11th International Conference on Marine Technology
MARTEC 2018

WAKE TURBULENCE AND DIMENSIONAL LENGTH CHANGES INFLUENCE ON CONVENTIONAL SHIP BASED ON RESISTANCE CRITERIA

Bagiyo Suwasono[1*], Ali Munazid[1], G.A.P. Poundra[1] and Sutiyo[1]

[1]Faculty of Engineering and Marine Science,
Hang Tuah University,
Sukolilo-Surabaya 60111 Jawa Timur, Indonesia

ABSTRACT

In the utilization as a floating vessel, ships shall always interact with the dynamic of the fluids in her surrounding environment. Through this interaction, some wake turbulence will be formed as the ship cruise along the water. On a conventional ship, length has become one of the factors which can create various numbers and forms of wake turbulence, this wake turbulence then believed can provides the positive Turbulence Kinetic Energy (TKE) that be able to enhance the ship acceleration or the negative one which creates more ship resistance. The investigation concerned on how the dimensional length changes of the conventional ship will affect the TKE through the wake turbulence analysis. The research carried out numerically with Ansys Fluent CFD code for the further turbulence impact analysing and will only focus on ship performance based on the resistance criteria with ideal condition boundary layer which can be the research parameter. The overall results were compared with published data for validation purposes. The results are also believed to be useful for the development of the maritime industry..

Keywords : *conventional ship, wake turbulence, dimensional length, ship resistance*

1.0 INTRODUCTION

Ship is a large watercraft that travels the world's oceans and other sufficiently deep waterways, carrying passengers or goods, or in support of specialized missions, such as defence or research. The first known vessels date back about 10,000 years ago, however could not be described as ships. This allowed men explore or travel widely for the settlement of Oceania for example [1]. Furthermore, ship have already developed and utilized as one of the most important main transportation mode in the modern society due to her ability to operate on and below the water surface [2].

Throughout the years, creating and building such an efficient ship have already become a challenge for the naval architects around the globe. One of the important factor in accelerating the design efficiency is to modify the shape and the dimension of the vessel herself. By optimizing these two variables, the total resistance of the hull can be decreased and turbulence kinetic energy can be increased as the hull becoming more streamlined or *hydrodynamically* smooth. In building such hull condition, changes and developments are needed to achieve the convergence in both shape and principle dimensions, mainly in dimensional length.

*Corresponding author: bagiyo.suwasono@hangtuah.ac.id

2.0 THE DESIGN

In an attempt to improve the efficiency while also increase the kinetic energy of the ship hull, some principle dimensions have been applied to the same vessel. In order to simplify the investigation, the research will only focus on the changes of dimensional length without changing the breadth, height, the draft and the shape of the ship. The main design and the principle particulars, are shown in Figure 1 and Table 1.

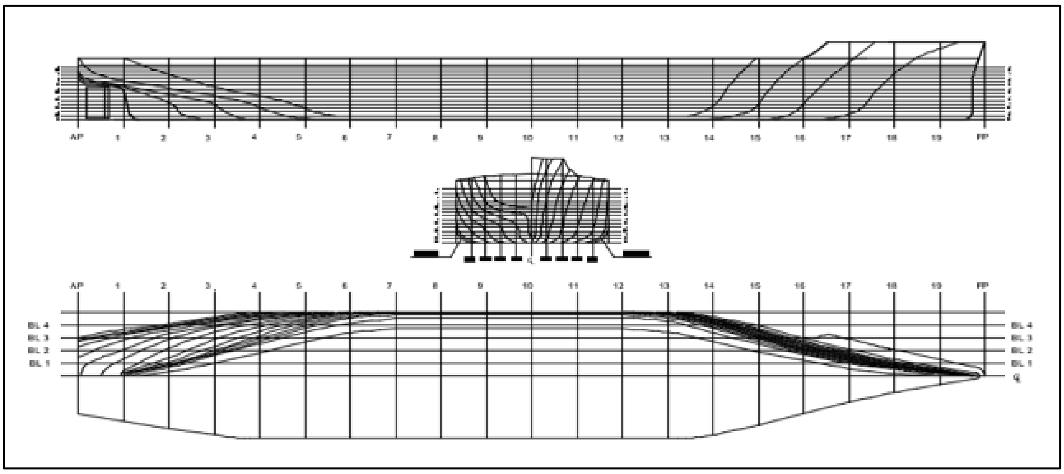

Figure 1. Lines Plan

The difference in dimensional length will further believed to create different TKEs in the ship hull both locally and globally. The variety of the kinetic energy occurs due to the wake pattern differences which then influenced the wave transport especially in the transition flow.

