Resistance Analysis of Rescue Boat in Calm Water Condition

I K A P Utama¹, Sutiyo¹, I K Suastika¹, A Sulisetyono¹, Hasanudin¹, Y A Hermawan¹, and W D Aryawan¹
¹Department of Naval Architecture, Institut Teknologi Sepuluh Nopember, Indonesia

Abstract. Search and rescue operations, especially in ship accidents, require speed and reliability from the rescue vessels in facing the challenges of extreme weather and sea waves. A rescue boat hydrodynamic performance study is very important to be carried out in order to determine the ship's ability to withstand dangerous conditions while in operation. A combination of using Maxsurf and CFD method are implemented to estimate the resistance of 3 (three) selected designs of crew boat, which demonstrated a good agreement between the two techniques. Selection of the final design also uses seakeeping criteria, i.e. having lower pitching and rolling responses. The final model is further tested in a towing tank belongs to the Institut Teknologi Sepuluh Nopember (ITS) and agrees well with Maxsurf and CFD calculation.

1. Introduction

Although the number of investigations has dramatically increased, there is an overwhelming need to study the design of an efficient high-speed craft. In recent years, motorboats, pilot boats, personnel carriers, search and rescue (SAR) boats and some passenger boats have all been considered for attaining high speeds. Accelerating to a high speed is directly related with resistance characteristics and hull form[1]. At high speeds, the hull is subjected to dynamic effects, because of the hydrodynamic characteristics of the geometry, as well as the heave and pitch motions, termed trim and sinkage. At high speeds, according to the dynamic conditions of the rigid body, the total resistance increases rapidly and leads to excessive fuel consumption[2]. Generally, with Froude numbers between 0.4 and 1.58, a typical high-speed craft enters the transient (hump) regime. As the vessel advances through higher speeds, the wave resistance also increases[3]. To estimate the total resistance of the full-scale ship, in addition to the drag values, all attitudes including trim and sinkage must be considered and predicted accurately[4].

The planing hulls moving at high speed also generate hydrodynamic lift, which reduces the wave-making resistance and may generate spray around the bow zone. The spray should also be considered and visualized in the early stages of ship design, because it is intricately tied to the geometry, trim and speed of the hull. For the most accurate results, high speed craft model tests have been the industry standard for decades[5]. Model tests estimate total resistance characteristics of a high-speed boat, but trim and sinkage values must also be measured correctly to be included in full-scale analyses[6].

Detailed sketches of ship resistance which consist of several ship resistance components, the interactions between the drag components and the quantity of their interactions, are some of the things that concern ship designers in designing fast ships[7]. Likewise, how to arrange the mechanism for the formation of ship resistance components including how to calculate or estimate the amount of resistance in the case of fast boats such as hovercraft, flying boat, and rescue boat. The results of the amount of resistance are especially indispensable in determining the amount of propulsion of a ship to meet the ship's speed as desired [8]. Ship resistance will be divided into several components which include resistance due to friction and pressure resistance due to viscosity, as well as resistance due to waves.
The interaction between the ship resistance components can be indicated by the amount of the ship drag coefficient and the Reynold and Froude numbers [9]. For fast ships, the mechanism for the formation of ship resistance components develops from the resistance formed due to the characteristics of the ship's motion. Fast ships get hydrostatic and hydrodynamic forces, the body of the ship that is immersed in the water is getting smaller, causing lower ship resistance.

Rescue boat have hard-chine planning hulls which the hard-chine planning hull is configured to develop positive dynamic bottom pressures at high speed. These positive pressures lift the hull and thereby reduce the buoyant component of hull support. Consequently, the wetted bottom area, when planning, is substantially smaller than the static wetted area. Unfortunately, the addition of induced drag associated with the development of dynamic lift results in a total resistance- weight ratio that is substantially greater than those for a displacement or semi-displacement vessel at their design speeds.

In this paper, a rescue boat analysis was carried out with a variation of 3 models with 3 approaches, among others: (a) Maxsurf Resistance, (b) CFD, and (c) Experiment. The Maxsurf software approach uses the Savitsky method where the ship uses a planning hull with Fr = 0.40 to 1.58 and a stern trim will occur. Furthermore, the CFD approach uses ANSYS CFX software with steady state simulations and multiphase domains [10]. The study objectives were to determine the total resistance of rescue boat will be validated with experimental model.

