026 kg/m^3.
- For the air phase, the dynamic viscosity ν=1.85 ×10^$^{-5}$ Pa.s; air density: 1.2 kg/m^3
- Turbulence model: k-omega (SST-Menter),
- Boundary condition: wall function for the boat hull, non-slip condition; for the deck: wall function with slip condition. The boundary conditions at the boundary of the domain are as follows: zmax, zmin: prescribed pressure, updated hydrostatic pressure; xmin, xmax, ymax: far field; ymin (at the center line of the hull): mirror boundary condition.
- The parameters for the boat motion are:
  - Velocity = 7m/s
  - Pitch angle: Ry1 = -0.067380 Rad
  - Solving for trim and sinkage, other degrees of freedom are kept fixed
  - Hull center of gravity (0.943,0,0.29273)
  - Hull mass: 106.7 kg
- The parameters for the solver are
  - Time step Δt=0.005 L_WL/V=0.00186s
  - Number of time step: 2000

The CFD calculation is performed in parallel on 8 cores and it takes around 8 hours to get converged result. The convergence of ship resistance and trim calculated by CFD method is shown in Figure 6 and Figure 7 below. We can see that the result converges after around 4 second (simulation time not calculation time).

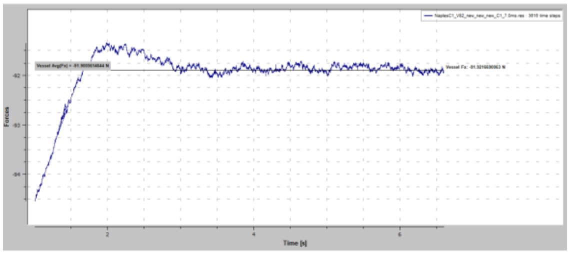

Fig. 6. Convergence of ship resistance by CFD method

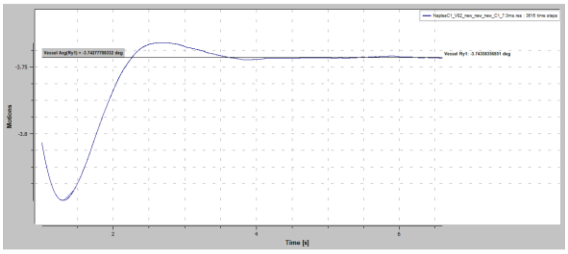

Fig. 7. Convergence of trim by CFD method

IV. RESULT AND DISCUSSION

The result of resistance and trim calculation by CFD method and Savitsky method in comparison with experimental result is presented in Table 2

Table -2 Calculation result

	CFD		Experiment
	Value	Diff	Value
Resistance (N)	183.8	**-2.10%**	187.75
Trim (degree)	3.842	**-10.65%**	4.3
	Savitsky		
	Value	Diff	
Resistance (N)	174.19	**7.22%**	
Trim (degree)	4.29	**0.23%**	

We can see that the CFD method gives us quite accurate resistance result with just 2% differences from experiment, but the trim angle is under estimation with 10% difference from experiment. The Savitsky method shows better estimation for trim angle (just 0.23% difference from experiment), but the ship resistance is less than 7% with experiment. Considering the complexity of the CFD method, the Savitsky method is better at the beginning of the ship resistance calculation process due to fast calculation time and simple data input. However, the CFD method is able to look at the flow around the hull in details, and is applicable for many types of hull. The Savitsky method is only valid with the boat with triangle cross section. So if the cross section of planning boat is much more different from triangle, Savitsky method will give not reliable result. Figure 8 shows the free surface elevation by CFD

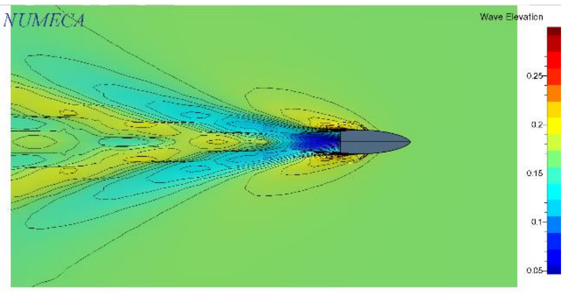

Fig. 8. Free surface elevation by CFD method

V. CONCLUSION

The paper has presented the result of calculation of ship resistance and trim of planning boat by two common method at the moment: CFD method and Savitsky empirical method. The Savitsky method is quick and the level of accuracy is acceptable and it requires that the cross section of the hull is close to a triangle to give reliable result. Besides, the CFD method is capable giving accurate resistance, but the cost is time consumption in meshing, calculation and post processing. So that the selection of the method is depending on the designers. The authors recommend that one should use Savitsky in the early design stage and CFD in more detailed design stage.

