EFFECT OF TURBULENCE MODELS IN ANALYZING VISCOUS RESISTANCE OF CATAMARAN BY USING COMPUTATIONAL FLUID DYNAMICS

Ronald Mangasi HUTAURUK

Department of Water Resources Utilization, Faculty of Fisheries and Marine Science
University of Riau, Pekanbaru.
E-mail: ronald.mhutauruk@yahoo.co.id

Abstract

In addition an experiments carried out both the wind tunnel test and the towing tank, the study of viscous resistance on the catamaran vessel can also be analyzed using numerical simulation by Computational Fluid Dynamic (CFD). The aims of the study is to analyzing the most accurate turbulence models compared with the results obtained through experimentation. Catamaran model size adapted to test models in wind tunnel experiments. The ratio of separation to length between demihulls (S/ L) selected is 0.3. Boundary conditions of the fluid in the numerical simulations including the physical condition based on experiments data. Speed varied from 10, 12, 14, and 15.4 m / s. Turbulence models tested were k-□□, Shear Stress Transport, and the RNG k-□□. Results of analysis the viscous drag coefficient on each of the turbulence models will be compared with the experimental value. Turbulence models using a Shear Stress Transport (SST) is the best model of turbulence with percentage difference about 2.122% higher than in the wind tunnel experiments.

Keywords: *CFD, k-Epsilon, Shear Stress Transport, RNG k-Epsilo, Catamaran*

1. Introduction

CFD abbreviation's stands for computational fluid dynamics. CFD represent a vast area of numerical analysis in the field of fluid's flow phenomena. In the applications of fluid dynamics, flow around the bodies is really important. Characteristics of forces acting on the bodies are needed to produce effcient aerodynamics (aircraft, cars, automobiles) and hydrodynamics design (for ships).

The classical analysis of fluid flow problems involves the initial representation of the fluid motion by a physical model. This physical model is then interpreted as a mathematical model by setting down the conservation equations which the flow satisfies. These governing equations are then subject to analysis leading to closed-form solutions with the dependent variables, such as the velocity components and pressure, varying continuously throughout the flow field. This general approach and the closed-form solutions which result provide powerful insights into many important fluid flow problems, including the effects of changes in geometry and flow parameters such as Reynolds number. However, there are many circumstances where it is not possible to derive analytical solutions while at the same time retaining a mathematical model which displays all the essential features of the physics underlying the flow. Here the limitations of the analytical approach become evident. By way of contrast, CFD is capable of delivering solutions to the governing equations which fully describe all aspects of the physics of the flow (Massey and Smith, 2006).

In this research, CFD is used in solving viscous resistance of catamaran as a set of the high-speed vessel that has been growing rapidly for the last ten years (Couser et al.,1997); Jamaluddin, et al., 2010). As reported by Drewry Shipping Consultants (Sahoo et al., 2006), catamarans account for 43% of the fleet by vessel numbers. Slender hull forms and higher speed capabilities provoked the need of technological evolution in predicting their preliminary characteristics of resistance. Resistance of catamarans in general is divided into two major components namely, viscous resistance and wave resistance.

Catamaran consists of two demihulls which are separated at certain distance. Both of demihulls induce interaction that generates viscous interference. Due to separation to length ratio between demihulls, interference effect occurs significantly over viscous resistance. These interference effects has been investigated by experimental test in the wind tunnel at Department of Mechanical Engineering, Sepuluh Nopember Institute of Technology, Surabaya (Hutauruk et al., 2011). Hutauruk et al., 2011 also analyzed the pressure distribution and velocity profile on the catamaran. The catamaran model tested without transition strip (turbulent stimulator) at various velocities, commenced from 10; 12; 14 dan 15,5 m/s in speed and separation to length ratio

(S/L) 0,2; 0,3 and 0,4. The conclusion of the research was increasing of body separation will increase pressure distribution values. However, its increasing will decrease velocity profile values.