2. Rescue boat model

The high-speed craft referred to in this paper is a type of rescue boat. Rescue modelling was carried out with 3 variations of the principal dimensions as shown in Table 1

| Parameter       | Units | Boat-1 | Boat-2 | Boat-3 |
|-----------------|-------|--------|--------|--------|
| LOA             | m     | 4.56   | 5.296  | 3.758  |
| LWL             | m     | 4.3    | 4.756  | 3.559  |
| B               | m     | 1.52   | 1.815  | 1.363  |
| T               | m     | 0.35   | 0.396  | 0.30   |
| H               | m     | 0.92   | 1.035  | 0.607  |
| Vol. Displacement | m$^3$ | 0.936  | 1.414  | 0.904  |
| Displacement    | kg    | 936.4  | 1414   | 904    |

The rescue boat model uses a v-hull with a variation of chine. The first variation is without using chine, as shown in Figure 1. The next model is using 1 chine as in Figure 2 and in the third variation using the double chine. the selection of this variation is expected to be able to obtain smaller resistance at high speeds.
Figure 1 Lines and Body Plan (Boat-1).

Figure 2 Lines and Body Plan (Boat-2).
3. Method

3.1. Maxsurf Resistance

Rescue Boat resistance calculations using the Maxsurf resistance with the Savitsky method. It was used for estimating the resistance of planing hulls when in the planing speed regime[11]. In 1964, Savitsky [12] proposed a semi-empirical technique to evaluate the hydrodynamic performance of planning hulls which is simple and fast. The results of [13] showed that his presented equation for resistance prediction is very accurate in comparison with experimental tests at higher Froude numbers Fr > 1. It was also shown in [14] that this semi-empirical method is possible to implement for different prismatic hulls and has an acceptable accuracy in comparison with measurements. Generally, the equations of Savitsky’s method were presented for prismatic hulls. In a more realistic view, a planning hull has a variable deadrise angle along its length; therefore, the simple assumption of prismatic hull form is not applicable to warped planning hulls. According to [15] and [16], the effect of variable deadrise could be implemented on the [12] equations by taking effective deadrise and beam at LCG section, as following formula (1), (2) and (3), as shown at Figure 4.

\[ D = D_p + \frac{D_f}{\cos \tau} \]  

Where:

- \( D \) = Total drag
- \( D_p \) = Pressure drag;
- \( D_f \) = Friction drag
- \( \tau \) = trim angle
- \( \Delta \) = load
- \( C_f \) = Friction Coefficient
- \( v \) = velocity
- \( \lambda b^2 \) = Viscous Drag
- \( \beta \) = deadrise angle

\[ D_p = \Delta \tan \tau \]  

\[ D_f = C_f \frac{v^2(\lambda b^2)}{2 \cos \beta^4} \]
3.2. Numerical Simulation

Ship Resistance analysis also uses CFD approach. Analysis using CFD involves solving the governing equations of fluid flow numerically. The three governing equations of fluid flow are the continuity equation or the mass conservation equation, the momentum conservation equation or the Navier-Stokes equations, and the energy conservation equation. Every CFD code solves the mass and momentum conservation equations in the background as this form the basis of any fluid calculations.

3.2.1 Governing Equations

The averaged continuity and momentum equations for incompressible flows may be given as in the following two equations [17]. The instantaneous equations of mass, momentum can be written as follows in Equations (4) and (5):

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0
\]  

(4)

\[
\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \times \mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{\tau} + \mathbf{S}_M
\]  

(5)

where the stress tensor, \(\mathbf{\tau}\) is related to the strain rate as follows:

\[
\mathbf{\tau} = \mu \left( \nabla \mathbf{U} + (\nabla \mathbf{U})^T - \frac{2}{3} \nabla \mathbf{U} \right)
\]  

(6)

The computation which is applied uses the SST model that provides boundary layer modelling with high accuracy because it uses k-\(\varepsilon\) and k-\(\omega\) combined for numerical simulations. The SST model equation provides high accuracy by separating fluid flow in turbulent flow areas. By applying those two models, it includes the boundary layer area that is close to the model wall and which is far from the model wall can be appropriately covered. Bradshaw’s relationship is also applied in SST model to perform predictive turbulence with good separation [18], as following Equations (4) and (5):

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma \omega \frac{\partial \omega}{\partial x_j} \right) + G^\omega - k + Yk + Sk
\]  

(7)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \Gamma \omega \frac{\partial \omega}{\partial x_j} \right) + G^\omega - Y \omega + D \omega + S \omega
\]  

(8)
where $G^\sim k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, $G\omega$ represents the generation of $\omega$, $\Gamma k$ and $\Gamma \omega$ represent the effective diffusivity of $k$ and $\omega$, respectively. $S_k$ and $S\omega$ are user-defined source terms. $Y_k$ and $Y\omega$ represent the dissipation of $k$ and $\omega$ due to turbulence. $D\omega$ represents the cross-diffusion term.