Table 1. Principle Particulars for Vessel Model 1, 2, 3 and 4

Designation	**Model 1**	**Model 2**	**Model 3**	**Model 4**	**Units**
Length Over All	98	128	144	152	meters
Breadth Moulded	16.5	16.5	16.5	16.5	meters
Draught	5.4	5.4	5.4	5.4	meters
Height	8	8	8	8	meters
Displacement	5258	8446	9735	10382	tons
Cruising Speed	11.9	11.9	11.9	11.9	knots
L/H Ratio	12.25	16	18	19	-

These variety of lengths were created to satisfy the parameters of Indonesian Bureau of Classification (BKI) 2017 in the approximation ratio of ship principle dimension selection i.e.

1. L/H = 16 for Unlimited Range of Service
2. L/H = 18 for Coastal shipping
3. L/H = 19 for Sheltered Shallow Water Service [3]

3.0 FLUID ANALYSIS

The resistance analysis was carried out by using Computational Fluids Dynamics (CFD) software. CFD is a branch of fluids mechanics that uses numerical analysis and algorithms to solve and analyse problems that involve fluids flow, and computers are used to perform the calculations which were required to simulate the interaction of liquids and gases with surfaces defined boundary conditions [4].

In order to be able to observe the complex transport quantities phenomenon (in this case is turbulence), the continuity flow equation as a flux was used. In common applied mathematical definition, flux defined as a concept which describes the quantity which passes through a surface or substance [5], which described as:

$$\frac{\partial\rho}{\partial t} + \nabla . \mathbf{j} = 0 \quad (1)$$

Where, $\mathbf{j}$ is the mass flux which equal to the density of the fluids times velocity or can be described as:

$$\mathbf{j} = \rho u \quad (2)$$

In all of these approaches, some basic procedures are followed.

1. During pre-processing [6]:
   a. The geometry of the problem is defined.
   b. The volume occupied by the fluid is divided into discrete cells (mesh).
   c. The physical model is defined. Boundary condition is defined.
2. The simulation is started and the equations are solved iteratively as a steady-state or transient.
3. Post-processor is used for analysis and visualization of the resulting solution.

The fluid flow simulation through CFD used to analyze the flow dynamics phenomenon thus influencing the total resistance measurements. The other equations which being used to express the flow movement further is the Navier-Stokes equation and Adverse Pressure Gradient equation for expressing the flow interaction in every models.
In the term of flux influence in fluid continuity which state:

$$\frac{d}{dt}(\rho u) + \nabla.(\rho u) + S = 0 \quad (3)$$

Since the flux is being one of the parameter in the equation, the pressure variable in the continuity determined as the total particles accumulation transported in one fluid acceleration with constant volume and density in accordance of its boundary condition.
Furthermore, in the term of fluid interactions along the boundary layer adverse pressure gradient approach is being used, mathematically described as:

$$\frac{d}{dx}\left(\rho\frac{v^2}{2} + P\right) = 0 \quad (4)$$

In the process of turbulence model analysis, the K-epsilon model was used. The K-epsilon (k-ε) is the most common model used in Computational Fluids Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equations model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.
The transport equation for standard K-epsilon (k-ε) can be express as follows [7]:

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon U_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(P_k + C_{3\varepsilon}P_b) - C_{2\varepsilon p}\frac{\varepsilon^2}{k} + S_\varepsilon \quad (5)$$

In this research, the environmental situation was assumed for having steady flow so that the velocity of the external flow and wave were also being steady. In this boundary layer, the turbulence intensity of the liquid is set to 0.1% in both inlet and outlet flow as the ship was situated at the open ocean boundary condition. Thus, the adverse pressure gradient on the steady flow where $v = v(x) = v(x(t))$ written as:

$$\frac{v^2}{2} + \frac{P}{\rho} = C \quad (6)$$

The fluid simulations of the CFD were running quite successfully, the results with various characteristic differences which showed in the figures below:

