Journal of Unmanned System Technology

Design of Diver Propulsion Vehicle Ganendra RI-1 using SolidWorks Flow Simulation

Hujjatul Anam†, Luqman Haris†, Aris Budiyarto†, Agus Budiyono‡

†Bhimasena Research, Technology and Development, Jatinangor, Indonesia

‡School of Engineering, Aerospace & Aviation Program, RMIT University, Melbourne, Australia

***Abstract*—** A diver propulsion vehicle DPV, also known as an underwater propulsion vehicle or underwater scooter is an item of equipment used by scuba diving and rebreather divers underwater to increase diving range. Simulation studies produce data in the form of comparison between DPV without tunnel, with side tunnel, bottom tunnel. The results are presented as data containing surface pressure, environmental pressure, cut plot speed, and 3 dimensional flow velocity and drag force. From the data analysis, we found that the side tunnel design provide the overall best performance to achieve desired vehicle specification.

***Keywords*—** DPV, flow simulation, underwater vehicle design.

Copyright©2017. Published by UNSYSdigital. All rights reserved.
DOI: 10.21535/just.v7i1.1040

I. INTRODUCTION

A. DPV Ganendra RI-1

DIVER propulsion vehicle DPV, also known as an underwater propulsion vehicle or underwater scooter is an item of equipment used by scuba diving and rebreather divers underwater to increase the range. The range is limited by the amount of breathing gas that can be done, the rate at which the breathing gas consumed under power, and the time limits imposed by the dive tables to avoid decompression sickness [1].

A DPV generally consists of a pressure-resistant waterproof casing that contains a battery-powered electric motor, which drives the propeller. The design should ensure that the propeller cannot endanger divers, diving equipment or marine life, the vehicle cannot be accidentally started or run away from divers, and remain neutral buoyancy when used under water. DPVS useful to extend the reach of divers who otherwise limited by the amount of breathing gas that can be done, the rate at which the breathing gas consumed under power, diver fatigue, and the time limits imposed by the dive tables to avoid decompression sickness. Common uses include cave diving and technical diving in which a vehicle is used to help move large equipment and make better use of time under water limited by the requirements imposed in diving decompression [1].

Corresponding author: Agus Budiyono
(e-mail: agus.budiyono@rmit.edu.au)

This paper was submitted on December 1, 2015; revised on December 21, 2015; and accepted on December 26, 2015.

B. Hydrodynamics and Bernoulli Equation

Hydrodynamic forces and moments related to the speed of the body and control surface deflections are given in this subsection. Damping of the DPV that moves in 6 DOF at high speed is nonlinear and coupled. Assumptions of the DPV is a non-coupled motion [2].

Hydrodynamic damping force equation:

$$D(u) = -\frac{1}{2}\rho C_D A|u|u \quad \textbf{(1)}$$

where CD, ρ, and A are coefficient of drag, density of the fluid (kg/m^3), and frontal area (m^2), respectively.

Hydrodynamics is one branch of science that studies fluid dynamics, especially the forces acting on an object that is in a fluid flow. Solving problems of hydrodynamics in general involves counting a wide range of properties of the flow that occurred, such as speed, pressure, time and temperature type, as a function of space and time. By studying the flow patterns that exist, it will be possible to calculate or estimate the forces and moments acting on the body found in the stream. In a simplified form, in general there are two forms of the Bernoulli equation. The first applies to non-compressed flow (incompressible flow), and the other is for a compressed fluid (compressible flow).

The above equation applies to non- compressed stream with the following assumptions:

- The flow is steady (steady state)
- There is no friction (inviscid)

In another form, the Bernoulli equation can be written as follows:

$$p_1 + \rho g h_1 + \frac{1}{2}\rho v_1^2 = p_2 + \rho g h_2 + \frac{1}{2}\rho v_2^2 \quad \textbf{(2)}$$

C. Drag

In fluid dynamics, drag (which is sometimes called the fluid resistance or drag) is the force that inhibits the movement of a solid object through a fluid (liquid or gas). Drag is composed of friction force, which acts parallel to the surface of the object, plus pressure force, which acts in a direction perpendicular to the surface of the object. For a solid body moving through a fluid, drag is the component of the resultant fluid dynamic force that acts in the direction of the movement of the body. Component perpendicular to the direction of body movement is seen as a lift. Here are some of the effects of drag on some form surfaces in the form of flat plate, cylinder, and a streamlined shaped in which the effect of the flow is very visible difference in Figure 2.