VI. ACKNOWLEDGEMENTS

The authors show the gratitude to the Vietnam Maritime University for their support to finish this paper.

VII. REFERENCE

[1] De Luca, F.; Pensa (2017), C. The Naples warped hard chine hulls systematic series. Ocean Eng, 139, 205–236.

[2] Daniel Savitsky (1964), "Hydrodynamic Design of Planing Hulls," Marine Technology, SNAME, Paramus, NJ..

[3] Larsson, L. and H. Raven (2010), Ship resistance and flow, Society of Naval Architects and Marine Engineers.

[4] Bertram, V.,(2011) Practical ship hydrodynamics, Elsevier.

[5] Artjushkov, L., (1968) Wall effect correction for shallow water model tests. NE Coast Institution of Engineers and Shipbuilders.

[6] Geerts, S., Verwerft, B., Vantorre, M., and Van Rompuy, F., (2010) Improving the efficiency of small inland vessels. Proc., 7th European Inland Waterway Navigation Conf., Budapest Univ. of Technology and Economics, Budapest, Hungary.,

[7] Prakash, S. and B. Chandra, (2013) Numerical estimation of shallow water resistance of a river-sea ship using CFD. International journal of computer applications, **71**(5).

[8] Pacuraru, F. and L. Domnisoru (2017). Numerical investigation of shallow water effect on a barge ship resistance. in IOP Conference Series: Materials Science and Engineering. IOP Publishing.

[9] Patel, P.K. and M. Premchand (2015), Numerical investigation of the influence of water depth on ship resistance. International Journal of Computer Applications, **116**(17).

[10] Tezdogan, T., A. Incecik, and O. Turan (2016), A numerical investigation of the squat and resistance of ships advancing through a canal using CFD. Journal of Marine Science and Technology, **21**(1): p. 86-101.

[11] Molland, A.F., S.R. Turnock, and D.A. Hudson (2017), Ship resistance and propulsion. : Cambridge university press.

[12] Karpov (1946), A., Calculation of ship resistance in restricted waters. TRUDY GII. T. IV, Vol. 2 (in Russian).

[13] Perali, P., T. Lloyd, and G. Vaz (2016). Comparison of uRANS and BEM-BEM for propeller pressure pulse prediction: E779A propeller in a cavitation tunnel. in Proceedings of the 19th Numerical Towing Tank Symposium

[14] Brizzolara, S., D. Villa, and S. Gaggero (2008). A systematic comparison between RANS and panel methods for propeller analysis. in Proc. Of 8th International Conference on Hydrodynamics, Nantes, France

[15] Tu, Tran Ngoc; Chien, Nguyen Manh (2018), Comparison Of Different Approaches For Calculation Of Propeller Open Water Characteristic Using RANSE Method, Naval Engineers Journal, Volume 130, Number 1, pp. 105-111(7)

IJEAST

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE
AND TECHNOLOGY

ABOUT IJEAST

International Journal of Engineering Applied Science and Technology (IJEAST) is a peer-reviewed, open access journal that publishes high-quality research papers in the field of Engineering, Applied Science and Technology.

IJEAST aims to provide a platform for researchers, academicians, and professionals to share their innovative ideas, research findings, and practical experiences with the global scientific community.

FOCUS AREAS

- Engineering
- Applied Science
- Technology
- Innovation & Development
- Interdisciplinary Studies

PEER REVIEWED
All submissions are rigorously peer reviewed to ensure quality.

OPEN ACCESS
Free and unrestricted access to research for all.

GLOBAL REACH
Connecting researchers and professionals worldwide.

TIMELY PUBLICATION
We ensure a swift and efficient publication process.

For more information, visit our website
www.ijeast.com

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE
AND TECHNOLOGY

editor@ijeast.com
www.ijeast.com
India

2455-2143