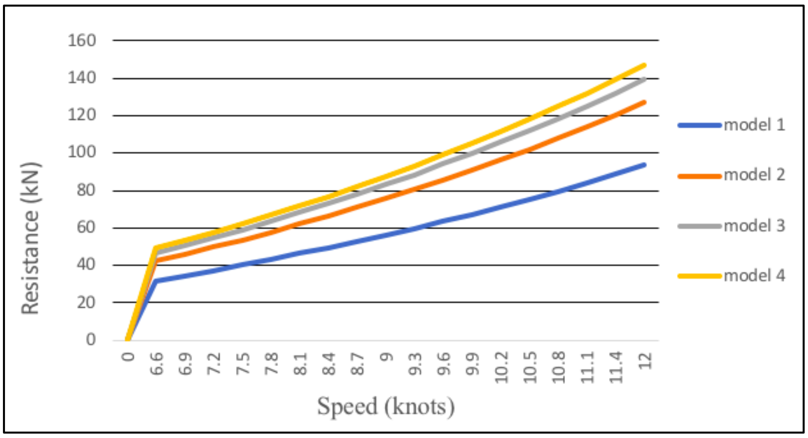

Figure 2. Total Resistance vs Speed

The resistance calculations in conventional ship hull showed that a longer ship will have a larger amount of resistance due to a larger amount of wetted surface area and submerged hull area. This result also showed that the dimensional length differences shall create significant changes in ship resistance. Although the amount of the drag force of each model are gradually increased as the vessel accelerated, the length differences also provide a quite significant deviation among the simulation results in every models. This phenomenon occurs because the existence of the turbulent kinetic energy (TKE) in every models which led to the drag force differences.
Assuming density and viscosity both constant, the full form of the TKE equation is [8]:

$$\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = -\frac{1}{\rho_\circ} \frac{\partial \overline{u_i' p'}}{\partial x_i} - \frac{1}{2} \frac{\partial \overline{u_j' u_j' u_i'}}{\partial x_i} + v \frac{\partial^2 k}{\partial x_j^2} \quad (7)$$

From the simulation, the TKE of each model increased gradually in various ways. The result shows that kinetic energy from the model 1 and model 2 increased fluctuatively while the model 3 and model 4 creates a quite stable kinetic energy escalation. Although having the same fluctuative increases, the model 2 tend to be more stable TKE in her starting point when the vessel accelerated her initial speed compared to the model 1. This shows that the longer the ship, the more stable her TKE will be. This phenomenon occurs due to the longer ship will have longer parallel middle body which make the fluid flow tend to have the same wake form which leads to the stable turbulence pressures escalation.

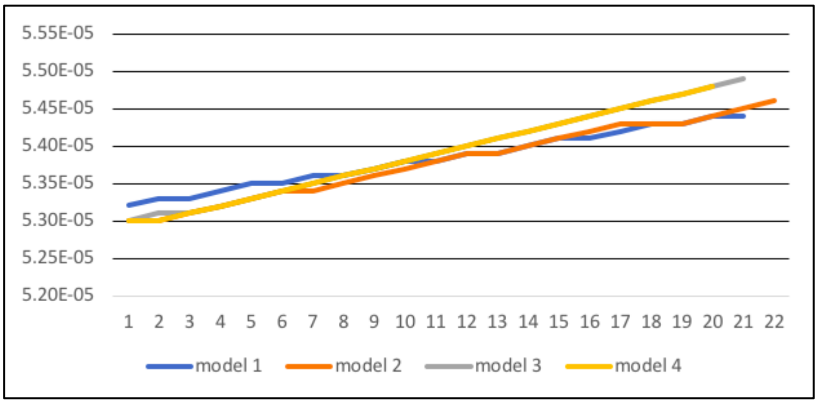

Figure 3. Turbulent Kinetic Energy (TKE)

As an attempt to enhance the result of the research, the interaction of the ship and her surrounding environment has been expressed by calculating the adverse pressure gradient which occurs due to the existence of the Turbulent Kinetic Energy (TKE) differences. The calculation was conducted by investigating the turbulence pressure difference phenomenon in every ship hull models which have already simulated iteratively.

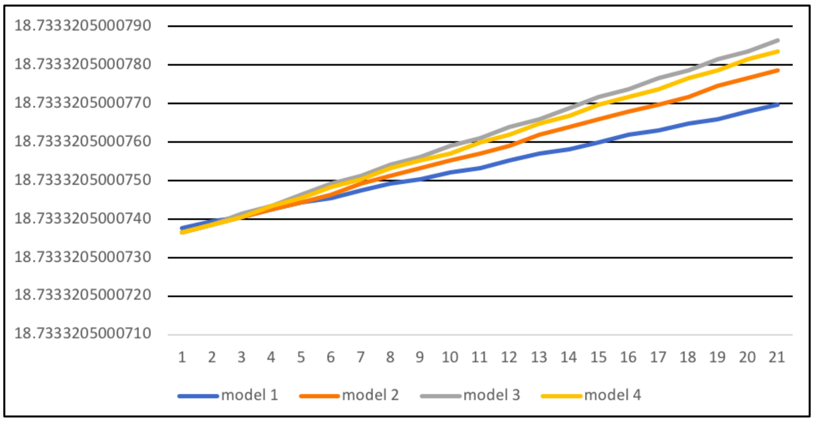

Figure 4. Adverse Pressure Gradient (Pascal)

