Journal of Applied Science, Engineering and Technology
e-ISSN: 2722-8363 p-ISSN: 2722-8371
DOI: 10.47355/aset.v4i2.66

J. ASET

Design And CFD Analysis Of A 250 Kg Capacity Boat For Access Between Small Islands In The Lampung Tourism Area

Rizal Nazarrudin [1], and Sabiqunassabiqun [1,*]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Lampung, Gedung H Lt. 2, Jl. Prof. Soemantri Brojonegoro No. 1, Bandar Lampung 35143, Indonesia

* Correspondence: sabiqun11@eng.unila.ac.id

Received: 15.09.2024; Accepted: 09.11.2024; Published: 31.12.2024

Abstract: Characteristics from design boat capacity 3-4 people or 250 kg good for used access between island small in area tour with depth shallow base. The resistance of the boat's hull when operating plays an important role in knowing the power required by the boat. This is very important in designing a hull. Therefore, in design This will be done in numerical simulation using Computational Fluid Dynamics (CFD) to know how much power is required by the boat with a number of variation speeds. Design simulation moreover formerly created in Autodesk Inventor software, then design that has been designed will be simulated using the Ansys Fluent CFD application program. The data variables used in simulation numeric using varying speed limits namely 20 knots, 16 knots, 12 knots, 8 knots, and 4 knots, as well as condition rate flow is turbulent, wall side and top is symmetry, and the part the design that will be simulated is part hull the boat that sank in the water. The result of simulation numeric boat 250 kg capacity indicates a number of pressure and velocity contours of part hull as well as obtained Power highest at a speed of 20 knots 13938.511 Watts.

Keywords: Boat Design, Resistance, CFD, Power

1. Introduction

The charm of marine tourism in Lampung City is very attractive to both local and foreign tourists. Therefore, adequate transportation between islands is needed, namely by using boats or ships. One factor that plays an important role in planning a boat is total resistance.

The purpose of boat planning is to design the shape of the boat and determine the materials used to obtain a small or as small resistance force as possible. So that effective and efficient use of energy consumption can be obtained.

Studies related to numerical simulation methods on gillnet vessels have been conducted which obtained average results of differences in total resistance below 5% between experimental data and simulation data [1]. CFD is also used to obtain ship configurations with variations in hull shape and width that have the smallest total resistance value [2]. The optimal design will be obtained in order to obtain ship speed with the use of the smallest possible engine power [3].

Numerical simulation is the process of testing the magnitude of a model using a computer [4]. Numerical simulation has been proven to be able to replace the role of laboratory testing with virtual testing. The values produced through virtual testing are very accurate when the two are compared [5].

e-ISSN: 2722-8363 p-ISSN: 2722-8371

In this research, an analysis will be carried out on how much big total resistance and power required by the ship with variation speed use simulation numeric Ansys fluid flow (fluent). Application Ansys Fluent CFD simulation will simulate the design of the boat that was created in the application program Autodesk Inventor 2015 with capacity 250 kg and varying speed limit conditions namely 20, 16, 12, 8, and 4 knots. Conditions rate flow is turbulent, wall side and top are symmetry, design simulated ship is part of the sinking ship into the water.

1.1 Ship Obstacle

Resistance using the numerical method is the total of all forces working against the motion of the ship. Where, the resistance is always opposite to the direction of the ship's motion at a certain speed. The size of the ship's resistance is greatly influenced by the ship's speed (V). The magnitude of the ship's resistance is described by Eq. 1

$$C_T = \frac{R_T}{0.5\ \rho v^2 S} \tag{1}$$

Where:

C_T = Koefisien hambatan total
R_T = Besar hambatan total
V = Kecepatan kapal
S = Permukaan yang tenggelam

1.2 Computational Fluid Dynamics (CFD)

CFD is one of the method simulation numeric that has draft base Navier – Stokes equations, with principles:

1. Conservation of Mass
2. Conservation of Momentum
3. Conservation of Energy

CFD is used in a way wide for finish from problem in a way experiment. In an experiment, not done in a way comprehensive in simulation situations actually, so matter This No reflects in a way overall in numerical simulation.

In numerical simulation, if the input data is not entered with condition actually, then parameters can be used to reflect condition actually.