This research analyze the using of different turbulence models namely k-□□, Shear Stress Transport, (SST) and the RNG k-□□ on the catamaran by S/L 0,3 numerically. The viscous resistance will be investigated by using ANSYS CFX ver. 14. S/L 0,3 was choosen because from the experimental results derived and concluded that on S/L 0,3; the effect of flow interference decrease but not the smallest. The result will be validated with previous experimental result.

2. Methodology

2.1. Grid Independence

Model in the wind tunnel test designed as reflect models by using Maxsurf. The use of reflex models in a wind tunnel and CFD provides an approximate means of directly measuring the viscous drag of the models without the generation of waves (Armstrong, 2003). All its size including its principal dimension measured both models and test section then poured in the drawing. The process of CFD will be continued by using ANSYS, started from ICEM CFD as meshing step (**Figure 1**). Mesh quality will effect the result of the simulations. To derive the suitable number of mesh elements and nodes, it is calculated by doing grid indepence. Grid independence was investigated by applying grid densities from 500.000 to 1.200.000 cells. Grid independence, with up to 2 percent changed in drag, was reached at about 1.007.880 cells (**Figure 2**), which were then used in the further investigations as applicated by Utama (2002).

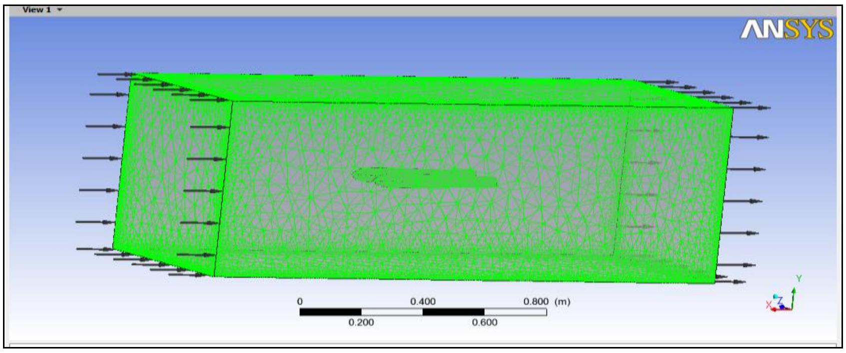

Figure 1. Model meshing

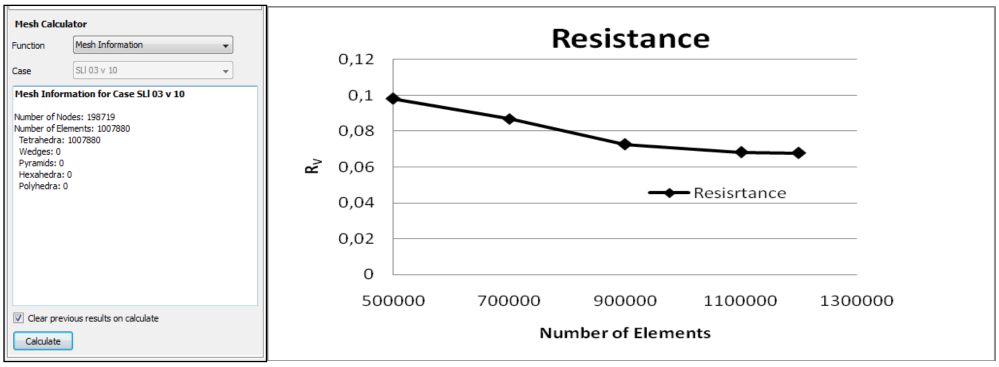

Figure 2. Model Mesh Calculator and Grid Independence Graph

2.2 Boundary Condition

Boundary conditions were set for the model must satisfy the real condition on the wind tunnel test, in other could minimize the results deviation. Inlet velocity conditions were set to 31,3 deg. C as temperature in the test section, with directional velocity 10; 12; 14; and 15,4 m/s. Reference pressure was set to be 1 atm. The outlet back pressure was set to zero to ensure no upstream disturbances. The walls of the wind tunnel were set to slip conditions to not negate near-field effects, such as wall turbulence. In addition, the surrounding enclosure walls were set to an isothermal, adiabatic surface of 31,3 deg.C. The surface of the models was set to a no-slip condition. The fluid domain used for the simulation was incompressible and isothermal. Vary of turbulence model as k-□□, Shear Stress Transport, and the RNG k-□□ was selected. All other default solver values were retained. Once the model is fully constrained, the solver is initiated. The actual time to solve the flow field depended directly upon the number of elements in the mesh (Mitchell and Webb, 2008). The description of boundary condition shown as **Figure 3**.