D. Computational Fluid Dynamics (CFD)

Computational fluid dynamics (CFD) are having a strong impact on both ship design and model test evaluation. The impact on ship model basins is occurring in a number of ways. As applications for CFD increase, the purpose of ship model tests is shifting from evaluation to validation. Also, systematic series investigations are becoming computer based rather than experiment based. Model tests required by the designer are becoming more selective and, when performed, include more detailed measurements. Also, extensive flow measurement experiments are conducted today for verification of CFD results [6]. Additionally, CFD is also defined as an analysis of the system as a problem of fluid flow, heat transfer, and other similar phenomena using computer simulation [7].

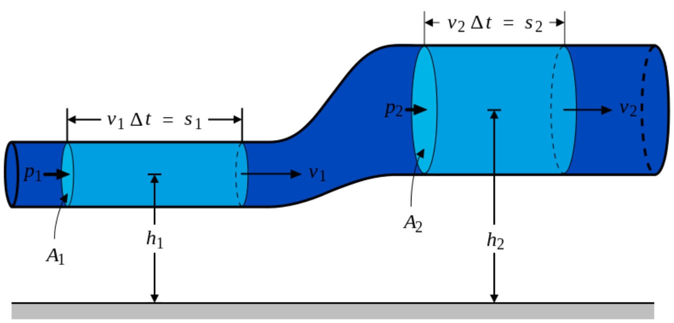
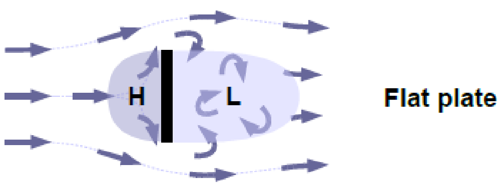
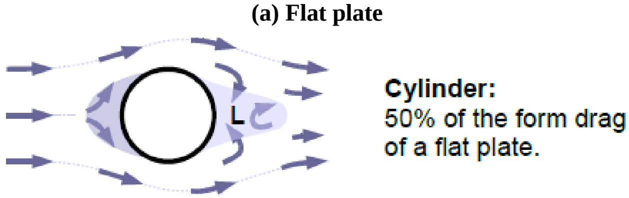


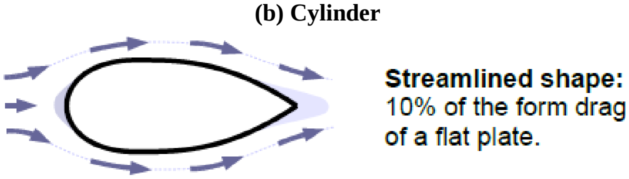
Figure 1 Diagram of the Bernoulli's law [3]



(a) Flat plate



(b) Cylinder



(c) Streamlined shaped

Figure 2 (a), (b), and (c) form drag pressure [4]

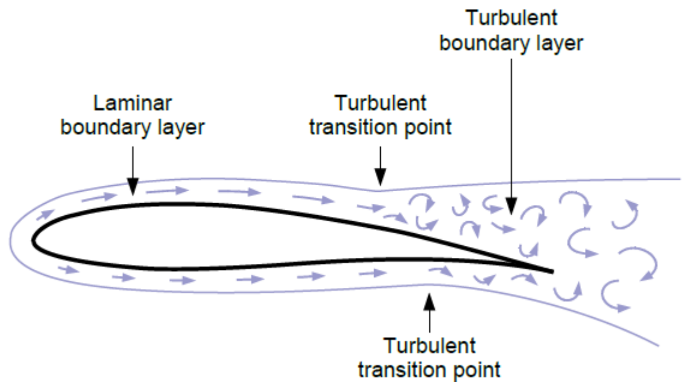


Figure 3 Boundary layer around a typical wing [4]