Computational fluid dynamics (CFD) programs use method volume of fluid (VOF) or finite difference equations (FDE). This method uses the system coordinate Cartesian with finish Navier-Stokes equations

2. Materials and Methods

In planning and analysis study Here, primary data and secondary data are used secondary following This:

2.1 Boat Design Shape and Size

The hull formed with specification part wall side right and left as well as on is symmetrical. Size design hull and body plan design can each seen in Table 1 and Figure 1

Table 1. Size design hull

Parameter	Size
L	3 m
L	1 m
h	600 mm

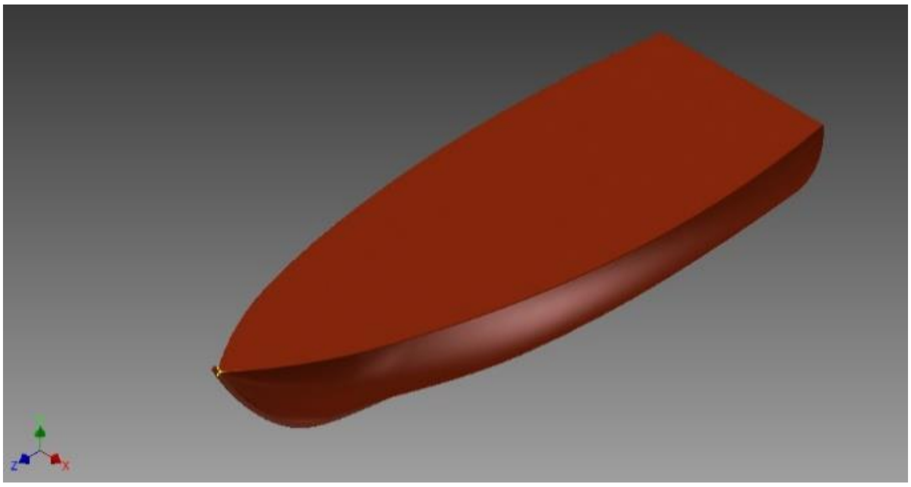

Figure 1. Initial Ship Design

2.2 CFD Analysis Design

In the simulation numerically performed that is focused on the part of the hull of the ship that sank in the water. Therefore, count uses Eq. 2

$$F = \rho . g . V \quad (2)$$

Obtained gastric volume sinking boat, so tthen used for creating geometry on the tunnel with size width 5 m, high 2 m and length 20 m shown in Figure 2.

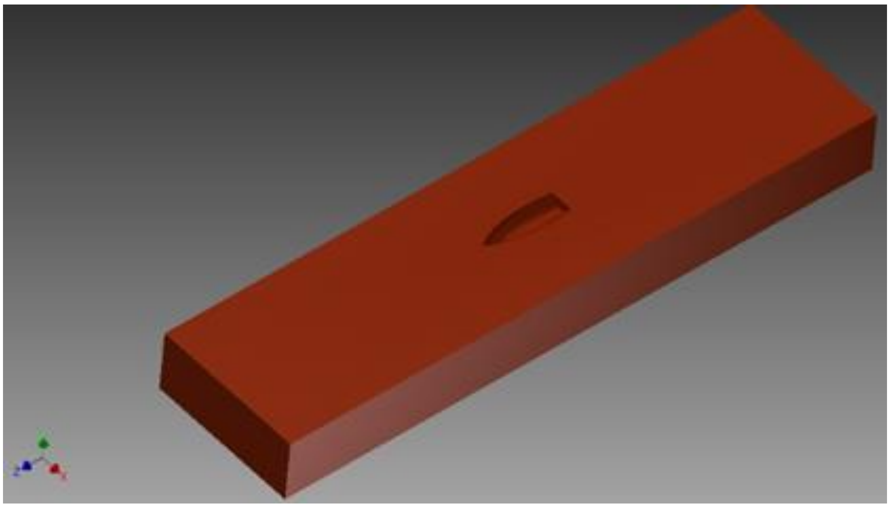

Figure 2. CFD Geometry

After geometry simulation numeric has been obtained, steps furthermore are a mesh process. The grid topology is taken as a hybrid with a focus on each side near the wall boat brought together mesh. Figure 3 shows results meshed geometry.

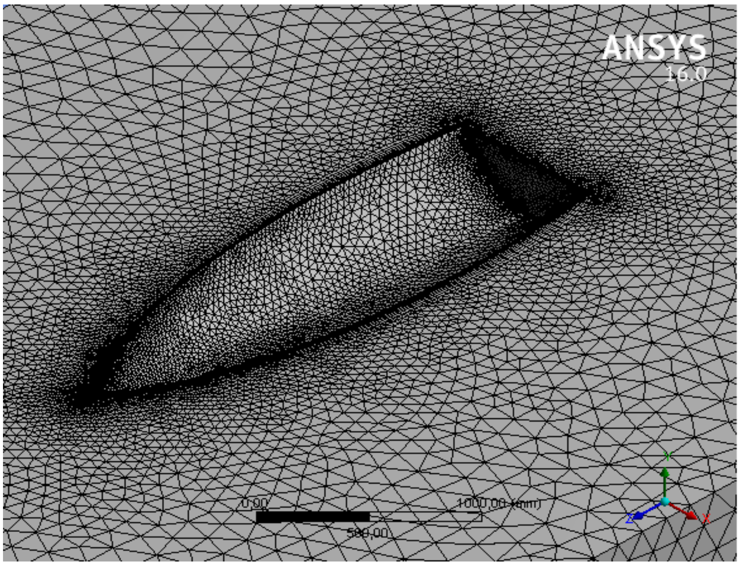

Figure 3. Meshing Results

3. Results

After doing simulation numeric using ansys fluent, obtained some of the data including contour pressure static and contour velocity on variation speeds of 20, 12, and 4 knots are shown in Figures 4 and 5 respectively.

