MECHANICAL STRESS-STRAIN ANALYSIS OF A PORTABLE OIL SPILL SKIMMER FRAME FOR RESPONSE AND RECOVERY ACTIVITIES

R. Ramanathan[1], L. Abdullah[1]*, M.H.F. Md Fauadi[1], M.S. Syed Mohamed[1], M.S.M Aras[2] and A.S. Nur Chairat[3]

[1]Faculty of Manufacturing Engineering,
Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia.

[2]Faculty of Electrical Engineering,
Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

[3]Institut Teknologi PLN, Jakarta 11750, Indonesia

*Corresponding Author's Email: lokman@utem.edu.my

Article History: Received 28 October 2022; Revised 3 April 2023; Accepted 12 April 2023

ABSTRACT: Oil spillage has been a problem for over a decade, and proper measures must be taken to address it. Boom barriers and skimmer absorbent materials can be used to contain oil spills. However, problem occurs during high tides and strong waves, where the boom and skimmer reconstruction is necessary as the oil could pass over the barrier. Therefore, a portable oil spill skimmer prototype complemented the existing technique. However, it lacked robustness in the frame structure while coping with rough seas. To improve the portable oil spill skimmer's mechanical performance, this article addresses the enhancement of the Portable Oil Spill Skimmer's frame. Thus, this study objective is to analyze the structural analysis of the improved portable oil spill skimmer frame. The structure was analyzed in terms of deformation, equivalent stress, and strain analysis to identify the structural behavior and failure before fabricating the product. The significance of the study is to analyze the frame structure and determine if the frame can be fabricated and used without failure. The FEA results show that the maximum equivalent stress, maximum equivalent strain, and total deformation values are 16.851 MPa, 0.25896e-3 mm/mm, and 0.3742 mm, respectively. Moreover, the enhanced frame could withstand 39kg of weight, and the safety factor is 3.019. In conclusion, the portable oil spill skimmer frame could survive rough seas and withstand 39kg of weight.

KEYWORDS: *Finite Element Analysis (FEA); Stress-Strain Analysis; Portable Oil Spill Skimmer; Material Selection*

1.0 INTRODUCTION

An oil spill is when petroleum hydrocarbons are unintentionally or intentionally released into the environment, particularly marine ecosystems. Wind and tide are capable of carrying an oil spill [1]. The oil that has been dispersed either vaporizes, produces a surface slick, disperses in water, or sinks and accumulates in the sediments [2]. Thus, several established methods are used in the Oil Spill Response and Recovery (OSRR) techniques. There are mechanical techniques, chemical techniques, in-situ burning, and biodegradation methods, which serve different purposes according to the level of the oil spill and oil slick, types of oil, and weather conditions [3]. However, each existing mechanical technique has common disadvantages, such as costly, intensive workforce, and additional treatment to segregate the collected oil from the water [4].

As a result of these limitations, researchers have conducted studies to develop a prototype of a portable oil spill skimmer to serve as a complementary solution to the current mechanical technique of oil recovery from the water surface [5]. The prototype has a vast operation area covering the ocean, seashore, water treatment plants, rivers, and reservoirs. The prototype uses an oleophilic and hydrophobic roller skimmer to collect oil from the water surface. However, based on the results and observations obtained from the study, there are shortcomings in the prototype, such as the low robustness of the frame structure when it reaches an area of rough water conditions. Therefore, this study aims to analyze the structural analysis of the improved portable oil spill skimmer frame in terms of deformation, equivalent stress, and strain analysis.

Mechanical analysis is one of the crucial analyses to be performed as its results will be required in the subsequent design process, such as electrical and electronics integration designs, controller designs, aerodynamic analysis, and industrial machines [6]–[8]. The Finite Element Analysis (FEA) is one of the mechanical analyses, which is a numerical analytical technique for approximating solutions to various engineering issues. The definition of FEA is the discretization of a domain using points called "nodes" with Degrees of Freedom (DoF) and coupled to one another by geometrical objects called "elements" for the transmission of data [9]. ANSYS is a FEA software that calculates bending stresses, deformations, equivalent stress and strain analyses, and other things [10]. The finite element analysis has three fundamental steps: pre-processing, solution, and post-processing [11]. Pre-processing entails discretizing or meshing the geometry and defining the materials, loads, and boundary conditions. In an era where the industry seeks faster results, FEA offers quicker solutions with logical and reasonable accuracy [12]. FEA can reduce the need for laboratory testing, which allows the task to be accomplished more quickly and at a lower cost.