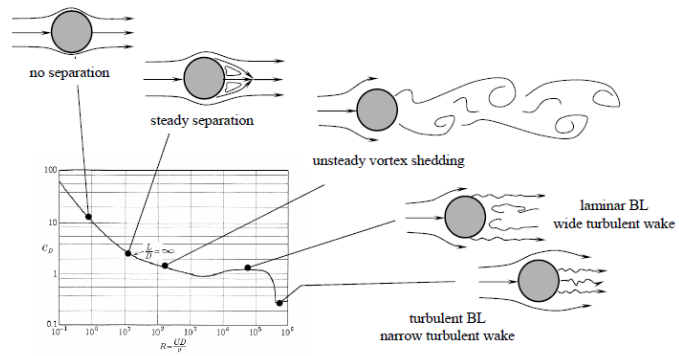


Figure 4 Drag on a smooth circular cylinder [4]

II. Diver Propulsion Vehicle (DPV) Design

The concept of the DPV to be created using 3D CAD design is without tunnel, side tunnel, and bottom tunnel. The aim of this design is to obtain the most efficient design of the DPV in order to reduce drag hence reduce the need of excessive engine power. The following DPV design with three models namely without tunnel, side tunnel, and bottom tunnel.

A. Without Tunnel Design of the Diver Propulsion Vehicle (DPV)

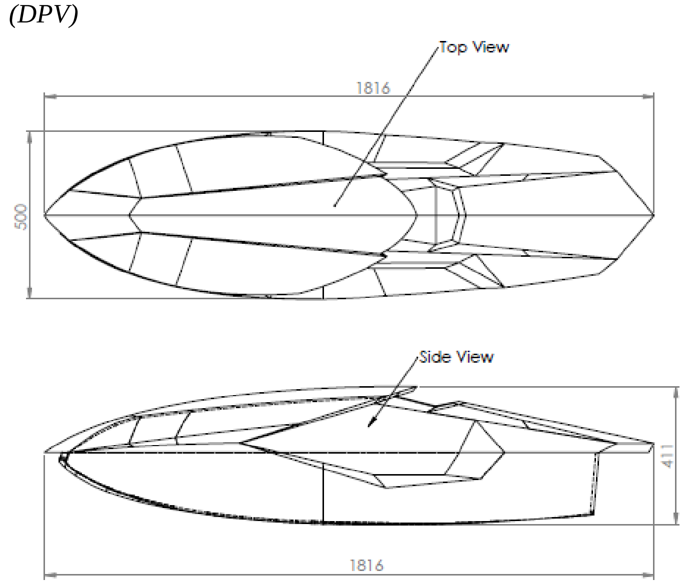


Figure 5 DPV design without tunnel

B. Side Tunnel Design of the Diver Propulsion Vehicle (DPV)

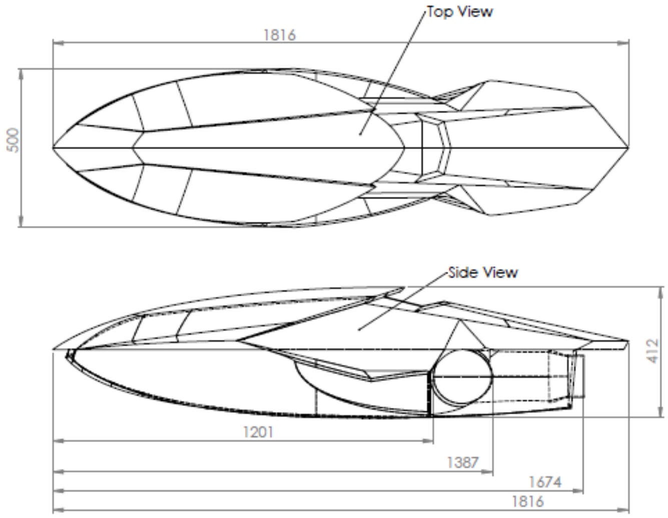


Figure 6 DPV design with side tunnel

C. Bottom Tunnel Design of the Diver Propulsion Vehicle (DPV)

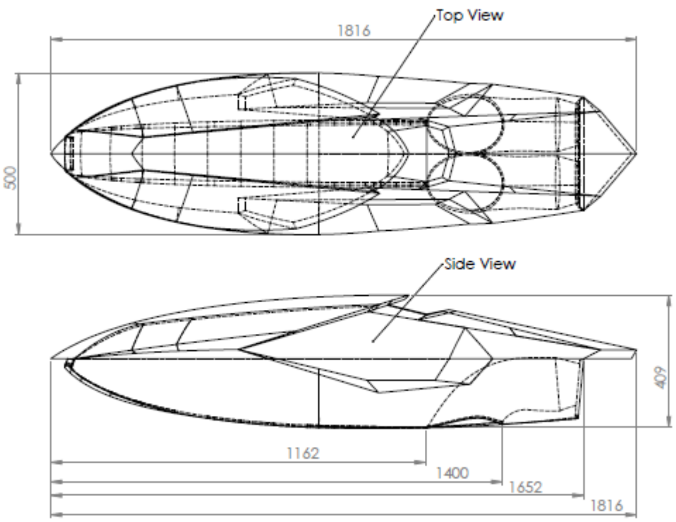


Figure 7 DPV design with bottom tunnel

III. Input Data

The software used in this simulation is SolidWorks 2014 by utilizing flow simulation toolbox.

A. Input data

1) Initial Mesh Settings

- Automatic initial mesh: On
- Result resolution level: 3
- Advanced narrow channel refinement: Off
- Refinement in solid region: Off

2) Geometry Resolution

- Evaluation of minimum gap size: Automatic
- Evaluation of minimum wall thickness: Automatic

3) Computational Domain

- Size

Table 1 Computational Domain

X min	-10.000 m
X max	10.000 m
Y min	-2.000 m
Y max	2.000 m
Z min	-5.000 m
Z max	5.000 m

4) Boundary Conditions

Table 2 Boundary Conditions

2D plane flow	None
At X min	Default
At X max	Default
At Y min	Default
At Y max	Default
At Z min	Default
At Z max	Default

5) Physical Features

- Heat conduction in solids: Off
- Time dependent: Off
- Gravitational effects: On
- Flow type: Laminar and turbulent
- Cavitation: Off