3.1. Static Pressure Contour

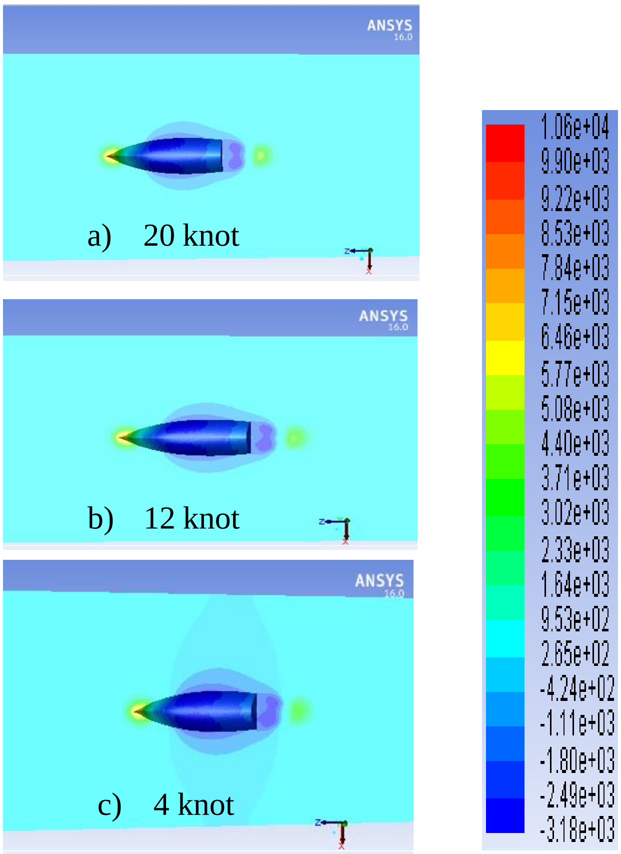

Figure 4. Pressure Contour a) 20 b) 12 c) 4

Figure 4 shows the pressure of the speed variation of 20, 12, and 4 knots. The contour in the figure shows that the pressure around the wall at a speed of 20 knots is obtained at 1355.8863 N. The greatest static pressure at any speed occurs at the front of the boat where it receives pressure from the fluid against the boat walls, and the pressure at the rear of the boat tends to be negative because the fluid flow pushes forward.

3.2. Velocity Contour

Figure 5 shows contour speed from variation speed of 20, 12, and 4 knots. The contours in the image seen the speed at the part that is not affected by the boat wall is stable in each variant, but the area on the boat wall experiences a decrease in speed due to the friction from the boat wall.

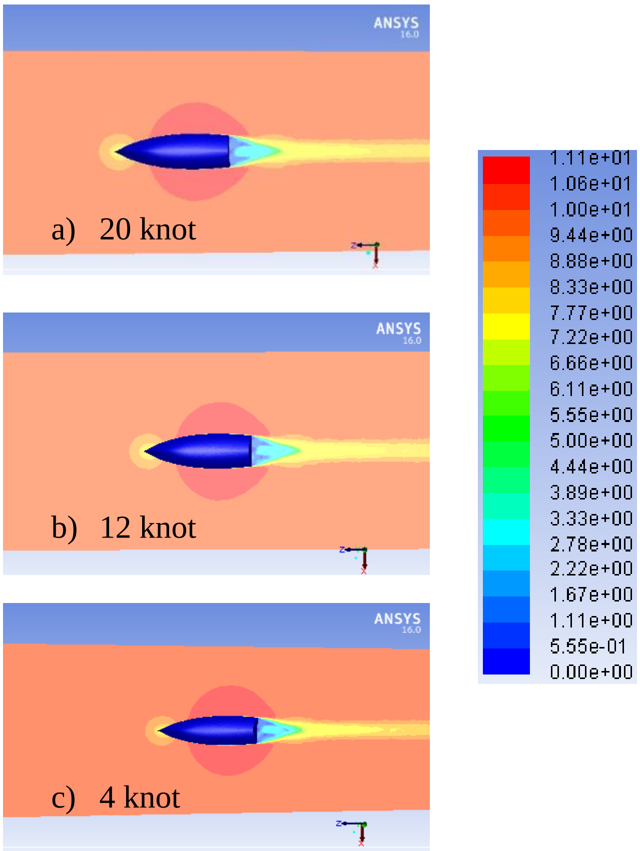

Figure 5. Velocity Contours a) 20 b) 12 c) 4 knots

4. Discussion

4.1 Power

Style and power gained from simulation presented respectively in Table 2

Table 2. Graph Data Table

No	Data		
	Speed (knot)	**Force (N)**	**Daya (W)**
1	20	1355.8863	13938.511

e-ISSN: 2722-8363 p-ISSN: 2722-8371

No	Data		
	Speed (knot)	Force (N)	Daya (W)
2	16	870.47852	7158.815
3	12	497.67123	3069.636
4	8	219.42239	902.2649
5	4	56.62468	116.4203

