https://doi.org/10.31436/iiumej.v26i3.3253

SAFETY ENHANCEMENT OF PORTABLE OIL SPILL SKIMMER (POSS) VIA COMPUTATIONAL FLUID DYNAMICS FOR LIQUID SLOSHING ANALYSIS

RPRAKASH RAMANATHAN [1,2], LOKMAN ABDULLAH[2*], MUHAMMAD HAFIDZ FAZLI MD FAUADI[2], MUHAMMAD SYAFIQ SYED MOHAMED[2], RUZAIDI ZAMRI[2], RAHIMAH ABDUL HAMID[2], ZAMBERI JAMALUDIN[2], MOHD SHAHRIEEL MOHD ARAS[3]

[1]Seatrium Limited, 80 Tuas South Boulevard, 637051, Singapore
[2]Fakulti Teknologi dan Kejuruteraan Industri dan Pembuatan, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[3]Fakulti Teknologi dan Kejuruteraan Elektrik, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

**Corresponding author: lokman@utem.edu.my*

(Received: 25 August 2024; Accepted: 31 May 2025; Published online: 9 September 2025)

ABSTRACT: The Portable Oil Spill Skimmer (POSS) was designed and developed due to several disadvantages of the current methods of Oil Spill Response and Recovery (OSRR). The POSS was designed as a complementary method to aid the OSRR tasks. However, during the POSS maneuverability testing, the POSS experiences instability when moving in different directions. The imbalance occurs when there is the presence of oil in the oil tank. Based on the literature study, the liquid sloshing effect was the reason why the POSS experiences instability. Thus, this research aims to analyse the impact of liquid sloshing in an oil tank and the implementation of baffles to reduce the effect. The analysis was conducted using SolidWorks Flow Simulation to simulate the liquid sloshing in the oil tank. The simulation was conducted in two situations, with and without baffles, to compare the results. According to the obtained results, with the implementation of 3 baffles, the sloshing effect was reduced to 392 N of torque force from 1195.43 N without baffles. The reduction was significant as the sloshing effect cannot be eliminated, thus the torque force of 392 N was enough to minimise the stability issue of the POSS.

ABSTRAK: Penapis Tumpahan Minyak Mudah Alih (POSS) telah direka bentuk dan difabrikasi kerana terdapat beberapa kelemahan kaedah semasa iaitu Tindak Balas dan Pemulihan Tumpahan Minyak (OSRR). POSS direka bentuk sebagai kaedah pelengkap bagi membantu operasi OSRR. Walau bagaimanapun, semasa ujian kebolehgerakan POSS, ia mengalami ketidakstabilan gerakan arah berbeza. Ketidakseimbangan ini berlaku apabila terdapat minyak dalam tangki minyak. Berdasarkan kajian, POSS mengalami ketidakstabilan disebabkan oleh kesan percikan cecair (liquid sloshing). Oleh itu, kajian ini bertujuan bagi menganalisis kesan percikan minyak (oil sloshing) dalam tangki minyak dan mengkaji keberkesanan pelaksanaan penyekat dalaman (baffles) bagi mengurangkan kesan percikan. Analisis dijalankan dengan menggunakan Perisian Simulasi Aliran SolidWorks bagi mensimulasikan percikan cecair dalam tangki minyak. Simulasi dijalankan dalam dua keadaan, dengan dan tanpa penyekat dalaman bagi membandingkan keputusan. Dapatan kajian mendapati melalui pelaksanaan 3 penyekat dalaman (baffles), kesan percikan telah berjaya dikurangkan kepada 392 N daya kilas (torque force) berbanding 1195.43 N tanpa menggunakan penyekat dalaman. Pengurangan ini adalah ketara kerana kesan percikan tidak dapat dihapuskan sepenuhnya. Oleh itu, daya kilas 392 N adalah cukup bagi meminimumkan isu kestabilan POSS.

https://doi.org/10.31436/iiumej.v26i3.3253

KEYWORDS: *Liquid Sloshing, Portable Oil Spill Skimmer, Oil Spill Response and Recovery, Baffles.*

1. INTRODUCTION

Oil spills have a range of immediate and long-term effects on the environment and wildlife. Environmental damage is a significant consequence, with oil spills causing harm to marine ecosystems, coastal areas, coral reefs, and wetlands. The causes of oil spills are diverse, including accidents during transportation, offshore drilling and production, refinery and storage accidents, and natural disasters. However, efforts are being made to prevent and respond to oil spills, including improved safety regulations, emergency response plans, and the development of technologies for containment and cleanup. Several technologies were invented to address this problem, such as mechanical techniques, chemical techniques, biodegradation, and open burning. Out of all these techniques, the mechanical technique is the most favoured and most effective as it does not alter the oil's physical properties. The mechanical method consists of booms and skimmers. However, this method has the disadvantage of containment issues, and skimmers are static or moved by external force. Thus, a portable oil spill skimmer was developed to complement the current Oil Spill Response and Recovery (OSRR) task.

The developed portable oil spill skimmer (POSS) has a self-propelling mechanism. The POSS could travel to limited-access areas, equipped with an oil tank to store the recovered oil. Meanwhile, existing oil spill skimmers are typically designed without an onboard oil tank, as the recovered oil is pumped into external storage. In addition, a limited study indicates the self-propelling features of the oil spill skimmer. Moreover, the developed POSS has an issue of instability during the maneuvering process. This instability of the POSS occurs when the oil tank is filled with the recovered oil. Thus, this study aims to analyse POSS's self-propelling capability and instability issues. The traveling speed of the POSS will be analysed both in simulation and experiment. Performing mechanical analysis is essential, as the findings are needed for various design processes, including electrical and electronics integration designs, controller designs, aerodynamic analysis, and industrial machine designs [1]–[3]. Computational Fluid Dynamics (CFD) is a numerical analytical approach used in mechanical engineering to approximate solutions to diverse engineering problems. For the simulation, a CFD analysis will be conducted to identify the drag and thrust forces produced. The travel distance of POSS will be used to perform the experimental analysis over time. In addition, the causes of POSS instability were investigated to enhance the fabrication process of POSS and address the issue. The designed POSS is equipped with a pair of propellers to maneuver it. Self-propelling propellers are selected as the propulsion system's propellers. USVs generally use a catamaran-like twin hull configuration for better roll stability compared to single hull designs [4]. Thus, the POSS was fabricated with a twin hull design to increase its stability.

