Aerodynamic Efficiency Analysis on Modified Drag Generator of Tanker-Ship Using Symmetrical Airfoil

Starida Moranova1)*, Rahmat Hadiyatul A., S.T.1), Indra Permana S., S.T.2)
1) Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung, Indonesia
2) PT Regio Aviasi Industri, Indonesia
*staridamaronova@gmail.com

Abstract. Time reduction of tanker ship spent in the sea should be applied for solving problems occurred in oil and gas distribution, such as the unpunctuality of the distribution and oil spilling. The aerodynamic design for some parts that considered as drag generators is presumed to be one of the solution, utilizing our demand of the increasing speed. This paper suggests two examples of the more-aerodynamic design of a part in the tanker that is considered a drag generator, and reports the value of drag generated from the basic and the suggested aerodynamic designs. The new designs are made by adding the NACA airfoil to the cross section of the drag generator. The scenario is assumed with a 39 km/hour speed of tanker, neglecting the hydrodynamic effects occurred in the tanker by cutting it at the waterline which separated the drag between air and water. The results of produced drag in each design are calculated by Computational Fluid Dynamic method.

1. Introduction
A lot of problems happened when distributing offshore and oil by a tanker ship from one land to another, these are the off-shedule distribution and the large amount of oil spill during shipping. If the time spent in the water could be reduced, hopefully there is such an efficient way to minimalized these problem. One solution to decrease the wasted time of tanker ship in the water is by reducing the drag that could be occurred around the body of the ship.

Angle-edged body like prism have a 1.14 coefficient of drag, larger compared to streamlined body which is 0.45 [11]. A part, located in the deck of the basic tanker design in figure 1, considered as drag generator regarding to its sharp edges of shape. This motivates the making of new design, smooth-edged shape, in order to reduce drag coefficient of the tanker.
The objective of this paper is to present the effect of aerodynamic shapes in the deck of a tanker ship to the total drag produced in the entire body of tanker. The drag generator will be replaced by a streamline-shaped airfoil to reach the need of aerodynamic efficiency. The drag generated on each design will be analysed using ANSYS software. Tanker ship model used is a basic oil tanker, created by Harris Nubly (downloaded from grabcad.com) [12], length 500 m.

2. Design and Simulation

Assumptions
Assumptions used in the design process and simulation are:

1. The basic model used is a 500 meter length tanker, scaled 1:500. The result calculated is not the real value, but this paper will only focus on the comparison of drag generated between each design.
2. Simulation will only analyse the part of tanker body which in contact with the air as fluid. The tanker is cut at the imaginary line which separate between surface contacted with air and water (next will be called waterline) [6].
3. The analysis uses a condition when the tanker ship floats on height of water 27.5 m above the lowest surface of tanker body. In this scaled 1:500 analysis, the waterline is 55 mm height.

Airfoil Selection
Thinner camber and closer camber position towards the leading edge of the airfoil will result to lower drag coefficient obtained [1]. These will be put in consideration of deciding airfoil parameter. We choose the lowest number maximum camber and maximum camber position as possible, which is zero. The thickness is then adjusted to the deck of the tanker. Finally, the selected airfoil is NACA 0024, which parameters are shown in table 1.

| Table 1. Airfoil Parameter |
|---------------------------|
| Max Camber                | 0%          |
| Max camber position       | 0%          |
| Max thickness             | 24%         |
Design Modelling
The design concept is to insert the geometry of airfoil and align the shape of the drag generator constraining the shape of NACA 0024 using CATIA V5. The three-view drawing of basic tanker ship design is shown in figure 2.

![Figure 2. Three view drawing of basic tanker design.](image)

There are two suggested designs. The first design uses airfoil aligning the whole drag generator surface (next will be mentioned as Design 1). Three view drawing of Design 1 is shown in figure 3 below.

![Figure 3. Three view drawing of Design 1](image)

The second design uses airfoil aligning each angle-edged shape, like three stairs on the deck (next will be mentioned as Design 2). Three view drawing of Design 2 is shown in figure 4 below.
Simulation
The simulation process begins by importing the CAD model as the geometry from CATIA V5 into ANSYS Workbench. The geometry will be given an enclosure as a boundary layer to define the airflow around the tanker. In analyzing the force acting on the vehicle in an unbound fluid domain, ideally an infinite large fluid domain is needed, which is not practicable in CFD and even in experiment. According to reference [2], [3], the assumption used for an unbound fluid domain is 20 times the length, width, and height of the object. Thus, the dimensions of enclosure surrounding the tanker are shown in table 2.

| Table 2. Dimensions of enclosure (fluid domain) |
|-----------------------------------------------|
| Length, y axis (m)                             | 20    |
| Width, x axis (m)                              | 4     |
| Height, z axis (m)                             | 2.5   |

The negative of z axis is not included, since the hydrodynamic effect is neglected in this analysis. The enclosure is then coincide the symmetry plane x-y, as shown in figure 3.
To start the simulation, the model needs to be meshed finely to get the most accurate result. The mesh is a collection of elements which serve to break-up the CAD model into smaller structural pieces - each piece (or element) can be thought of as a little spring with a three-dimensional spring equation, and the collection of elements represents a structure.