The benefits of implementing FEA can be seen in a study on steel frame connection analysis [13]. The finite element analysis result shows the deformed shape of the connection and gives a more accurate location of the affected zones through the simulation. Furthermore, the FEA was utilized to analyze the deformation of a vehicle chassis [14]. The author states that implementing FEA aids in identifying whether a frame with specific material could deform in several impact scenarios of a Formula Society of Automotive Engineers (SAE) vehicle. A research was conducted to evaluate the safety of automobile seat structures made of various materials [15]. The findings from the FEA simulation revealed that when Carbon Fiber Reinforced Polymers (CFRP) materials are applied to the actual seat frame, indicates to provide a higher safety in comparison to other steel materials in terms of deformation, displacement, and yield strength.

It is well known that previous researchers have investigated and reported on the FEA in different frame types. However, to the best of the authors' knowledge, however, the structural analysis of frame implementation in developing a portable oil spill skimmer is still inadequate to be reviewed. Moreover, the existing prototype has minimal robustness in the frame structure. Since the application of FEA is insufficient in any oil spill skimmer research, the literature discusses the FEA application in other areas. Section 3 explains the step-by-step procedure of structural analysis of the prototype in detail, followed by the result and discussion in Section 4. Finally, the conclusion and future recommendations are enlightened in Section 5.

2.0 MATERIALS AND METHODS

Before conducting the structural analysis, the product development process is crucial to identify the design and structure of the product. The methodology flow chart is shown in Figure 1. Firstly, the problem must be analyzed and the Kansei Engineering approach was utilized to identify the design element to develop conceptual designs. The design elements can be seen in the previous study [16]. The design development must follow the specified elements when designing the prototype. The designed prototype then undergoes several structural analyses such as equivalent stress-strain analysis, total deformation and factor of safety before the fabrication process. These analyses were conducted to identify whether the designed structure is safe to be fabricated.

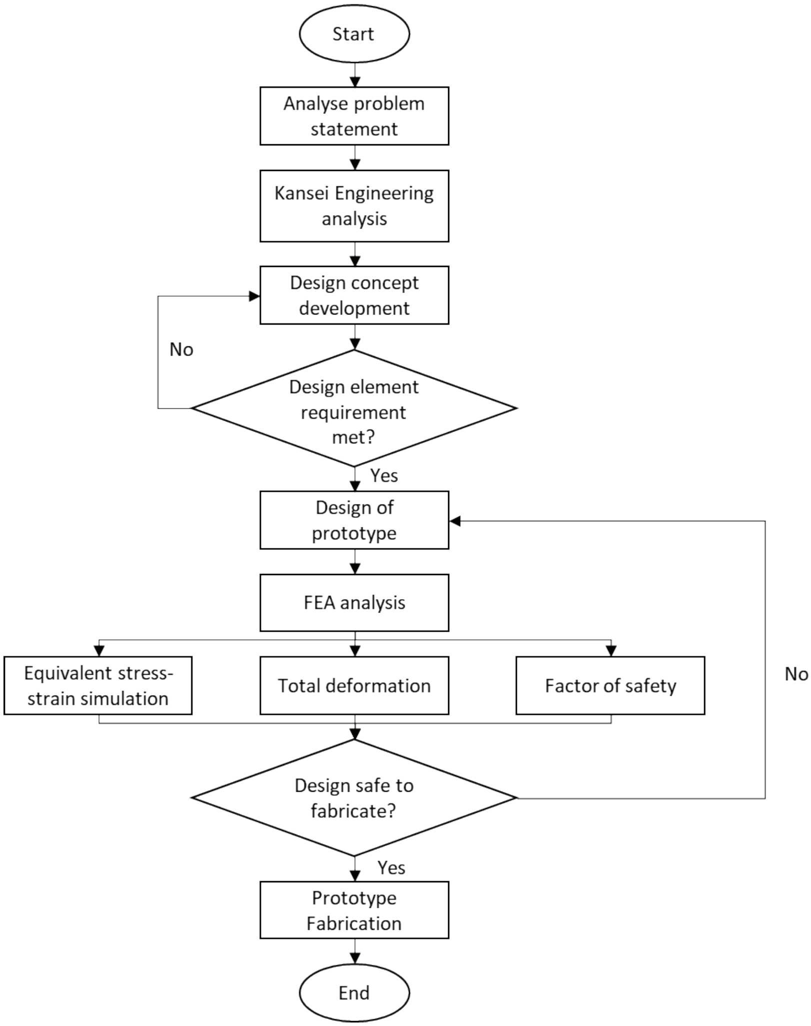

Figure 1: The methodology flowchart

The design is considered to be safe when the total equivalent stress value is smaller than the yield strength, small structural deformation (measured in mm) and a factor of safety is larger than 1.0. Finally, the prototype is fabricated when the performed analysis is successful.