However, during the experimental testing, the POSS still experiences instability when accelerating, decelerating, and turning. Also, the effect of sloshing impacts the structural integrity of the oil tank. From the testing, it is observed that the instability occurs when oil is in the oil tank. The instability of the POSS increases when the oil tank is partially filled. Thus, based on several literature reviews, it is found that the POSS undergoes an effect called liquid sloshing. Liquid sloshing refers to the movement of a free liquid surface within a partially filled tank due to external forces [5]. The research on the impact of liquid sloshing involves various engineering domains, such as aerospace, marine, oil tankers, and more [6]-[8]. According to research, the free oil liquid surface experiences different types of motion depending on the tank shape and type of disturbance [9]. The study found that higher excitation frequencies and amplitudes result in more intense sloshing, noticeable splashing waves, and significant breaking of the liquid surface [10].

Furthermore, research was conducted to analyse the dynamic pressure of the sloshing effect with various filling and motion conditions [11]. The result indicates that the partial filling of the oil tank has the most significant dynamic pressure that affects the safety of the fuel tank. Other researchers are also concerned about the safety of the fuel tank; thus, the mitigation of sloshing motion must be achieved [6], [12]-[15]. Several researchers have analysed the importance of implementing baffles to reduce the oil sloshing effect. Based on the research conducted, the incorporation of baffles in the oil tank shows significant improvement, reducing the impact of oil sloshing [16]-[21]. Baffles are usually installed in oil tank conFigureurations to mitigate or diminish sloshing and support the structures of the oil tanks [22-23].

Although the analysis of liquid sloshing has been conducted in various applications, to the best of the author's knowledge, it is less studied in the context of an oil spill skimmer. As the typical oil spill skimmer was developed without an oil tank, this analysis was not explored. Thus, in this research, the significant gaps in understanding the impact and ways to overcome it of oil sloshing are investigated to enhance the safety features of the POSS.

2. METHODOLOGY

2.1. Research Framework

Computational Fluid Dynamics (CFD) was conducted as a preliminary test to analyse the drag force exerted on the hull of the POSS [23]. Ansys Fluent was utilised to perform the analysis. Furthermore, the capability of the propellers was analysed to identify the thrust force produced. Thus, the result of the drag and thrust forces can be compared. The maneuverability of the POSS was experimented with, and the speed traveled was tabulated. Finally, the oil sloshing effect was also analysed by utilising CFD and investigating whether the implementation of baffles reduces the sloshing effect in the designed oil tank of a POSS.

2.2. Computational Fluid Dynamics (CFD) Analysis

2.2.1. CFD Analysis of Hull

ANSYS Fluent is a cutting-edge computer tool that allows you to model fluid flow in complicated shapes. It enables the user to enter several models for mesh creation of various forms and refine or coarsen the mesh of the model based on the flow solution. Figure 1 shows the steps of simulating the hull's resistance. This research utilizes SolidWorks software to create a 3D model, which is then simulated in ANSYS Fluent to calculate the resistance when traveling over the water surface. Several steps were conducted in this research, as follows:

- Step 1: Draw a 3D model of the hole in SolidWorks.

  The hull's dimensions are measured using a tape measure and a vernier caliper. Next, the camera captured the top, side, and front views of the peculiar form of the hall at the gun. Thus, the solid modeling of the hull was done by following the previously measured dimensions.

- Step 2: Export to the NSYS program

  The 3D model of the file is imported to ANSYS Design Modeler in the SAT format.

- Step 3: Generate domains for hull

The domain is traced around the hull, with the stern hall at the start of the plane. There are two types of domains: a global domain and a subdomain. The global domain ship is a cuboid with dimensions of 4 metres in length, 0.5 metres in width, and 1.5 metres in height, while the subdomain has dimensions of 4.22 metres in length, 0.6 metres in width, and 0.17 metres in height.

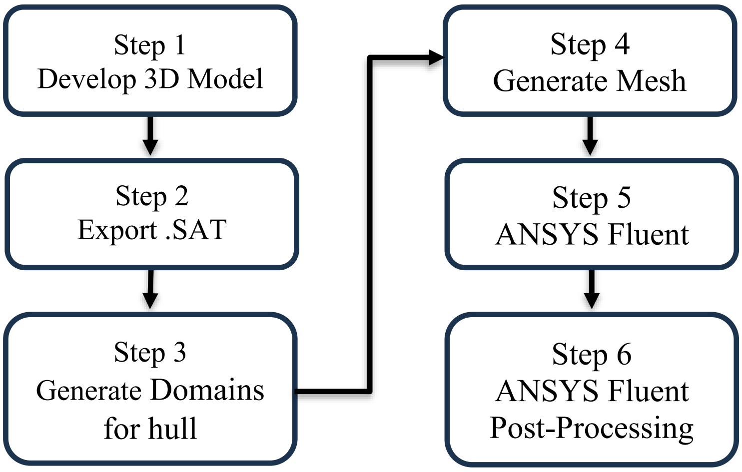


Figure 1. Steps of simulating the hull's resistance.

- Step 4: Generate mesh

The development of fine mesh is a crucial stage in the modeling of hull resistance because it allows the solution to generate more precise results. A suitable machine could produce the computational time, but it mainly relies on the computer's CPU. The primary domain's intake is positioned to face the bow, while the hull's outflow is positioned to face the stern. Body sizing on the global domain and face sizing on the hull are the two-mesh sizing employed. The meshing parameters are listed in Table 1.

Table 1. Meshing parameters of the simulation

Mesh Properties	**CFD**
Solver Preference	**Fluent**
Element Order	**Linear**
Element Size	**Default (225.92 Mm)**
Body Sizing	
Type	**Body Of Influence**
Element Size	**30.0 Mm**
Growth Rate	**Default (1.2)**
Face Sizing	
Type	**Element Size**
Element Size	**15.0 Mm**
Proximity Size Function Source	**Faces And Edges**
Statistics	
Nodes	**102612**
Elements	**540272**

- Step 5: ANSYS Fluent setup

In this step, the solution includes a pressure-based solver type, an absolute velocity formulation function, a constant time condition, and a gravitational acceleration. In addition, the viscous model is set to k-epsilon for the model set. Finally, the simulation applied the near-wall treatment of typical wall functions and a realisable k-epsilon model. The realisable k-epsilon model has an advanced formulation for turbulence compared to the k-epsilon model, thus providing more accurate results.