- High Mach number flow: Off
- Default roughness: 0 micrometer

6) Gravitational Settings

Table 3 Gravitational Settings

X component	0 m/s^2
Y component	-9.81 m/s^2
Z component	0 m/s^2

7) Default wall conditions: Adiabatic wall

8) Ambient Conditions

Table 4 Ambient Conditions

Thermodynamic parameters	Static Pressure: 4.00 atm Temperature: 298.00 K
Velocity parameters	Velocity in X direction: 3.000 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Intensity: 0.10 % Length: 0.004 m

IV. Simulation and Result

Simulations are carried out by the above input parameters, with a depth of 30 m or conditions of a pressure of 4 atm. The input velocity of 3 m/s is given for various design configurations: without tunnel, side tunnel, bottom tunnel. For the purpose of analysis, plots are generated in the form of forces and moments in the x, y, and z-axis, the pressure at the surface of the DPV Ganendra RI-1, the environment pressure, as well as the maximum speed that can be achieved by DPV Ganendra RI-1 for all three different designs.

A. Comparison of Surface Pressure: without Tunnel, with Side Tunnel and with Bottom Tunnel

In Figure 8, the surface pressure of all the three design shows a similarity. This is because the pressure is received in the same DPV body and slightly affected by velocity for laminar flow (Re < 2000).

B. Maximum velocity in cut plot

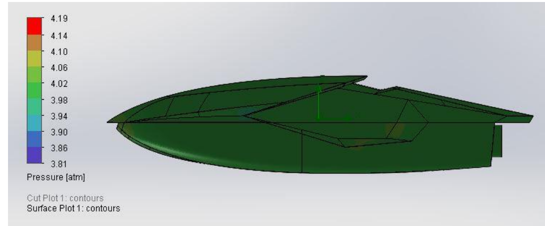
In the Figure 9, maximum velocity in cut plot that the comparison of different maximum velocity between 3 designs DPV is the maximum velocity without tunnel 3.858 m/s, the side tunnel 4.011 m/s, bottom tunnel 3.850 m/s so that the highest maximum speed is a side tunnel. The design generated the least turbulence and is the promising configuration to achieve desirable velocity specification.