If all the above processes are done, setup the simulation with the configurations that needed to be changed, listed in Table 3.

| Table 3. Setup configuration |
|-------------------------------|
| Fluid Material                | Air Ideal Gas |
| Reference Pressure            | 0 atm         |
| Normal Speed                  | 10.833 ms⁻¹   |
| Turbulence                    | Medium (Intensity = 5%) |
| Opening Relative Pressure     | 1 atm         |
| Relative Average Static Pressure | 1 atm |

When the setup is finished, the model is ready for simulation using CFX Solver Manager. There are three types of models that are analyzed and each model is set equitably according to all the processes described above.
3. Results and Discussions
The final results of drag generated on each design are shown below:

- Basic tanker model: 12.767 Newton
- Design 1: 3.491 Newton
- Design 2: 2.021 Newton

Or the ratio of 63:17:10.

Validation
As shown in this paper, we use CFX from ANSYS 17.0 to solve and analyze the flow of the fluid surrounding the tanker ship that we modify. ANSYS 17.0 is a Computer Aided Engineering software that is used for analyzing variety of disciplines such as finite element analysis, structural analysis, computational fluid dynamics, and many others. In our study, we are using this software because it is commonly used to define the characteristics and properties of the air flow surrounding the model in a virtual environment which we want to analyze.

There are two evidences that can validate the method of utilizing ANSYS CFX that we use in this analysis. First, there are references with similar method that already proven and used worldwide with many varieties in terms of models [5], [6], [7]. Secondly we can see from the result of the CFX whether it is convergent or divergent.

From the result of simulations, the drag plots during iteration are shown in figure 7, 8, and 9.

![Figure 7](image)

**Figure 7.** Normal force (drag) generated on tanker body for basic design
4. Conclusions or Concluding Remarks

Conclusion

Results of calculation describes that the ratio of drag generated in Basic Design, Design 1 and Design 2 is 63:17:10. In other words, with the modified drag generator using symmetrical airfoil, the tanker ship with Design 1 configuration manages to gain 72.65% reduction from Basic Design (or 9.3 Newton), and with Design 2 configuration reaches 84.17% reduction from Basic Design (or 10.746 Newton). This percentage shows that the modified tanker ship can get less burden from the airflow which will lead to a more profitable cruise in a particular depth of water.

Acknowledgements

This work would have not been finished without the support of everyone that helped the authors along the way. We would like to thank Mr. Ony Arifianto, Ph.D. as our mentor. We would also like to thank FTMD ITB who allowed us to use the computational facility of the department and all of our friends from Aeronotika & Astronotika ITB such as Inggi, Rio, Fadli, Njet, Darda, Tasya, Horas, Bembi, and Agas for providing a full support in our work.
References

[1] Abbott, Ira H. 1959. *Theory of Wing Sections Including Summary of Airfoil Data*. New York: Dover Publications, Inc.

[2] Chin, Cheng Siong. 2013. *Computer-Aided Control Systems Designs: Practical Applications Using MATLAB and Simulink*. Boca Raton: Taylor & Francis Group.

[3] Patel, Rocky and Ramani, Satyen. Determination of Optimum Domain Size for 3D Numerical Simulation in ANSYS CFX. *International Journal of Innovative Research in Science, Engineering and Technology*, Vol. 4, Issue 6, June 2015.

[4] Priyadarshini, J.Sita, Abhinav, A.V.S., Chandra, B. Sharath, and Swathi, R.S. Use of Aerodynamic Lift in Increasing the Fuel Efficiency of Heavy Vehicle. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, Vol. 12, Issue 3 Ver. 1, July-August 2015.

[5] Latorre, Dr. Robert. Ship Hull Drag Reduction Using Bottom Air Injection. *Ocean Engineering*, Vol. 24, No. 2, November 1995.

[6] Choi, Jin-Keun, and Chahine, Georges L. Numerical Study on the Behavior of Air Layers Used for Drag Reduction. *28th Symposium on Naval Hydrodynamics*, September 2010.

[7] Padagannavar, Praveen and Bheemanna, Manohara. Automotive Computational Fluid Dynamics Simulation of a Car Using ANSYS. *International Journal of Mechanical Engineering and Technology (IJMET)* Vol. 7, Issue 2, March-April 2016.

[8] Tejaas, Vishan Ravi. Low Reynolds Number Flow Over Thick Symmetrical Airfoil with Dimple (NACA 0018). *International Journal of Scientific and Research Publications*, Vol. 6, Issue 9, September 2016.

[9] Mosaad, M. A., Gaafary, M.M., El-Kilani, H., and Amin, I.A.. Effect of Airfoil Camber on WIG Aerodynamic Efficiecy. *Port Said Engineering Research Journal*.

[10] [http://airfoiltools.com/airfoil/naca4digit](http://airfoiltools.com/airfoil/naca4digit)

[11] [https://www.grc.nasa.gov/www/k-12/airplane/shaped.html](https://www.grc.nasa.gov/www/k-12/airplane/shaped.html)

[12] [https://grabcad.com/library/simple-oil-tanker-vessel-2](https://grabcad.com/library/simple-oil-tanker-vessel-2)