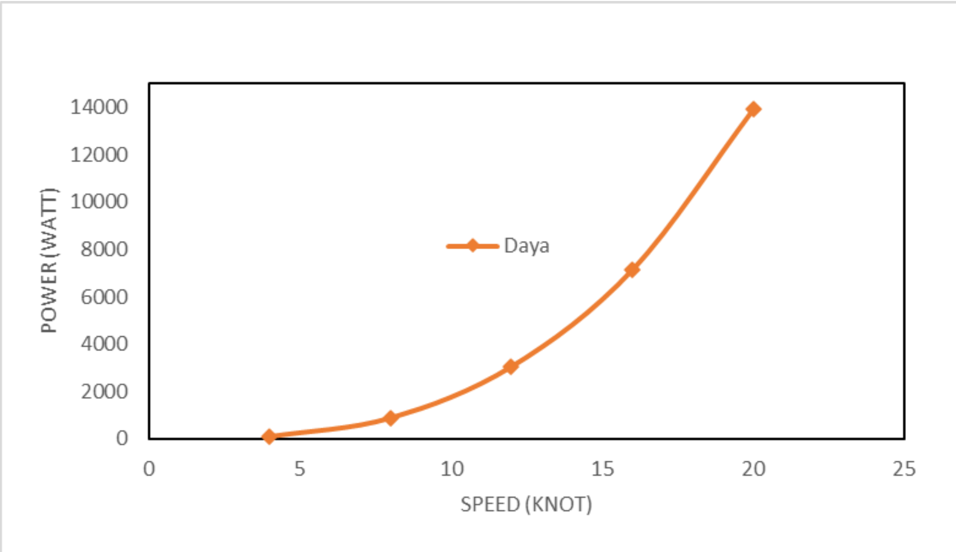

Figure 6. Graph between power – speed

Design results modeling based is the ideal possible outcome for happened. The result can be made into reference for analyzing the big power required by the boat for can drive.

Design results based on experimental condition speed boats will greatly affect the big power required by the boat the Foregoing. This is because of obstacles that occur between water and the wall the boat will also be big. So that need exists evaluation of the design that influences systems like form boat, speed water flow is needed under consideration for determining the motor to be used, and so on.

5. Conclusions

Based on results and Discussion design based on modeling using Ansys Fluent CFD application with variation rate flow mass, can be concluded as following: 1. Pressure static on the fifth variation speed owns the same tendency. The most complex static pressure occurs in the boat wall section. 2. The power obtained in each variation is different, power the biggest required by variation speed of 20 knots, namely 13938.511 Watts and power the smallest obtained at a speed of 4 knots, namely 116.42034 Watts.

 doi: 10.47355/aset.v4i2.66

References

[1] R. M. . Hutauruk, "Simulasi Numerik Tahanan Kapal Gillnet Menggunakan Pendekatan Computational Fluids Dynamics. Jurnal Perikanan dan Kelautan, 18(1), pp. 35-47.," *J. Perikan. dan Kelaut.*, vol. 18, pp. 1–13, 2013.

[2] D. dkk Chrismianto, "Pengaruh Variasi Bentuk Hull Kapal Catamaran Terhadap Besar Hambatan Total Menggunakan CFD," *Kapal*, vol. 11, no. 2, pp. 99–106, 2014.

[3] B. W. Sasongko, "Analisa Pengaruh Variasi Bulbous Bow Terhadap Hambatan Total pada Kapal Katamaran untuk Penyebrangan di Kepulauan Seribu menggunakan CFD," *Kapal*, vol. 3, no. 4, pp. 439–450, 2015.

[4] M. R. D. A. Kusuma, D. Chrismianto, and S. Jokosiworo, "Pengaruh Posisi Foil Terhadap Gaya Angkat Dan Hambatan Kapal Katamaran," *Kapal*, vol. 14, no. 2, p. 58, 2017, doi: 10.14710/kpl.v14i2.16487.

[5] T. J. Blackwell, "Subsonic Wind-Tunnel Wall Corrections on a Wing With a Clark Y-14 Airfoil," no. May, p. 222, 2011.