C. Streamlines velocity in cut plot

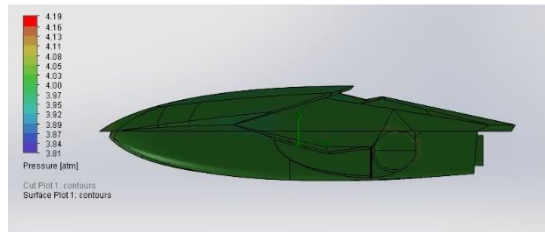
In the Figure 10, streamlines velocity in cut plot, shows that the simulation results for the streamlined body surface velocity around DPV has a different contour when seen from the results. For which without tunnel that will streamline the flow but the rear section of the body turbulent flow, thus making the flow becomes slightly streamlined. As for the tunnel beside a little stream formed turbulent because of the side tunnel is created, namely hydrodynamic flow that the function tunnel to direct the flow so as not to disturb the turbulent flow in velocity. For a bottom tunnel the condition looks a bit turbulent flow due to tunnel input and output of different tunnel indentation in the head means there is turbulent flow which can interfere with a maximum speed of DPV so that the direction of flow be streamlined.

D. Flow Simulation velocity

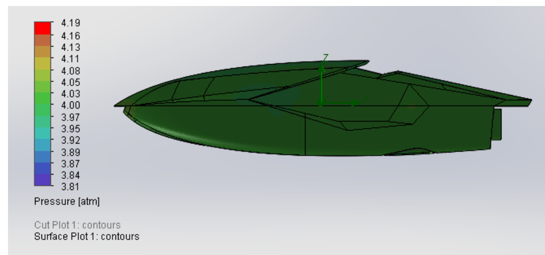
In the Figure 11, 3D Flow Simulation velocity, shows the 3 dimensional flow simulation results in which the result is to the conditions without tunnel that will streamline the flow of turbulent flow but there are indentations rear section of the body, thus causing the maximum speed reached was 3858 m/s because of the slowdown caused by the flow velocity turbulence in this part of the back of the body. Then to produce a side tunnel streamlined flow without any turbulent flow, this is because the flow has been directed by a tunnel without any distinction head because the input and output tunnel has no head difference so flow streamlined and maximum speed of 4.011 m/s highest base than design another. For a bottom tunnel the condition indicates that turbulent flow occurs in the tunnel, because of differences in input and output so that the head turbulence effects cannot be ignored, so for these three conditions can slow down the maximum speed of only 3850 m/s.