The multiphase model is otherwise configured to fluid volume with open channel flow and an implicit body first formulation. Water and air are the two fluid materials used in the material settings, with a water density of 1000 kg/m3. They were chosen for this multiphase model because the hull is partially immersed in the body of water where the procedure is carried out. Fiberglass is chosen as the solid material for the research.

For boundary conditions, five velocities in the inlet towards the bow at 1.39 m/s (5 km/h) to 1.95 m/s (7 km/h) were evaluated against the hull to determine the force acting on it. Standard initialisation, automatic time-step approach, 0.5-time scale vector, and 250 iterations are used in the solution.

- Step 6: ANSYS Fluent Post-Processing

  In Fluent Post Processing, the iso-surface and vector are generated by the force acting on the hull and can be determined using the calculator function. This is because the resistance on the hull is similar to the resistance when the hull is moving forward.

2.2.2. CFD Analysis of BlueRobotics T200 Propeller

The self-propelling propeller is analysed using CFD analysis in this research to confirm the data supplied by the propeller performance chart. In addition, the thrust force generated at a given rotational speed may be calculated using the propeller's CFD analysis. Finally, the results of the CFD study will be compared to the thruster data given by BlueRobotics to support the thruster selection for the portable oil spill skimmer. The ANSYS Fluent T200 propeller is detailed here.

- Step 1: Conversion of the 3D model of BlueRobotics T200 to the (.STEP) format.
- Step 2: Export to the ANSYS Program in the SAT format.
- Step 3: Generation of domains for propeller

  The propeller is centred at the top of the page, and the domain is drawn around it. The enclosure and the revolving body are the two domains. The rotating body is a cylinder with an 85 mm diameter and a 32 mm thickness covering the propeller. For the examination of the spinning body, enclosure, and propeller, there are two subtracting Booleans.

Table 2. Meshing properties of propeller analysis

Mesh Properties	CFD
Solver Preference	**Fluent**
Element Order	**Linear**
Element Size	**Default (30.619 mm)**
Body Sizing	
Type	**Element Size**
Element Size	**20.0 Mm**
Capture Curvature	**No**
Statistics	
Nodes	**102612**
Elements	**540272**

- Step 4: Mesh Generation

https://doi.org/10.31436/iiumej.v26i3.3253

The propeller is subjected to face sizing with a 20 mm element size for this study. On the Z-axis, the propeller, intake, and outlet rotate in a clockwise direction. The default mesh size for the spinning body and enclosure is 30.619 mm. Table 2 shows the meshing characteristics.

- Step 5: ANSYS Fluent setup
    - The solution in this work uses a pressure-based solver type, an absolute velocity formulation function, a transient time condition, and gravitational acceleration. Additionally, the viscous model is set to the k-epsilon model in the model settings.
    - The water used in the CFD analysis has a 1000 kg/m3 density.
    - The rotating body zone is set to mesh in the cell zone condition setting. The motion end propeller is configured to rotate at 3000 RPM, and the intake is subjected to a velocity of 3.8 m/s for the boundary condition.
    - It is set to hybrid initialisation for the solution, which means the software guesses the variables. With 100 time steps and a maximum iteration of 10, the time step size is set to 0.05 seconds.
- Step 6: Post Processing

  Streamline is created as the container to represent the flow pattern in Fluent Post Processing. The propeller's force may be calculated using the calculator tool. When the propeller rotates in its blade direction, the created force is the benefit of distress.

2.3. Manoeuvrability Testing

The analysis of maneuverability was conducted using a series of tests at different distances (m). Table 3 shows the expected results of the analysis. A Knots unit was used as the POSS traveled on the water surface. The speed of the portable oil spill skimmer varies due to the external factors of wind and water currents.

Table 3. Expected result of maneuverability testing

Test	Distance (m)	Time Taken (s)	Speed (m/s)	Knots
1	30			
2	60			
3	100			
	Average speed			

2.4. CFD Analysis of Oil Sloshing

The study framework provides a comprehensive explanation of the methodology employed for liquid analysis. The liquid sloshing investigation will use SolidWorks Computational Fluid Dynamics (CFD) simulation. Table 4 displays the sequential procedure of the computational fluid dynamics (CFD) simulation. The following instructions must be followed to conduct this simulation.

https://doi.org/10.31436/iiumej.v26i3.3253

Table 4. Step-by-step process of the oil sloshing analysis

Pre-processor
1. Define Solid Model Geometry
2. Define Gravity Axis and Motion Axis
3. Define Gases and Fluid Types
4. Define the Initial Condition and Substance Concentration
5. Global Mesh
Solution
6. Define the Global Goals
7. Select Transient Explorer Parameters
8. Run the Simulation
Postprocessor
9. Plot, View, and Export the Results
10. Compare and Verify the Results

2.4.1. Define Solid Model Geometry

The basic steps were conducted in the pre-processor. The solid model geometry from the SolidWorks software was converted into the Ansys software. Then, the element type was defined according to the analysed element. Following the conversion of the solid model, the appropriate element type was determined based on the characteristics and requirements of the specific analysis being conducted. This step ensures that the model is accurately represented and analysed using the designated elements within the Ansys software.

2.4.2. Gravity Axis and Motion Axis

To obtain precise data, it is necessary to establish the gravity and motion axes with the correct value for the simulation. The sequential process for defining the gravity and motion axis is as follows:

- The oil tank's gravity must be specified on the y-axis. Gravity has a constant value of -9.81 m/s2 since it exerts a continuous force towards the Earth's center.