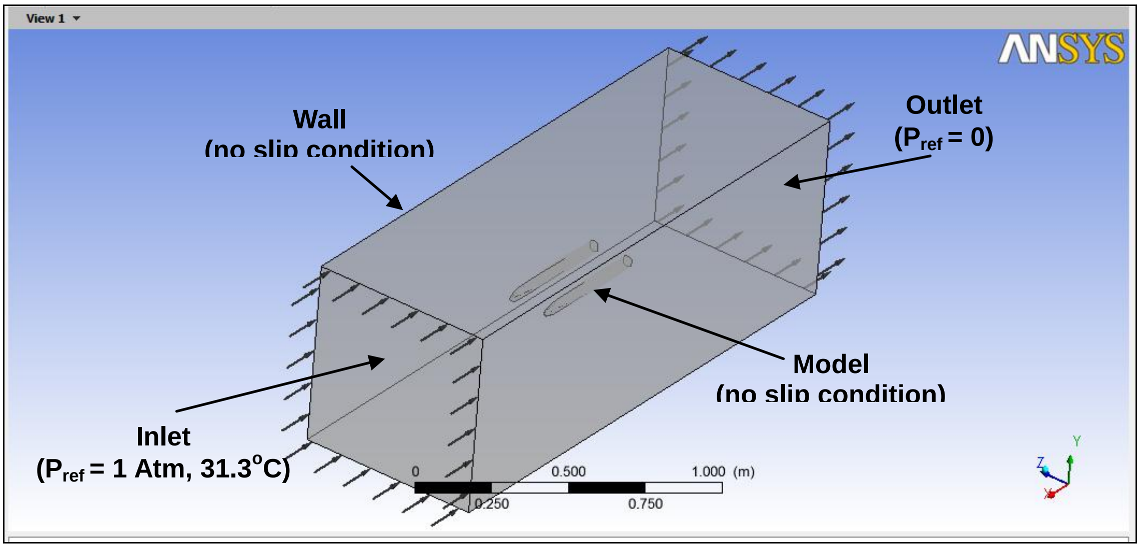

Figure 3. Boundary Condition

2.2. Turbulence Models

Turbulence is a fluid flow phenomenon characterized by unsteadiness, fluid motions that appear irregular or topologically complex, fluid motions occurring on a wide range of physical scales, rapid mixing of passive contaminants (smoke, heat, concentrations of chemical species) (Recktenwald, 2009). Whenever turbulence is present in a certain flow it appears to be the dominant over all other flow phenomena. Successful modeling of turbulence greatly increases the quality of numerical simulations (Sodja, 2007). In this research, varied of turbulence models applicated was k-□□, Shear Stress Transport, and the RNG k-□□□.

2.3. Concept of Computational Fluid Dynamics

The starting point of CFD is the specification of the governing equations of fluid dynamics as the equations of continuity, momentum and energy. These equations are then replaced by equivalent numerical descriptions which are solved by numerical techniques to yield information on the dependent variables, for example, the velocity components and pressure, at discrete locations in the flow field. The continuity, momentum and energy equations can be set down in a variety of different, but equivalent, forms and the choice of the appropriate form is an important aspect of successfully obtaining numerical solutions.

CFD is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numericals methods (discritization methods, solvers, numerical parameters, and grid generation). The process is as **Figure 4**. Fluid problem is solved by understanding the physical properties of fluid by using fluid mechanics. Then, mathematical equations is used to describe these physical properties. Generally the mathematical equation stated as navier-Stokes Equation that governing equation of CFD.