From the static pressure simulation result, model 3 achieved much more amount of pressure compared to the other models. This shows that a vessel with a value of 18 in the L/H ratio, will achieve more pressure due to a higher Turbulent Eddy Dissipation (TED) value. However, both model 3 and model 4 creates the same stable value of Turbulent Eddy Dissipation and Adverse Pressure Gradient which led to fluctuation decreases in their Turbulent Kinetic Energy values. This result also shows that with a larger area of parallel middle body, the Adverse Pressure Gradient of the turbulence around the hull will generate in the same value, which tend to create a lower fluctuative pressure escalation. This too be in accordance with the TED values, where a longer vessel will dissipate less eddies which then give the vessel higher amount of kinetic energy. A longer vessel will also decrease the amount of pressure and TED as the area of the parallel middle body increased and make more possibilities for the energy of the turbulence generate in a better stability due to a longer period of energy rest. This proofed by the result which shows that the model 4 is having less TED and pressure so the amount of TKE increase in the same state like model 3.

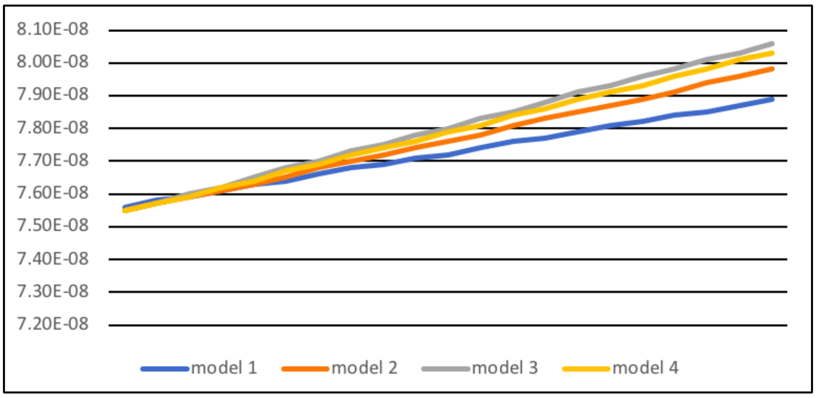

Figure 5. Turbulent Eddy Dissipation m^3/s^2

Based on the results, the fluctuation which exist in the Turbulent kinetic energy increases due to the dimensional length changes will not just creating an impact to the vessel but also to the other substances which works around the ship body. This phenomenon showed by the total pressure distributions throughout the hulls to the fluid substances around them.