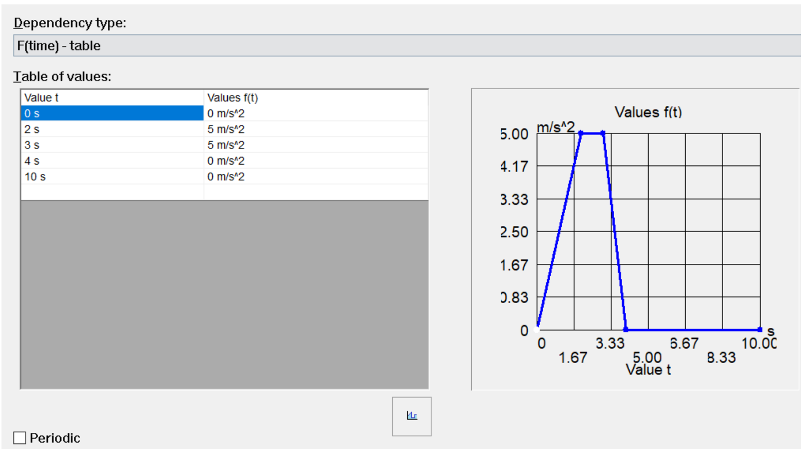


Figure. 2. Motion defined with a time variable

- The X-axis is the motion axis that indicates the direction of movement for the oil spill skimmer. The movement is defined by a variable that changes over time, as seen in Figure 2. The oil spill skimmer starts operating from a fixed position. The oil spill skimmer starts accelerating at 2 seconds, increasing at a constant rate of 5 m/s^2 until it reaches 3 seconds.

https://doi.org/10.31436/iiumej.v26i3.3253

The oil spill skimmer decelerates and stops completely after 4 seconds, with an acceleration of 0 m/s^2. The oil spill skimmer stops and stays still for 10 seconds.

2.4.3. Gases and Fluid Types

Air and propane are selected as the components for the simulation, namely in the states of gases and fluids. The oil tank consists of two main parts: the lower section is for liquid propane, and the upper section is for gases. Propane acts as a replacement for oil. Figure 3 illustrates the selected gases and fluids.

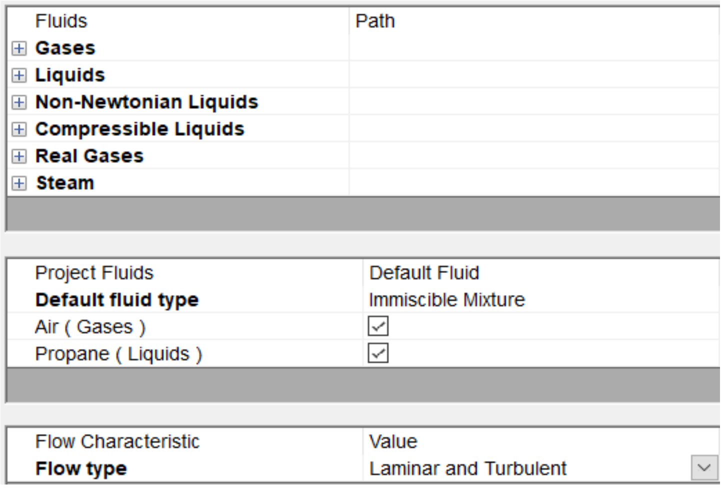
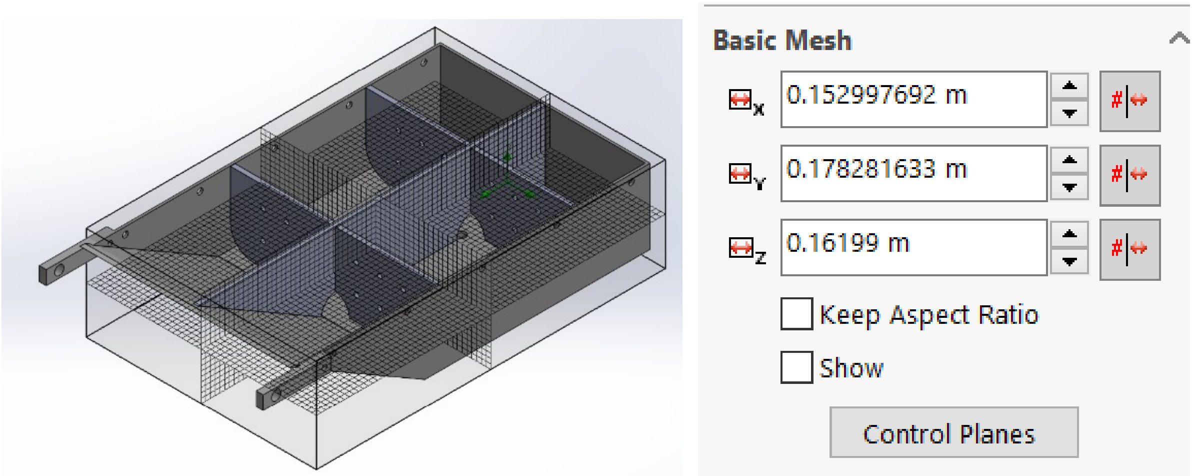


Figure 3. Selected gases and fluids

2.4.4. Meshing

Before simulating the design, meshing must be applied. The configuration for meshing in the flow simulation is displayed in Figure 4(a). The basic mesh refinement level is multiplied by two to obtain more accurate results, as shown in Figure 4 (b). Increasing the level of meshing detail directly correlates with an increase in the accuracy of the resulting data.



(a) (b)

Figure 4. (a) Meshing detail of oil tank, (b) the basic mesh values

3. RESULT AND DISCUSSION

3.1. Result of CFD Analysis on Hull

The hull design of the Portable Oil Spill Skimmer was analysed using ANSYS Fluent. The findings were obtained after 250 iterations to achieve satisfactory solution convergence. Increasing the number of iterations leads to more precise computational fluid dynamics (CFD) findings. Five distinct examples have been examined, and the outcomes are presented in Table 5. According to the CFD analysis, the hull experiences a drag force of 2.7312 N when moving forward in the water at a speed of 1.39 m/s, or approximately 5.0 km/h. Figure 5 shows the increasing trend and the relationship between the drag force and the hull speed. Meanwhile, an increase of 2 km/h from 7.0 km/h produces a drag of 13.5542N on one hull, showing a rise of 396.27%, almost 5 times the drag force value at a speed of 5.0 km/h. These drag forces show increments when the speed of the Portable Oil Spill Skimmer increases.