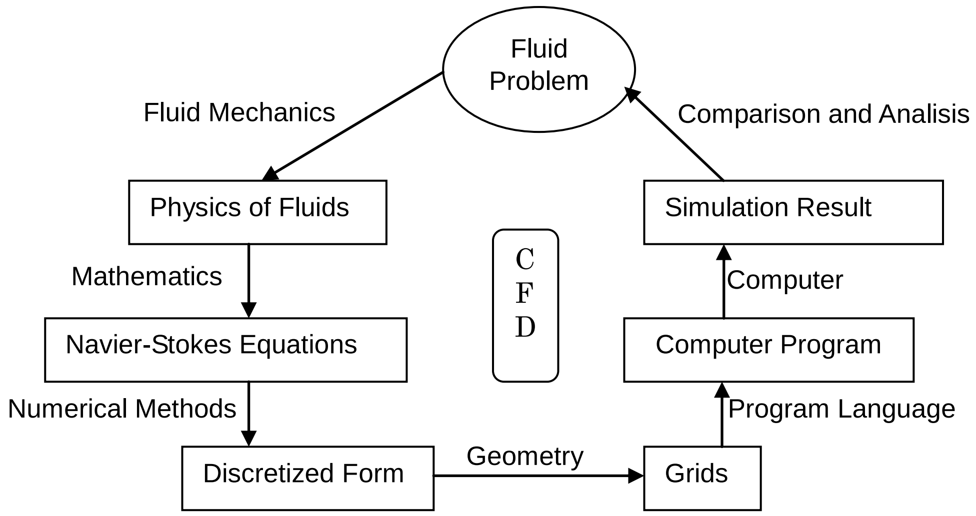

Figure 4. Process of CFD (Zuo, 2007).

3. Results and Discussion

3.1. The k-□, SST, and RNG-k-□□model

One of the most prominent turbulence models, the (k-□) model, has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the model offers a good compromise in terms of accuracy and robustness (ANSYS, 2010).

The SST (Shear Stress Transport) model of Menter (1994) is an eddy-viscosity model which includes two main novelties i.e. (1) It is combination of a k-□ model (in the inner boundary layer) and k-□ model (in the outer region of and outside of the boundary layer) (2) A limitation of the shear stress in adverse pressure gradient regions is introduced.

The k -□ model has two main weaknesses; it over-predicts the shear stress in adverse pressure gradient flows because of too large length scale (due to low dissipation) and it requires near-wall modification (i.e. low-Re number dumping function/terms)(Gerhard and Richard, 1985).

The RNG k-□ model is an alternative to the standard k-□ model. In general it offers little improvement compared to the standard k-□ model. The RNG k-□ model is based on renormalization group analysis of the Navier-Stokes equations. The transport equations for turbulence generation and dissipation are the same as those for the standard k-□□model, but the model constants differ.

3.2. Viscous Resistance Coefficient

The total resistance of catamaran which is composed of two hulls place in close proximity, generally different from the sum of the resistance of the two hull when separately tested. This phenomenon is called interference. Interference arises from the change in the flow pattern as a result if the two bodies in close proximity. Total resistance of a ship is bredivided into viscous resistance (based on Reynolds number) and wave resistance (based on Froude number) components. Resistance equation shown in equation (1).

$$R_T(F_N, R_N) = R_w(F_N) + R_V(R_N) = R_W(F_N) + (1+k)(F_N)R_F(R_N) \quad (1)$$

The wave resistance (R_W) contains the inviscid component and the viscous resistance (R_V) includes the resistance due to shear stress (friction drag) and the viscous pressure component. In practice, the viscous resistance is usually estimated by using ITTC-1957 correlation line (C_F) together with a suitable form factor (1+k). Here a C_F is an approximation for the skin friction of a flat plate; and the form factor is used to calculate the three dimensional of ship hull (Utama, 1999). Formula of C_F can be written as equation (2).

$$C_F = \frac{0{,}075}{(\log_{10} Rn - 2)^2} \quad (2)$$

Total viscous resistance (C_V) can be measured by a wake transverse. In this case, form factor is calculated as equation (3).

$$(1 + k) = \frac{C_V}{C_F} \quad (3)$$

And relation between C_V and R_V can be written as equation (4).

$$C_V = \frac{R_V}{0{,}5\rho V^2 WSA} \quad (4)$$

By using equation 4, the result of C_V can be analyzed by numerical simulation using ANSYS CFX and the result shown in the **Table 1**. Increasing of Reynolds number in each test, both experiment and numerical, shown that the C_V gradually tend to decreasing. These conclude that the results of these research very strong and more accurate.