(a) Without tunnel

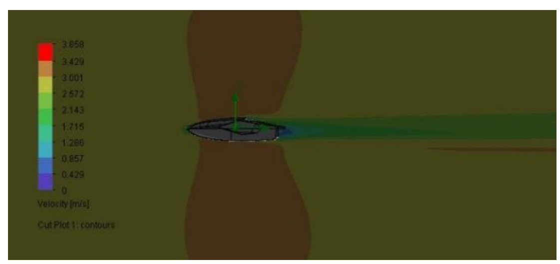


(b) With side tunnel

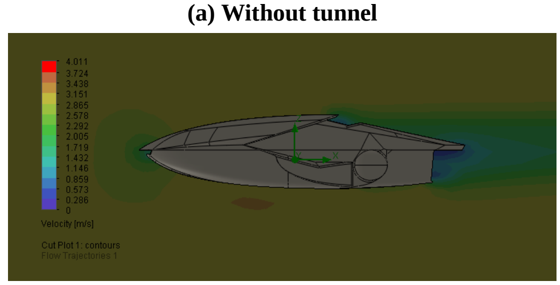


(c) With bottom tunnel

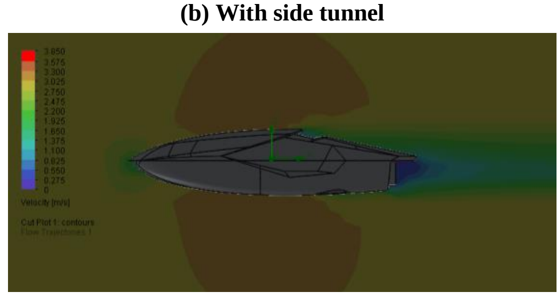
Figure 8 Surface pressure



(a) Without tunnel

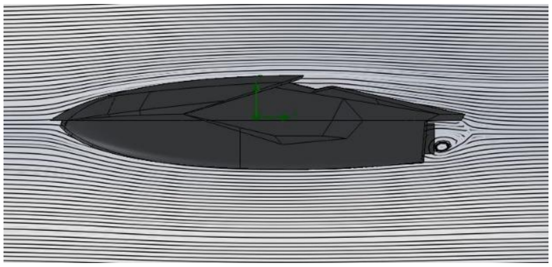


(b) With side tunnel

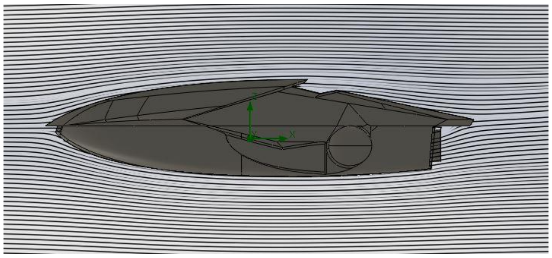


(c) With bottom tunnel

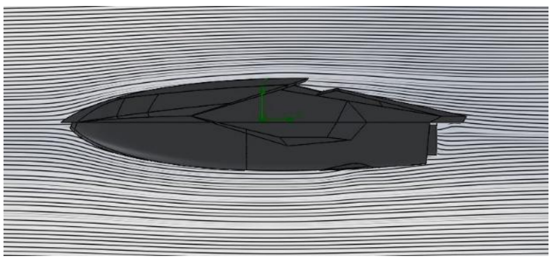
Figure 9 Maximum velocity in cut plot



(a) Without tunnel

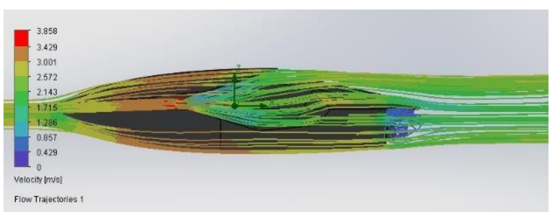


(b) With side tunnel

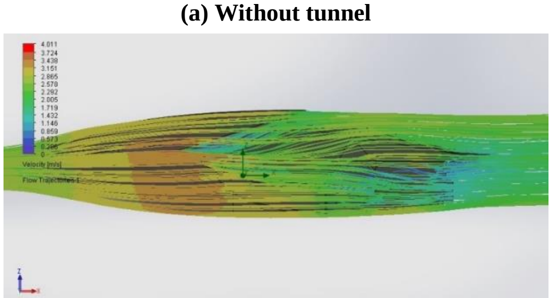


(c) With bottom tunnel

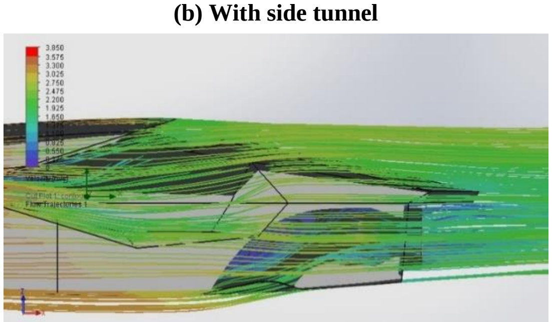
Figure 10 Streamlines velocity in cut plot



(a) Without tunnel



(b) With side tunnel



(c) With bottom tunnel

Figure 11 Velocity contour in the 3D flow simulation

E. Drag Forces

Drag is the result of a simulation of the flow at design conditions DPV without a tunnel, side tunnel, bottom tunnel which is a requirement force that inhibits the flow rate of the DPV, following the results of the third design in