2.1 Detailed Design

The Portable Oil Spill Skimmer's conceptual design has been developed using SolidWorks. This oil spill skimmer design comprises a catamaran hull held together by a frame structure. The frame structure also serves as mounting components for the skimmer and thrusters, and it is most prone to failure while in use. Figure 2 shows the detailed design of a portable oil spill skimmer.

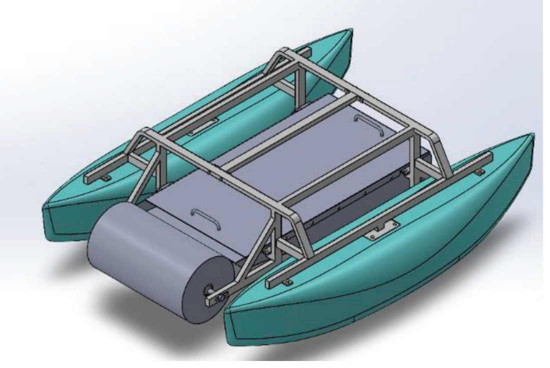

Figure 2: The detailed design of the portable oil spill skimmer

2.2 Materials Selection of the Frame

In order to categorize engineering materials, "groups" such as ceramics, metals, natural materials, glasses, polymers, and composites have been created. Data gathering, screening, and scoring go hand in hand, which is the most effective material selection method. The collection of material properties will be done by using CES EduPack software (Table 1). Figure 3 shows the CES EduPack material graph for the selected materials. The second step was concept screening, which only involves comparing each material; if the material is better than the reference, a "+" is given. If it is worse than the reference, a "-" is presented. If it is similar to the reference, a "0" is used.

Concept scoring is a refined and in-depth kind of concept screening. The material with the higher score is the best option based on the created criteria. Four materials were chosen for the screening process for the frame and to enhance the skimmer stability: polyvinylchloride (PVC), aluminium, stainless steel, and polycarbonate (PC). The material properties listed in the table are based on the design requirements in Table 2 for screening purposes.

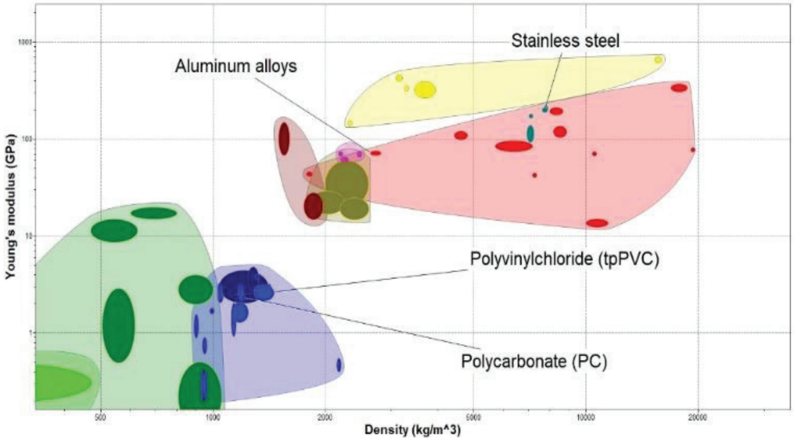

Figure 3: The CES EduPack material graph for the selected materials

The characteristics of corrosion resistance and young modulus are essential to consider when selecting a frame for a portable oil spill skimmer. Thus, the weightage of 30% is given, followed by fracture toughness and density of 15%, and lastly, the price weightage of 10% as it is the least essential characteristic to be considered. Furthermore, the ranking indicates the multiplier to the weightage.

The results from the scoring table show that aluminium ranks first among the four materials based on its properties. Aluminum has a higher value of Young's modulus and fracture toughness which will give advantages over polyvinyl chloride (PVC) and polycarbonate (PC) regarding the robustness of the portable oil spill skimmer. Nevertheless, the consideration between the material's strength and density as the design requirement's objectives focuses on robustness and weight. Aluminium is chosen as the frame's material for enhancing the mechanical performance of the portable oil spill skimmer.