Table 1. Comparison viscous resistance coefficient between experimental test and numerical simulation by using differ turbulence models.

Re (10^5)	C_V Wind Tunnel (10^{-3})	C_V Numerical Simulation (10^{-3})		
		k-□	SST	RNG-k-□
2,894	8,761	8,995	8,893	8,940
3,473	8,418	8,689	8,581	8,641
4,052	8,150	8,457	8,354	8,386
4,457	8,000	8,335	8,223	8,298

The graphs of C_V for each turbulence models as shown in **Figure 4**. The most accurate turbulence models in predicting viscous resistance by using numerical simulations is Shear Stress Transport. The different of its viscous resistance with previous research (wind tunnel test) about 2,122%. Then following by RNG k-□ about 2,731% and finally by k□□ about 3,329%. The difference is caused by characteristic of each turbulence models in using equation in solving fluids problem. In the SST model the coeffcients are smoothly switched from k-□ values in the inner region of the boundary layer and k-□ values in the outer region. Turbulence could be thought of as instability of laminar flow that occurs at high Reynolds numbers (Re). Such instabilities origin form interactions between non-linear inertial terms and viscous terms in N-S equation. These interactions are rotational, fully time-dependent and fully three-dimensional. Rotational and three-dimensional interactions are mutually connected via vortex stretching. Vortex stretching is not possible in two dimensional space. That is also why no satisfactory two-dimensional approximations for turbulent phenomena are available. Furthermore turbulence is thought of as random process in time. Therefore no deterministic approach is possible. Certain properties could be learned about turbulence using statistical methods. These introduce certain correlation functions among flow variables. However it is impossible to determine these correlations in advance (Sodja, 2007).