Table 5. The relationship of drag force against the hull speed

Run	Velocity (m/s)	Velocity (km/h)	Drag Force of one hull (N)	Drag Force on two hulls (N)
1	1.39	5	2.7312	5.4624
2	1.53	5.5	3.5844	7.1688
3	1.67	6	9.2366	18.4732
4	1.81	6.5	12.1222	24.2444
5	1.95	7	13.5542	27.1084

The water flow pattern was shown in Figure 6 when moving forward at the speed of 1.95 m/s. The wake velocity generated at the stern of the hull increases with the hull speed. In addition, a minor wave hits the hull's bow at 1.39 m/s, becoming more aggressive when the speed increases to 1.81 m/s and 1.95 m/s. In this case, the skimmer utilizes a catamaran with two hulls side by side, resulting in a total drag force of 27.1084 N. Hence, the two thrusters must generate forward thrusts of more than 27.1084 N to achieve the hull speed of 1.95 m/s.

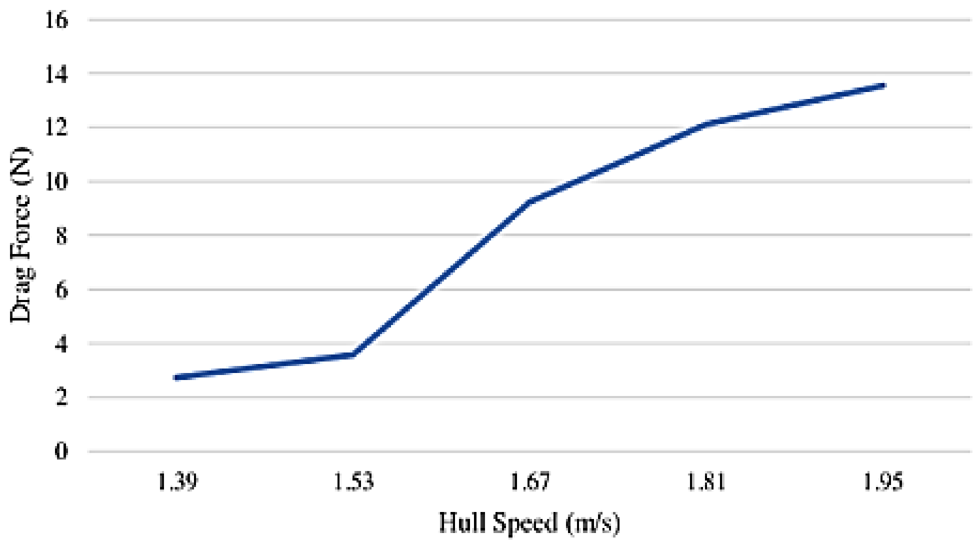


Figure 5. Relationship between the drag force and hull speed

https://doi.org/10.31436/iiumej.v26i3.3253

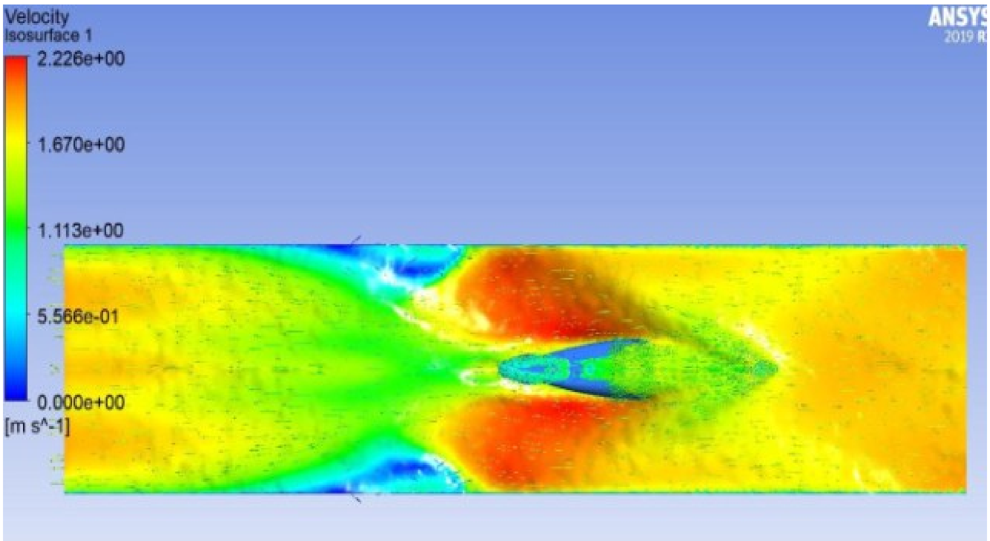


Figure 6. Flow pattern of water at a speed of 1.95 m/s

3.2. CFD Analysis on BlueRobotics T200 Propellers

From the CFD analysis that has been performed on the self-propelling propeller, it is found that the propeller can generate a thrust force of 36.8246 N at 3000 rpm and an inlet velocity of 3.8m/s. This thrust value is obtained from the ANSYS Fluent Post Processor of results, namely the function calculator, which is the function of thrust force, and the location is the propeller selection, as shown in Figure 7. In addition to that, the pressure contour in Figure 8 shows the maximum pressure of $5.109e^{+04}$ Pa and a minimum pressure of $-1.284e^{+05}$ Pa on the propeller blades.

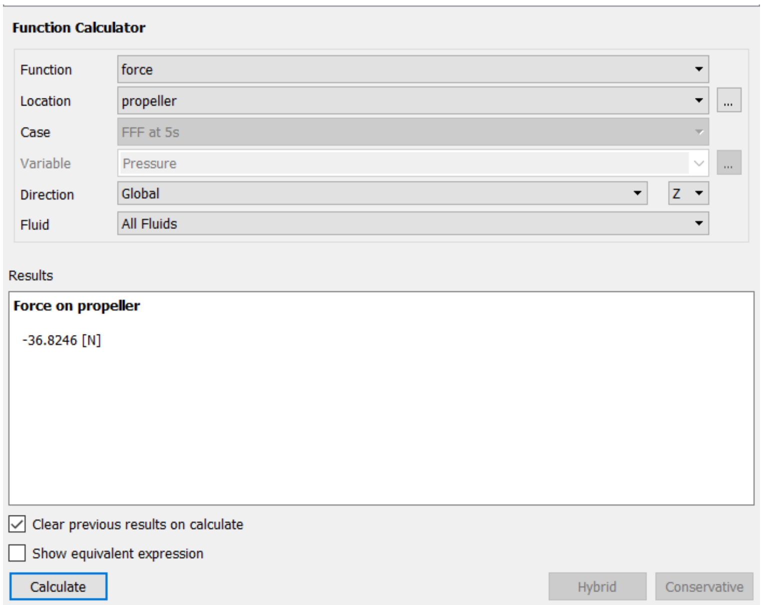


Figure 7. The value of thrust force exerted on the propeller.