### 3.2.2 Numerical Simulation

Calculation have been performed with the commercial CFD code ANSYS CFX. The code is based on Finite Volume method. The parameters and solver settings used to define the numerical solution are provided in Table 2.

| Properties          | Fine Mesh                                      |
|---------------------|-----------------------------------------------|
| Grid                | Unstructured with Inflation                   |
| Total Element       | Approx. 568 to 2,876                          |
| Domain              | Air + Water, SST Turbulence                   |
| Boundary physics:   |                                               |
| Inlet               | Velocity Inlet $Fr=0.40$ to 1.58              |
| Outlet              | Velocity Outlet $Fr=0.40$ to 1.58             |
| Top                 | Opening                                       |
| Bottom              | Wall with symmetry                            |
| Model Rescue Boat   | Wall with no-slip                             |
| Solver settings:    |                                               |
| Advection scheme    | High Resolution (ANSYS, 2017)                |
| Convergence criteria| Residuary type: RMS, Target: 0.0001           |

The computations using free-surface model apply, velocity outlet, and symmetry conditions for the inlet, the outlet, and centre/side/top/bottom boundary planes, respectively as shown at Figure 5. The standard wall function is used to enhance computational efficiency [19, 20].

![Figure 5: Computational Domain.](image1)

![Figure 6: Unstructured mesh with inflation.](image2)

The grid generator of Design Modeler was used for meshing the computational domain with tetrahedral grid elements with approx. 568 to 2,876 ($x10^3$) with inflation. The mesh was refined over the hull surface and in the region of water free surface near the ship hull to correctly predict the value of hull pressure resistance and to get a sharp free surface. Figure 6 show the unstructured tetrahedral grid with inflation mesh of the computational domain and on the Rescue Boat hull surface used with ANSYS CFX solver.

Selection of the meshing amount through the independent grid study approach as shown in Table 3 and Figure 7. Independent Grid Simulation was carried out on the boat at velocity 17 knot. The
Table 3 Grid Independence.

| Element (x 1000) | 568 | 1,103 | 1,583 | 2,876 |
|------------------|-----|-------|-------|-------|
| Resistance (kN)  | 2.969 | 2.465 | 2.265 | 2.228 |
| Difference (%)   | 17.00 | 8.11  | 1.63  |       |

Figure 7 Grid Independence Study.

3.3. Experiment
The experiments were carried out in the Hydrodynamic Laboratory of Naval Architecture Department at the Institut Teknologi Sepuluh Nopember (ITS) at Surabaya, Indonesia. Rescue boats were tested using a scale of 1: 11.3 in order to obtain the model sizes as shown in Table 4

Table 4 Physical Model Test.

| Parameter       | Boat-I | Model | Unit |
|-----------------|--------|-------|------|
| LOA             | 4.560  | 0.403 | m    |
| B               | 1.520  | 0.106 | m    |
| T               | 0.350  | 0.031 | m    |
| H               | 0.920  | 0.082 | m    |
| Displacement of model | 936.400 | 0.645 | kg |

The test model creation procedure followed ITTC recommendations 7.5-02-05-01[21]. The size of the model is 1: 11.3 from the actual ship size, $\lambda = 1/7$ [22]. The size of this winged ship test model adjusts the size of the towing tank test facility at the Hydrodynamic Laboratory of Naval Architecture Department at the ITS with a length of 50 m of pulling pool; width 3 m; and a depth of 2 m. The maximum towing carriage speed is 4.5 m/s. The mass density of the air towing tank is 999.1 kg / m3, and the mass density of the air is + 1.164 kg/m3, (for temperatures of 30 °C), while the water temperature ranges from 16 - 18°C.

The test model is attached to the towing carriage through the towing guide (see Figure 8) with the setup that the test model can only move heave and pitch freely, cannot move yawing or swaying so that no rolling or heeling moments arise. The trim meter was installed in an upright position at the front and at the rear of the test model in a position that did not interfere with the towing guide.
The test model is installed with the center line of the model right in the center line of the draw pool and parallel to the side wall of the pull pool. The instrumentation cable is laid in such a way that at the time of measurement it does not interfere with the motion of the test model.