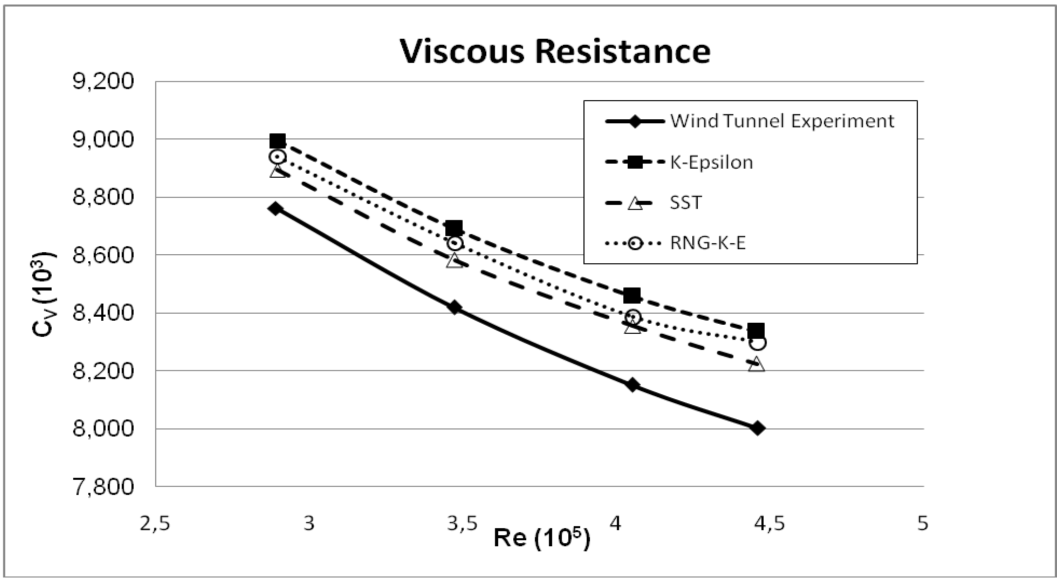

Figure 5. Viscous resistance experiment and numerical simulation

3.3. Contour of Pressure and Velocity Distribution

Contour of pressure and velocity provided in **Figure 4** and **Figure 5**. Pressure distribution shows the pressure along the hull. Figure 4 describes that maximum pressure occur on leading edge and trailing edges. **Figure 5** concludes that the maximum velocity occure on the midship of catamarans. This satisfy the Bernoulli principle. In fluid dynamics, Bernoulli's principle states that for an inviscid flow, an increase in the speed of the fluid occurs simultaneously with a decrease in pressure or a decrease in the fluid's potential energy. Bernoulli's principle can be derived from the principle of conservation of energy. This states that, in a steady flow, the sum of all forms of mechanical energy in a fluid along a streamline is the same at all points on that streamline. This requires that the sum of kinetic energy and potential energy remain constant. Thus an increase in the speed of the fluid occurs proportionately with an increase in both its dynamic pressure and kinetic energy, and a decrease in its static pressure and potential energy. If the fluid is flowing out of a reservoir, the sum of all forms of energy is the same on all streamlines because in a reservoir the energy per unit volume (the sum of pressure and gravitational potential ρ g h) is the same everywhere (Streeter,1966).

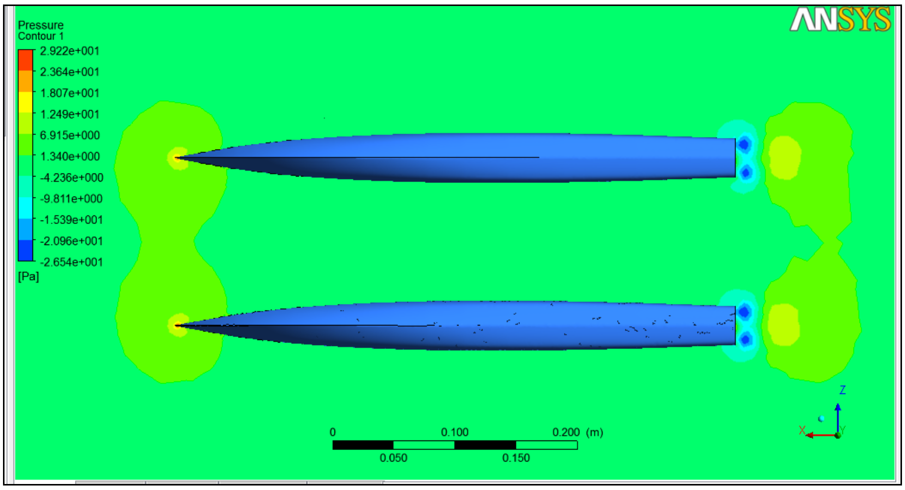

Figure 6. Contour of pressure distribution

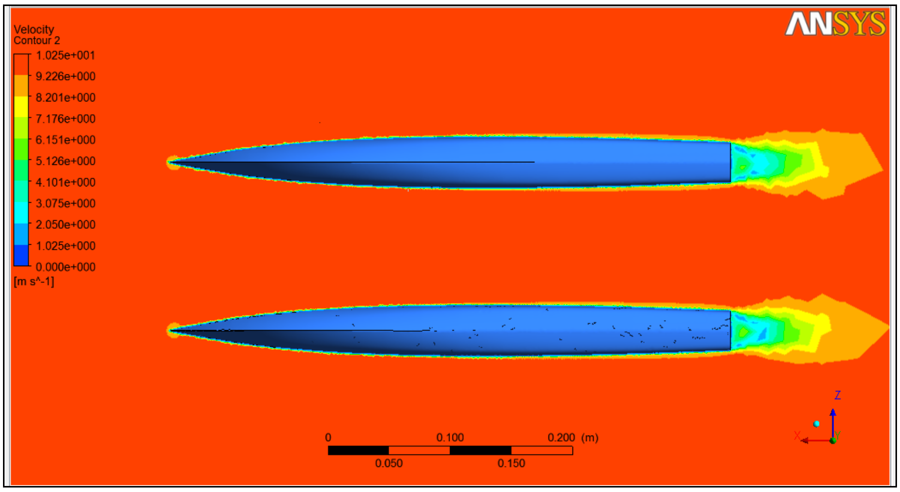

Figure 7. Contour of Velocity Distribution

4. Conclusion

In view of the analysis and validation process undertaken in this research work the following conclusions can be drawn:

1. Ansys CFX can be used as a tools to predicted numerical simulation to calculate viscous resistance of a catamaran.
2. The most accurate turbulence models in predicting viscous resistance by using numerical simulations is SST. The different of viscous resistance of numerical simulation comparing with the previous research (wind tunnel test) as follow: 3,329%; 2,122% and 2,731% respectively for k□□, Shear Stress Transport, and the RNG k-□.

Acknowledgment

This paper is dedicated to Dean of Faculty of Fisheries and Marine Science and Head of Department of Water Resources Utilization, University of Riau. This work was funded by both the Department and Faculty of Fisheries and Marine Science. We also would like to express sincere gratitude for the support of Research Agency of University of Riau.

References

ANSYS. (2010); Tutorial of CFX.

Armstrong, T. (2003); The effect of demihull separation on frictional resistance of catamaran, seventh international conference on fast sea transportation, FAST Ischia-Italy: Ischia, pp 22-30.

Couser, P. R., Molland, A. F., Armstrong, N. A. and Utama, I. K. A. P. (1997); Calm water powering predictions for high speed catamarans, Fast, Sidney, Australia.

Davidson, L. (2006): Turbulence modelling, http://www.tfd.chalmers.se/coct/comp_turb_model.

Gerhart, P, M. and Richard J. G., 1985; Fundamentals of fluid mechanics, Addison-Wesley Publishing Company, Inc., USA, 509-530.

Hutauruk R. M., Utama, I. K. A. P., Jamaluddin, A., Murdijanto, dan Djoni I M. A. (2011); Analisa distribusi tekanan dan kecepatan aliran pada kapal katamaran tanpa turbulen stimulator dengan uji terowongan angin,SENTA, 2011, ITS-Surabaya.

Jamaluddin, A., Utama, I. K. A.P. and Aryawan, W. D. (2010); Analisa eksperimen viscous form factor pada konfigurasi lambung, Prosiding Seminar Nasional Teori dan Aplikasi Teknologi Kelautan, FTK-ITS, pp 17-24.

Massey, B. and Smith. J. W. (2006); Mechanics of fluids, eight edition, solution manual. Taylor & Francis, London and New York.

Menter, F. R. (1994); Zonal two equation k-w turbulence for flows, AIAA Paper 93 – 290.

Mitchell, R. R. & Webb, M. B. (2008); A study of the base pressure distribution of a slender body of square cross-sectrion, AIAA Aerospace Sciences Meeting and Exhibit, 1-8.

Recktenwald, G. (2009): The k-ε turbulence model, Lecture note, Mechanical and Materials Engineering Department Portland State University, Portland, Oregon.

Sahoo, P. K., Doctors, L. R. and Pretlove, L. (2006); CFD Prediction of the wave resistance Of a catamaran with staggered demihulls. India: International Conference on Marine Hydrodynamics.

Sodja, J. (2007): Turbulence models in CFD, Faculty of Mathematics and Physics, Department of Physics, University of Ljubljana.

Streeter, V. L. (1966): Fluid Mechanics, example 3.5, McGraw–Hill Inc. New York.

Utama, I. K. A. P (1999); Investigation of the viscous resistance components of catamaran forms. UK: University of Southampton.

Utama, I. K. A. P. (2002); Studi hambatan kekentalan dan pengaruh bentuk. jurnal teknologi kelautan, Vol 1. No. 3.

Zuo, W. (2007); Introduction to computational fluids dynamics, FAU Erlangen-Numberg. JASS 05. St. Petersburg.