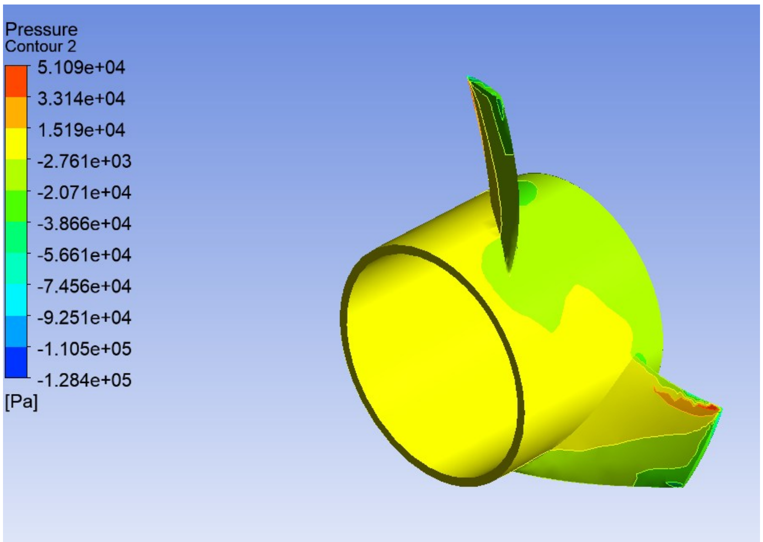


Figure 8. The pressure contour on the propeller blades

Furthermore, data provided by the propeller manufacturer, BlueRobotics, reveals that the T200 model can generate a maximum thrust of 3.71 kg or 36.383 N. This thrust level is achieved when the thruster rotates at a speed of 2995 rpm and is powered by a 12 V supply. Additionally, the results of the Computational Fluid Dynamics (CFD) simulation conducted on the propeller indicate a slight difference of 1.21% in the thrust value, comparing the supplier data (36.38 N) with the CFD result (36.4 N). This suggests that the setup parameters used in the CFD calculations are accurate and reliable. Based on the design of the Portable Oil Spill Skimmer, there are two T200 thrusters installed at the rear part of the frame to facilitate its movement on water. With these thrusters, the combined thrust force is 72.8 N, which is capable of overcoming the hull drag of 27.1084 N to propel the vehicle up to a speed of 1.95 m/s or even higher, provided the water conditions are ideal. These data prove that the T200 thrusters used on the Portable Oil Spill Skimmer are reliable.

3.3. Result of POSS Manoeuvrability

The portable oil spill skimmer consists of propellers capable of self-propulsion. Figureure 9 shows the portable oil spill skimmer manoeuvrability testing at Tasik Ayer Keroh. The testing was conducted by measuring the time the portable oil spill skimmer took to reach the desired distance. For instance, the distance for the portable oil spill skimmer is 100 meters; thus, the time taken to get the 100-meter distance is observed. A total of 3 tests were conducted with different distances of 30, 60, and 100 meters. The data collected for the manoeuvrability testing is shown in Table 6. Equation 1 shows the conversion from meters per second (m/s) to Knots. The result indicates that the POSS was able to be manoeuvred at a speed of 3.72 Knots, which adheres to the typical speed for an unmanned surface vehicle (USV) of 3 Knots and above [23]-[25]. The water currents and wind influenced the speed of the POSS during the experiment process.

$$\text{Knots} = \text{meters per second} \times 1.943844 \tag{1}$$

where the speed in knots is equal to the speed in meters per second multiplied by 1.943844, then we can obtain Knots = 1.88 × 1.943844 = 3.65.

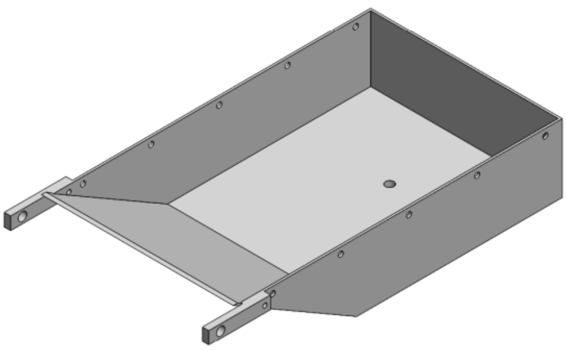
Figure 9. Portable oil spill skimmer manoeuvrability testing at Tasik Ayer Keroh

Table 6. The data collection of the POSS speed

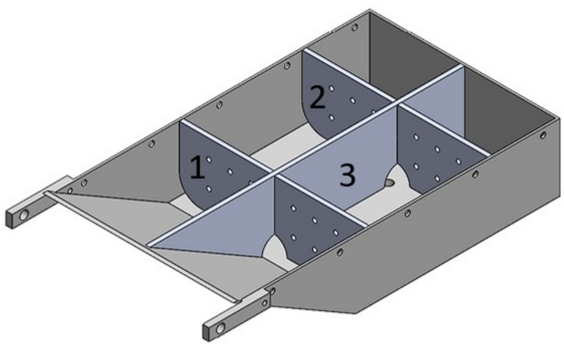
Test	Distance (m)	Time Taken (s)	Speed (m/s)	Knots
1	30	16	1.88	3.65
2	60	31	1.94	3.77
3	100	52	1.92	3.73
Average speed			1.91	3.72

3.4. Result of Oil Sloshing

The simulation was conducted on the oil tank design to prove that oil sloshing could be reduced. The design shape of the oil tank without a baffle is shown in Figure 10(a), while the oil tank design with a baffle is shown in Figure 10(b). The design was developed with a size of 773 x 450 mm, and a total of 3 baffles was implemented. Based on Figure 10(b), baffles 1 and 2 were designed to be parallel to each other, with baffle 3 positioned in the middle. Baffle 3 was explicitly designed to reduce the oil sloshing during left or right turns, and baffles 1 and 2 were designed to reduce the oil sloshing during acceleration and deceleration.