![Figure 8 Towing guide frame for holder of Boat-1 test model in Towing Tank.](image)

4. Results and Discussion
Rescue Boat resistance calculations with the Savitsky and CFD approaches show the same trend, as shown in table 5. In sequence, the total resistance coefficient of the rescue boat with Savitsky starting from the Boat-2 model produces a $C_T$ of 2.9696, then the Boat-1 model with a $C_T$ of 2.0453 and finally the Boat-3 model with a $C_T$ of 1.6963. As well, the largest order of total resistance coefficient CFD rescue boat by starting from the model Boat-2 generate $C_T$ of 3.2937, then the model Boat-1 with $C_T$ at 2.5946 and the last with $C_T$ models Boat-3 at 2.4330.

| Fn    | Savitsky | CFD         |
|-------|----------|-------------|
|       | Boat-1   | Boat-2     | Boat-3 | Boat-1 | Boat-2 | Boat-3 |
| 0.40  | 0.0217   | 0.0277     | 0.0270 | 0.0208 | 0.0269 | 0.0261 |
| 0.63  | 0.2567   | 0.4139     | 0.3303 | 0.1551 | 0.3454 | 0.2783 |
| 0.87  | 0.6068   | 0.9622     | 0.6759 | 0.5567 | 0.8283 | 0.6068 |
| 1.11  | 1.0288   | 1.5931     | 0.9933 | 1.1092 | 1.4807 | 1.0426 |
| 1.35  | 1.4858   | 2.2362     | 1.3126 | 1.8209 | 2.2564 | 1.6300 |
| 1.58  | 2.0453   | 2.9696     | 1.6963 | 2.5946 | 3.2937 | 2.4330 |
Presented numerically computed resistance force (see Table 6 and Figure 11) shows that its pattern mostly coincides with the experimental data. The accuracy of the presented numerical approach is due to the generated mesh around the hull and the prism layer mesh, which have been used to create the computational domain. Also, the other important factor in resolving the generated wave is the capturing scheme which is introduced in the volume of fluid section of the current paper. In addition to the CFD results with experiment have 1.09% in average, this is due to the sum of the negative and positive differences so that the result is small. However, when viewed from the difference at each speed, the difference is quite large. This sizable difference is caused by the behaviour of fast boats that occurred in trim during the test but in numerical simulations it is not treated the same (fix body)
as shown in Figure 10. Furthermore, the results of Savitsky's method for resistance are shown Table 6. The average resistance error for the Savitsky's method is 10.23% compared to the experimental results. Whisker spray and deadrise angle corrections had significant effects on the resultant magnitudes of the Savitsky's resistance force. It can be said that both CFD and Savitsky's method has an acceptable trend on the resistance force in comparison with experimental results.

| Fr   | Savitsky | CFD     | Experiment | Savitsky vs Experiment | CFD vs Experiment |
|------|----------|---------|------------|------------------------|------------------|
| 0.40 | 0.0217   | 0.0208  | 0.023      | -5.65                  | -9.57            |
| 0.63 | 0.2567   | 0.1551  | 0.1814     | 41.51                  | -14.50           |
| 0.87 | 0.6068   | 0.5567  | 0.4493     | 35.05                  | 23.90            |
| 1.11 | 1.0288   | 1.1092  | 0.9936     | -3.54                  | 11.63            |
| 1.35 | 1.4858   | 1.8209  | 1.9373     | -23.31                 | -6.01            |
| 1.58 | 2.0453   | 2.5946  | -          |                        |                  |
|      |          |         |            | 10.23                  | 1.09             |

**Figure 11 Resistance of Boat-1.**

**Figure 12 Wave elevation.**
5. Conclusion
The calculation of the resistance of rescue boats with the 3 models using the Maxsurf (together with the Savitsky method for the calculation), CFD and experiment showed good agreement and provided several interesting conclusions: (1) each approach gives different results, but nevertheless has the same trend of resistance; (2) the difference between the calculation using the Maxsurf-Savitsky and CFD methods varied from 3.5% to 41.5%, thus further study is recommended; (3) however, CFD calculation is trusted to provide a more accurate picture than the Savitsky method, including in providing visualization of the flow around the rescue boat.

Since the experimental study was only carried out on Boat-1, the comparative study between Boat-2 and Boat-3 against the Maxsurf-Savitsky and CFD cannot me made. Further experimental test on the two models is found to be necessary to provide optimum results on the resistance analysis of the rescue boat.

Acknowledgment
The authors wished to thank the Institut Teknologi Sepuluh Nopember (ITS) for funding the current work under a scheme called “Research Excellence” (Penelitian Unggulan) with contract number: 816/PKS/ITS/2020.