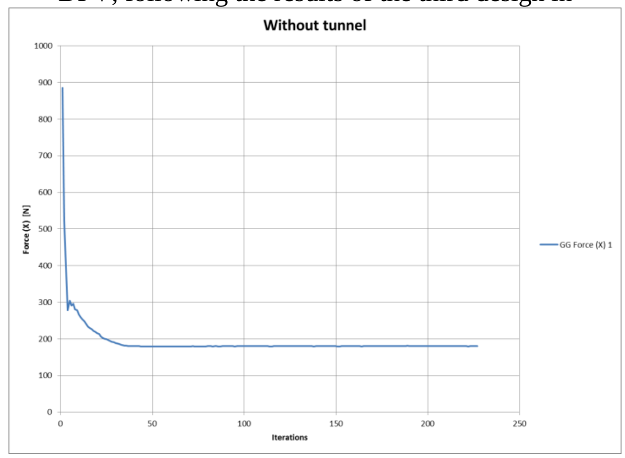


Figure 12 Drag force of without tunnel

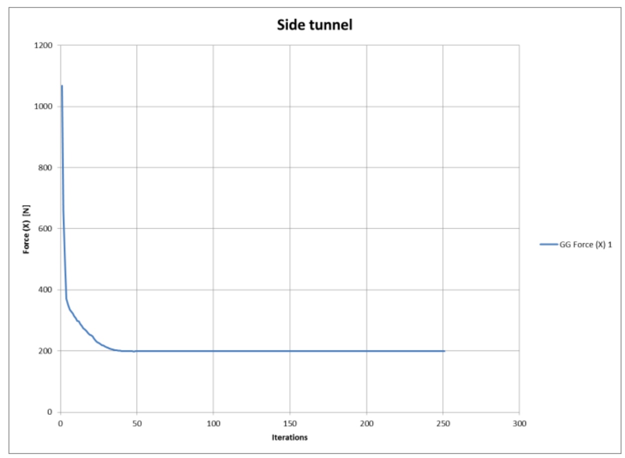


Figure 13 Drag force of Side tunnel

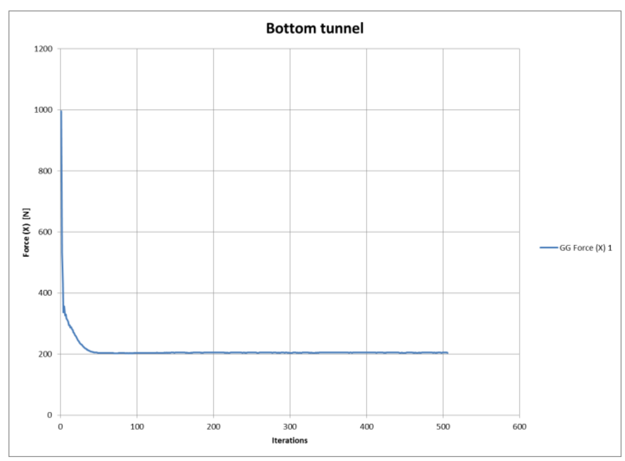


Figure 14 Drag force of Bottom tunnel

Table 5, with the condition without tunnel 180.40 N, side tunnel 199.58 N, and a bottom tunnel 204.43 N, and chart results for the drag iteration of Figure 12, Figure 13, and Figure 14. From these results, it turns out the biggest drag with a bottom tunnel design, this was due to turbulent flow occurs on the inside of the tunnel.