(a)



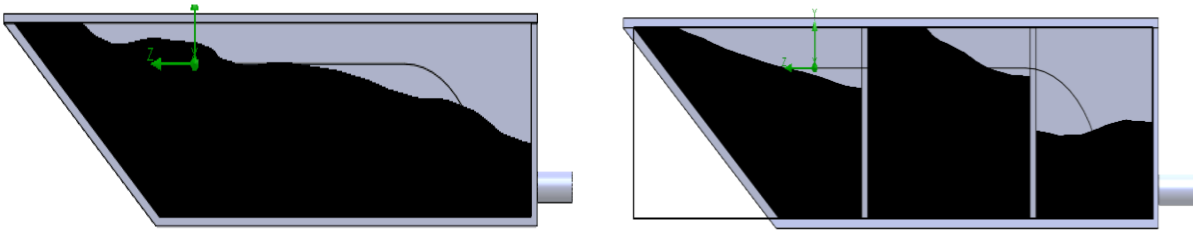
(b)

Figure 10. The design shape of the oil tank (a) without baffles, (b) with baffles

3.4.1. Liquid Sloshing Analysis on the Designed Oil Tank with and without baffles

The simulation was conducted using an oil tank design with and without baffles to investigate the difference in the impact of oil on the tank. The simulation measured and collected the torque and force acting upon the oil tank. Based on Figure 11 (a), it was observed that the time taken for the oil to stabilise in the oil tank without a baffle was more than 10s. The maximum torque exerted on the oil tank was 1195.43 Nm, as shown in Figure 11 (a). This indicates that the POSS has a risk of accidents and overturns due to the ample torque and force exertion.

Figure 12 (b) shows the liquid sloshing effect in the designed oil tank with baffles. The sloshing effect lasts only 5 seconds before the oil stabilises in the tank. This indicates that implementing a baffle in the oil tank has a reduced impact on the performance of the designed oil tank. The effect of the torque exerted on the oil tank is only 392 Nm, a reduction of 67.21%, minimizing the risk of the POSS overturning, as shown in Figure 12(b). This proves that implementing several baffles can eliminate the pitching and heaving effect and provide a safer working environment.



(a) (b)

Figure 11. Liquid sloshing effect in oil tank without baffle

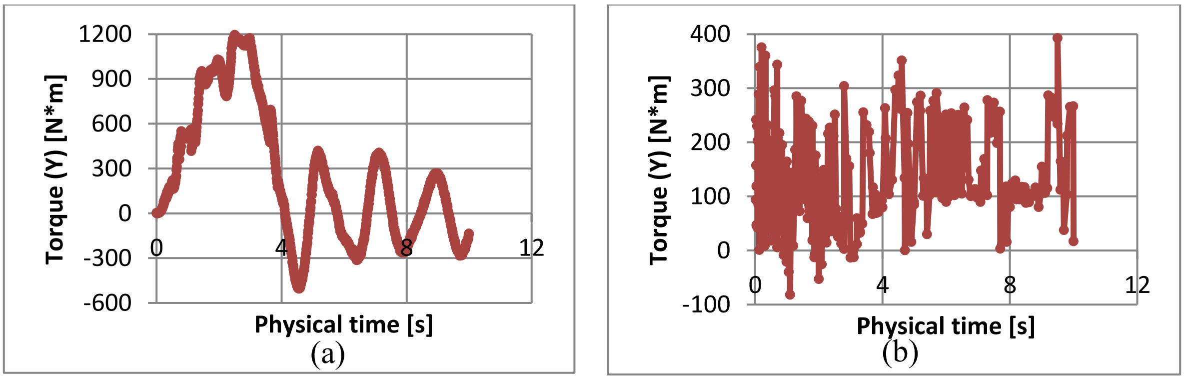


Figure 12. (a) The torque force of the sloshing effect without baffles, (b) The torque force of the sloshing effect with baffles

The result obtained from the oil sloshing analysis provides an essential understanding of the significance of utilising baffles when designing the oil tank of a motorised vehicle. Without the implementation of a baffle, the impact on an oil tank increases the risk of damage or failure to the tank [5], causing imbalance and potential damage to the POSS [10], and making it challenging to keep the POSS in control. To resolve this issue, baffle is implemented to reduce the impact caused by oil sloshing. The analysis results prove that designing an oil tank with baffles significantly reduces the effect of oil sloshing compared to an oil tank without baffles.

4. Conclusion

The Portable Oil Spill Skimmer (POSS) was designed to enhance the mechanical technology used in Oil Spill Response and Recovery (OSRR) operations. The developed POSS consists of an oil tank to recover oil and propellers for self-propelling capability. The POSS could travel at a speed of 3.72 Knots, adhering to the standard of USV specification. However, during the POSS experimental testing, the POSS experiences instability caused by the sloshing effect. The sloshing effect occurs when recovered oil is present in the oil tank. The instability increases when the oil tank is partially filled. The impact of oil sloshing damages the structure of the oil tank and the POSS's stability. Thus, baffles were implemented to reduce the sloshing effect on the POSS. The oil sloshing was analysed by comparing the impact of sloshing, with and without the implementation of baffles. The finding shows significant improvement where the oil sloshing torque force was reduced by 67% from 1195.4N to 392N with the implementation of 3 baffles. Increasing the number of baffles reduces the sloshing impact but increases the weight of POSS. Only 3 baffles were installed in the oil tank to maintain the portability feature of POSS. This paper successfully proved that the sloshing effect was reduced by implementing baffles, enhancing the safety feature of the POSS.

ACKNOWLEDGEMENT

The authors would like to acknowledge the financial support from Universiti Teknikal Malaysia Melaka (UTeM) through the research grant with reference no. PJP/2020/FKP/PP/S01776 for all the support.

REFERENCES

[1] Abdullah L, Jamaludin Z, Chiew TH, Rafan NA, Yuhazri MY. (2012) Extensive Tracking Performance Analysis of Classical feedback control for XY Stage ballscrew drive system. Applied Mechanics and Materials, 30(229):750-5. https://doi.org/10.4028/www.scientific.net/AMM.229-231.750

[2] Tantichukiad K, Yahya A, Mustafah AM, Mohd Rafie AS, Mat Su AS. (2024) Computational Fluid Dynamics Evaluations on New Designs of the Delta-Shaped Blade Darrieus Hydrokinetic Turbine. International Journal of Automotive and Mechanical Engineering (IJAME), 21(2):11247–11262. https://doi.org/10.15282/ijame.21.2.2024.6.0869

[3] Ali MA, Idayu N, Salleh MS, Sivarao S, Yamaguchi M. (2022) Optimization of injection machine parameters on high tensile strength of polypropylene/fibreglass. Journal of Advanced Manufacturing Technology (JAMT), 20(16):3.