References
[1] D. J. Kim, S. Y. Kim, Y. J. You, K. P. Rhee, S. H. Kim, and Y. G. Kim, “Design of high-speed planing hulls for the improvement of resistance and seakeeping performance,” Int. J. Nav. Archit. Ocean Eng., 2013, doi: 10.3744/JNAOE.2013.5.1.161.
[2] N. Bialystocki and D. Konovessis, “On the estimation of ship’s fuel consumption and speed curve: A statistical approach,” J. Ocean Eng. Sci., 2016, doi: 10.1016/j.joes.2016.02.001.
[3] E. O. Tuck, D. C. Scullen, and L. Lazauskas, “Wave Patterns and Minimum Wave Resistance for High-Speed Vessels,” 24th Symp. Nav. Hydrodyn., 2002.
[4] T. Tezdogan, Y. K. Demirel, P. Kellett, M. Khorasanchi, A. Incecik, and O. Turan, “Full-scale unsteady RANS CFD simulations of ship behaviour and performance in head seas due to slow steaming,” Ocean Eng., 2015, doi: 10.1016/j.oceaneng.2015.01.011.
[5] ITTC, “ITTC - Recommended Procedures and Guidelines - Resistance Test (7.5-02-02-01),” 2011.
[6] M. Fathi Kazerooni and M. S. Seif, “On the scale effects of resistance model tests of high-speed monohulls,” J. Brazilian Soc. Mech. Sci. Eng., vol. 41, no. 4, pp. 1–14, 2019, doi: 10.1007/s40430-019-1695-x.
[7] K. J. Paik, P. M. Carrica, D. Lee, and K. Maki, “Strongly coupled fluid-structure interaction method for structural loads on surface ships,” Ocean Eng., 2009, doi: 10.1016/j.oceaneng.2009.08.018.
[8] O. M. Faltinsen, Hydrodynamics of high-speed marine vehicles. 2006.
[9] L. Nikolaou and E. Boulougouris, “A study on the statistical calibration of the holtrop and Menn approximate power prediction method for full hull form, low froude number vessels,” J. Sh. Prod. Des., 2019, doi: 10.5957/JSJD.170034.
[10] ITTC, “ITTC – Recommended Procedures and Guidelines - Practical guidelines for ship CFD applications. 7.5-03-02-03 (Revision 01),” ITTC – Recomm. Proced. Guidel., 2014.
[11] BENTLEY, User Manual MAXSURF Resistance. 2018.
[12] D. Savitsky, “Hydrodynamic design of planing hulls,” Mar. Technol., 1964.
[13] D. Savitsky, “Procedures for Hydrodynamic Evaluation of Planing Hulls in Smooth and Rough Water,” Mar. Technol. SNAME News, 1976.
[14] T. C. Fu, R. Akers, T. O’Shea, K. Brucker, D. G. Dommermuth, and E. Lee, “Measurements and computational predictions of a deep-V monohull planing hull,” in 11th International Conference on Fast Sea Transportation, FAST 2011 - Proceedings, 2011.
[15] D. Savitsky, “On the Subject of High-Speed Monohulls,” *Greek Sect. Soc. Nav. Archit. Mar. Eng.*, 2003.

[16] N. Santoro, E. Begovic, C. Bertorello, A. Bove, S. De Rosa, and F. Franco, “Experimental study of the hydrodynamic loads on high speed planing craft,” in *Procedia Engineering*, 2014, doi: 10.1016/j.proeng.2014.11.143.

[17] J. H. Ferziger and M. Perié, *Computational Methods for Fluid Dynamics*. 2002.

[18] F. R. Menter, M. Kuntz, and R. Langtry, “Ten Years of Industrial Experience with the SST Turbulence Model Turbulence heat and mass transfer,” *Cfd.Spbstu.Ru*, 2003.

[19] H. A. Ahmad, A. F. Ayob, and P. History, “State of the Art Review of the Application of Computational Fluid Dynamics for High Speed Craft,” *J. Ocean. Mech. Aerospace-Science Eng.*, 2017.

[20] Sutiyo, E. Yuliora, and I. K. A. P. Utama, “Numerical Investigation Into The Pressure Distribution And Form Factor Effect Of Slenderbody Catamaran,” in *Proceeding of the International Conference on Ship and Technology (ICSOT) 2019, Developments Marine Design Architecture*, 2019, no. November, pp. 25–26.

[21] ITTC, “ITTC – Recommended Procedures Testing and Extrapolation Methods Resistance Test,” *Int. Towing Tank Conf.*, 2002, doi: 10.1017/CBO9781107415324.004.

[22] V. Bertram, “3 rd International EuroConference on High-Performance Marine Vehicles,” no. September, pp. 14–17, 2002.