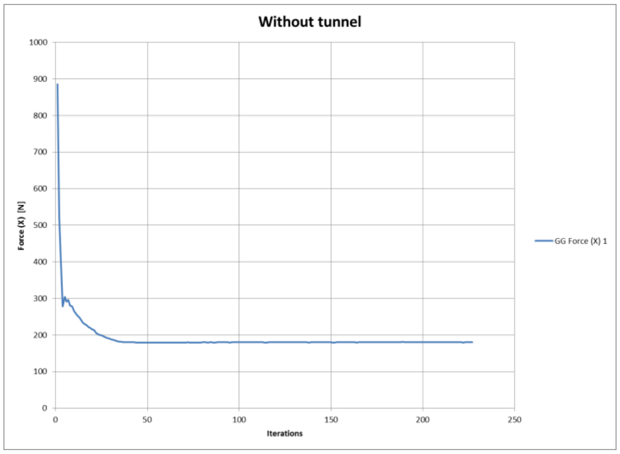


Figure 12 Drag force of without tunnel

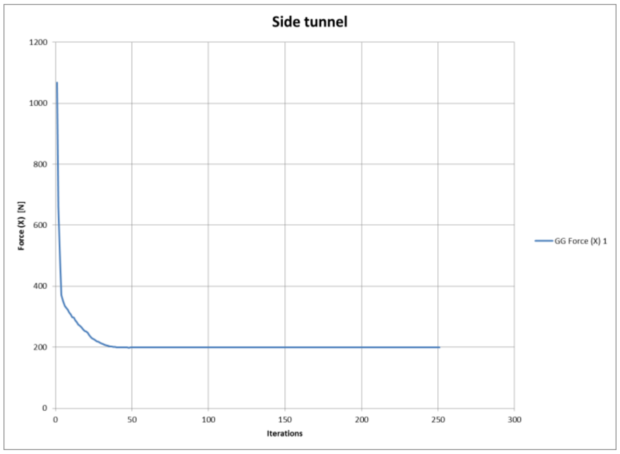


Figure 13 Drag force of Side tunnel

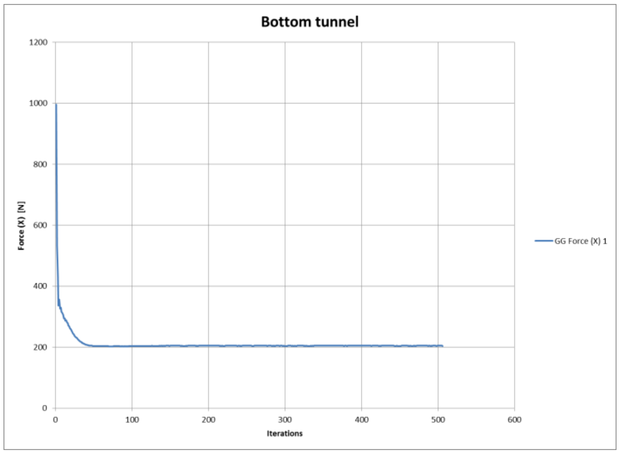


Figure 14 Drag force of Bottom tunnel

Table 5 Drag forces in *x*-axis

Goal name	Unit	Value		
		Without tunnel	With side tunnel	With bottom tunnel
Force (X)	N	180.4084	199.5876	204.4346

V. CONCLUSIONS

Based on data from the simulation results that the composition of the DPV Ganendra RI-1 design without a tunnel, side tunnel, and bottom tunnel are obtained in the form of surface pressure data, environmental pressure, velocity plot 2 dimensional and 3 dimensional, streamlined plot, and the data on the magnitude of the drag force. Of the three most efficient design after design simulation comparison is a side tunnel DPV Ganendra RI-1 because either plot of pressure, velocity and drag force, as well as a streamlined flow value efficient than others. From overall analysis, we conclude that the side tunnel configuration is best option for achieving design specification.

ACKNOWLEDGMENT

The authors would like to acknowledge the Bhimasena Research and Technology for providing facilities where the study in this work was conducted.

REFERENCES

[1] Kimura, S., Cox, G., Carrol, J., Pengilley, S., Harmer, A., (2012).” Going the Distance: Use of Diver PropulsionUnits, Underwater Acoustic Navigation, and Three—Way Wireless Comunication to Survey Kelp Forest Habitats’’.*In: Steller D, Lobel L, eds.Diving for Science 2012. Proceeding of the American Academy of Underwater Sciences 31st Symposium. Dauphin Island, AL*. Retrieved 2014-03-28.

[2] Chuan, T.S., 1999, Modeling and Simulation of the Autonomous Underwater Vehicle, Autolycus, Master Thesis, Department of Ocean Engineering, Massachusetts Institute of Technology, USA.

[3] https://en.wikipedia.org/wiki/Bernoulli%27s_principle#/media/File:BernoullisLawDerivationDiagram.svg.

[4] R.M, Bruce., F.Y, Donald., and H.O, Theodore, 2002, Fundamentals of fluid Mechanics Fourth Edition, John Wiley & Sons, Inc, USA.

[5] André Bakker. Lecture 11 – Boundary Layers and Separation Applied Computational Fluid Dynamics.

[6] W.B, Morgan., W.C, Lin., 1987, Computational Fluid Dynamics, Ship Design and Model, Proceedings of the 18th International Towing Tank Conference, Lobe, Japan, 18-24 Oct. 1987, Volume 2, p. 329.

[7] Versteeg, H.K., Malalasekera, W., 1995, An Introduction to Computational Fluid Dynamics The Finite Volume Method, Longman Scientific & technical, England.

[8] Solidworks, 2014, Solidworks Flow Simulation 2014 Tutorial.