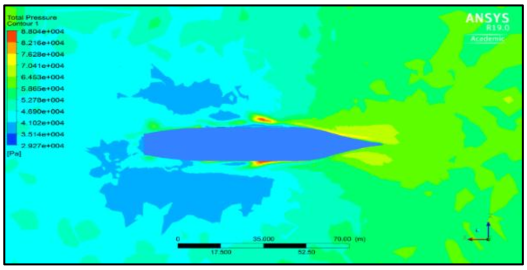

Figure 6.1. Model 1 Pressure Distribution

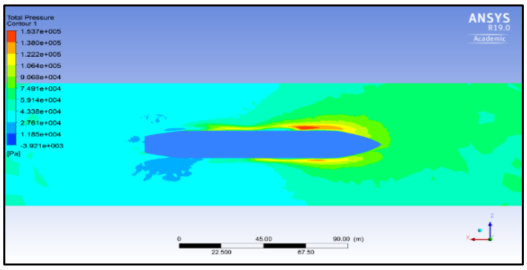

Figure 6.2. Model 2 Pressure Distribution

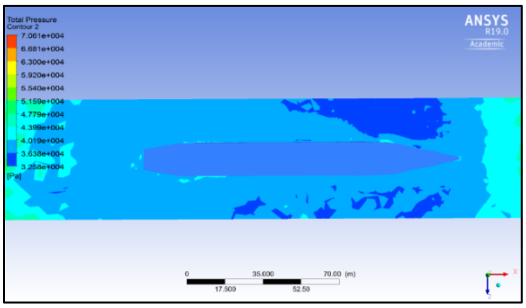

Figure 6.3. Model 3 Pressure Distribution

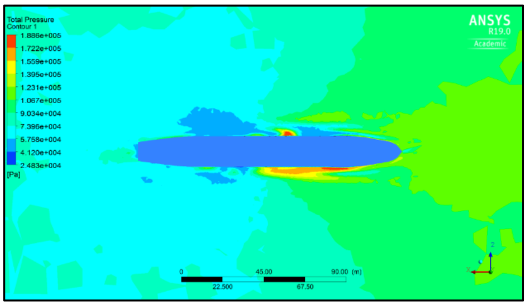

Figure 6.4. Model 4 Pressure Distribution

Through the experiments, the result also shown how the turbulent energy kinetic difference caused by the hull length changes will creates a different wake form due to the existence of the total fluid pressure distribution differences which created through the fluid substance and the hull interaction.

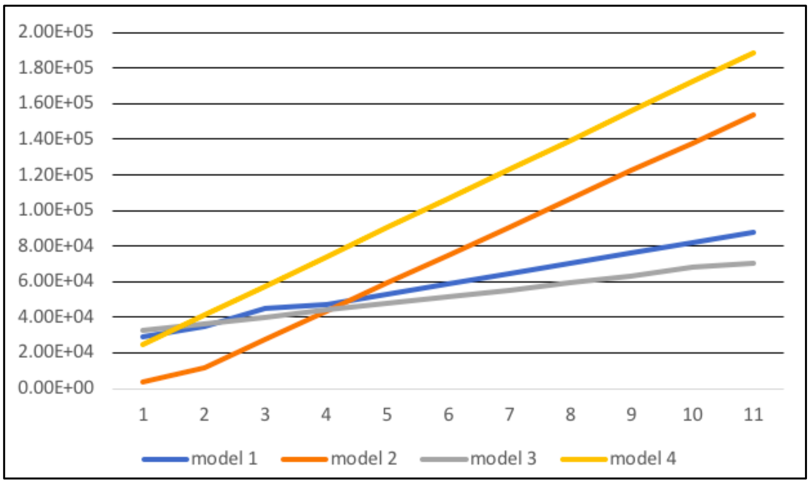

Figure 7. Fluid Total Pressure Interaction

According to the data, the result shown that the dimensional length also influences the external pressure distribution as well as the energy kinetic that works along the ship body. These data also shown that a longer ship will create a better pressure distribution to her environment. As the energies distribute better (both pressure and kinetic), this proves that a longer ship shall have less energy resistance due to a negative energy distribution.

4.0 CONCLUSIONS

The study of the wake turbulence and dimensional length changes influence on the conventional ship based on the resistance criteria has been demonstrated by using Ansys CFD software quite successfully. The results of the investigation led to some conclusions as follows:

1. The longer ship will have larger amount of resistance due to a larger amount of wetted surface area and submerged hull area.
2. The longer the ship, the more stable her TKE will be.
3. Longer Hull shall generate a stable Adverse Pressure Gradient and Turbulent Eddy Dissipation and could having a possibility in decreasing the amount of those values.
4. The shorter ship will have lower amount of resistance. However, a shorter ship will tend to have unstable TKE increases.
5. The longer ship will have less energy resistance

ACKNOWLEDGEMENTS

The authors gratefully thanked the LPPM of Hang Tuah University which funded and supported this research in fiscal year 2018.

REFERENCES

1. Romola, A. and Anderson, R. C.,2003. *A Short History of the Sailing Ship,* Dover Maritime Series.
2. UNCTAD (United Nation Conference on Trade and Development), 2007, p. 32.
3. Biro Klasifikasi Indonesia (BKI),2017. *Rules for The Classification and Construction*, Volume II, part 1.
4. Jameson, A. and Caughey, D.,1977. A Finite Volume Method for Transonic Potential Flow Calculations, *The Third AIAA Computational Fluid Dynamics Conference*, Albuquerque New Mexico, 77-635.
5. Bird, R. B., Stewart, W. E. and Lightfoot, E. N.,1960. *Transport Phenomena*. Wiley.
6. Luhulima, R. B, Utama, I. K. A. P. and Sulisetyono, A., 2016. Experimental Investigation into the Resistance Components of Displacement Trimaran at Various Lateral Spacings, *International Journal of Engineering Research & Science (IJOER) 2*, 21-29.
7. Versteeg, H. K. and Malalasekera, W.,2007. *An introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson Education.
8. Baldocchi, D.,2005. *Lecture 16, Wind and Turbulence, Part 1, Surface Boundary Layer: Theory and Principles*, Ecosystem Science Division, Department of Environmental Science, Policy and Management, University of California, Berkeley, USA.