[4] Aiman TMKTK, Thamrin NM, Hanif M, Ramli M. (2023) Underwater brushless motor performance analysis of twin hull-based unmanned surface vehicle for water quality monitoring. IEEE 6th International Conference on Electrical, Electronics and System Engineering (ICEESE), 20–25. https://doi.org/10.1109/ICEESE56169.2023.10278188

[5] Zhang E, Chen H, Wang L, Huang J, Xu F. (2024) Numerical analysis of oil liquid sloshing in fuel tanks of plug-in hybrid electric vehicles under different structure baffles. Journal of Physics: Conference Series, 2879(1):012024. https://doi.org/10.1088/1742-6596/2879/1/012024

[6] Liu Z, Y Feng Y, Lei G, Li Y. (2019) Hydrodynamic performance in a sloshing liquid oxygen tank under different initial liquid filling levels. Aerospace Science and Technology, 85:544–555. doi: 10.1016/J.AST.2019.01.005.

[7] Zheng XL, Zhang H, Ren YY, Wei ZH, Song XG. (2017) Rollover stability analysis of tank vehicles based on the solution of liquid sloshing in partially filled tanks. Advances in Mechanical Engineering, 9(6). doi: 10.1177/1687814017703894.

[8] Toumi M, Bouazara M, Richard MJ. (2009) Impact of liquid sloshing on the behaviour of vehicles carrying liquid cargo. European Journal of Mechanics - A/Solids, 28(5):1026–1034. doi: 10.1016/J.EUROMECHSOL.2009.04.004.

[9] Zhang E, He R. (2020) Numerical analysis on the sloshing of free oil liquid surface based on fuel tanks of different shapes. Journal of Mechanical Engineering Science, 234(18):3584–3599. doi: 10.1177/0954406220916483.

[10] Wu X, He R. (2024) Numerical study on characteristics and mechanism of fuel sloshing in automobile fuel tank. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 46(1):1–16. doi: 10.1007/S40430-023-04601-3/METRICS.

[11] Kim J, Kim Y, Kim SY, Lee KM, Sung YJ. (2020) Experimental Study of Slosh-induced Loads on an LNG Fuel Tank of a Container Ship. International Journal of Offshore and Polar Engineering, 30(02):141–148. doi: 10.17736/IJOPE.2020.JC766.

[12] Gurusamy S. (2023) Damping of liquid sloshing by floating balls. Physics of Fluids, 35(11). doi: 10.1063/5.0165870/2922927.

[13] Luo M, Su X, Lin P, Khayyer A, Zhao X. (2022) Investigation of Two-layer Liquid Sloshing by Using the Consistent Particle Method. International Journal of Offshore and Polar Engineering, 32(01):7–15. doi: 10.17736/IJOPE.2022.JC848.

[14] Liu Z, Feng Y, Liu Y, Lei G, Li Y. (2020) Fluid sloshing dynamic performance in a fuel storage tank under sinusoidal excitations. Applied Thermal Engineering, 168(4):114814. doi: 10.1016/J.APPLTHERMALENG.2019.114814.

[15] Liu D, Wu Y, Lin P. (2022) An experimental study of two-layer liquid sloshing under pitch excitations. Physics of Fluids, 34(5). doi: 10.1063/5.0093716/2846755.

[16] Yu L, Xue MA, Jiang Z. (2020) Experimental investigation of parametric sloshing in a tank with vertical baffles. Ocean Engineering, 213:107783. doi: 10.1016/J.OCEANENG.2020.107783.

[17] Nasar T, Sannasiraj SA. (2019) Sloshing dynamics and performance of porous baffle arrangements in a barge carrying liquid tank. Ocean Engineering, 183:24–39. doi: 10.1016/J.OCEANENG.2019.04.022.

[18] Tang YY, Liu YD, Chen C, Chen Z, He YP Zheng MM. (2021) Numerical study of liquid sloshing in 3D LNG tanks with unequal baffle height allocation schemes. Ocean Engineering, 234:109181. doi: 10.1016/J.OCEANENG.2021.109181.

[19] Zang Q, Liu J, Yu L, Lin G. (2019) Boundary element analysis of liquid sloshing characteristics in axisymmetric tanks with various porous baffles. Applied Ocean Research, 93:101963. doi: 10.1016/J.APOR.2019.101963.

[20] Ünal UO, Bilici G, Akyıldız H. (2019) Liquid sloshing in a two-dimensional rectangular tank: A numerical investigation with a T-shaped baffle. Ocean Engineering, 187:106183. doi: 10.1016/J.OCEANENG.2019.106183.

[21] Huang P. (2023) A comprehensive investigation on liquid sloshing of rectangular water tank with vertical baffles. Ocean Engineering, 288:116126. doi: 10.1016/J.OCEANENG.2023.116126.

[22] Jamalabadi MYA, Ho-Huu V Nguyen TK. (2018) Optimal design of circular baffles on sloshing in a rectangular tank horizontally coupled by structure. Water 10(11). doi: 10.3390/w10111504.

[23] Nagalakshmi PSS, Vijaya N, Ibrahim SM. (2023) Entropy Generation of Three-Dimensional Williamson Nanofluid Flow Explored with Hybrid Carbon Nanotubes over a Stretching Sheet. CFD Letters, 15 (7):112-130. https://doi.org/10.37934/cfdl.15.7.112130

[24] Chi C, Pallayil V, Chitre M. (2020) Design of an adaptive noise canceller for improving performance of an autonomous underwater vehicle-towed linear array. Ocean Engineering, 202:106886. doi: 10.1016/J.OCEANENG.2019.106886.

[25] Zhang H, Zhang S, Wang Y, Liu Y, Yang Y, Zhou T, Bian H. (2020) Subsea pipeline leak inspection by autonomous underwater vehicle. Applied Ocean Research, 1(107):102321. doi: 10.1016/J.APOR.2020.102321.