Table 1: Properties of selected materials for frame and hull

Material Properties	Polyvinyl Chloride (PVC)	Aluminium Alloy	Stainless Steel	Polycarbonate (PC)
Price (MYR/kg)	10.3-10.5	10.1-10.9	12.2-13.1	13-14.3
Density (kg/m3)	1.29e3-1.45e3	2.642e3-2.81e3	7.61e3-7.83e3	1.19e3-1.21e3
Corrosion resistivity to salt water	High	High	Low	High
Young's Modulus (GPa)	2.19-3.11	69-75	190-210	2.24-2.52
Fracture Toughness (MPa M0.5)	3.63-3.85	23-38	57-137	1.94-2.48

Table 2: Screening method of the selected material for the frame

Selection Criteria	Weight	Polyvinyl Chloride (PVC)		Aluminium Alloy	
		Ranking	Weighted Score	Ranking	Weighted Score
Price (MYR/kg)	10%	4	0.4	4	0.4
Density (kg/m3)	15%	4	0.6	3	0.45
Corrosion resistivity to salt water	30%	5	1.5	5	1.5
Young's Modulus (GPa)	30%	2	0.6	3	0.9
Fracture Toughness (MPa.m$^{0.5}$)	15%	2	0.3	3	0.45
Total Score	100%		3.4		3.7
Rank		2		1	

Table 3: Scoring method of the selected material for the frame

Material Properties	Polyvinyl Chloride (PVC)	Aluminum Alloy	Stainless Steel	Polycarbonate (PC)
Price (MYR/kg)	+	+	0	-
Density (kg/m3)	+	0	-	+
Corrosion resistivity to salt water	+	+	-	+
Young's Modulus (GPa)	-	0	+	-
Fracture Toughness (MPa.m$^{0.5}$)	-	0	+	-
Sum of (+)	3	2	2	2
Sum of (0)	0	3	1	0
Sum of (-)	2	0	2	3
Net Score	1	2	0	-1
Rank	2	1	3	4
Continue	**Yes**	**Yes**	No	No

2.3 Boundary and Loading Condition

In phase two, the step-by-step procedures for conducting the FEA in ANSYS software is explained meticulously, as shown in Figure 4. To obtain results with precise data in the form of equivalent von Mises stress to ascertain whether there is any failure in the design phase, the boundary and loads for the simulation must be specified at the appropriate location of the

3D model. As a result, it is mostly performed on the structure where the frame and hull are joined, simulating the components during assembly using static loads with uniform distribution. The static loading and imposed constraint on the frame shape are shown in Figure 5.

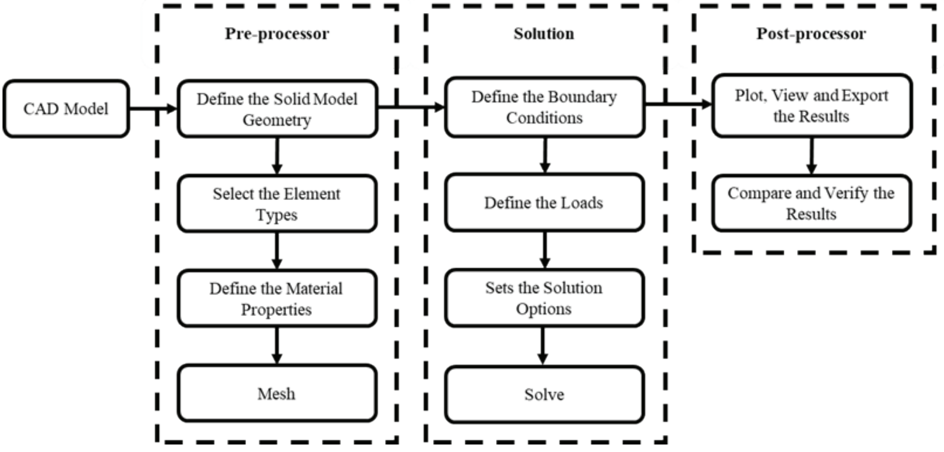

Figure 4: The proposed flowchart of the finite element analysis (FEA) simulation steps

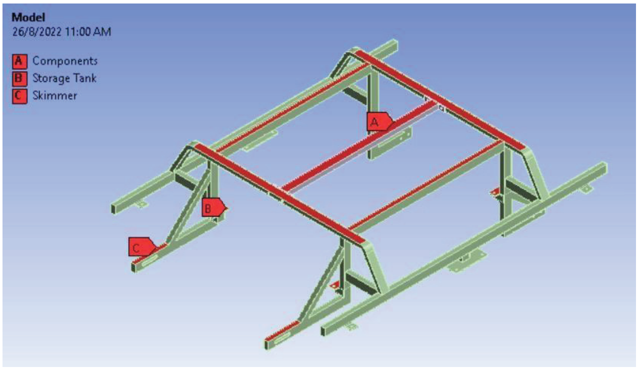

Figure 5: Static loading and applied constraint on the frame geometry

A load of 15 kg, or 147.15 N, is applied to the frame as the result of the combined weight of multiple components that are mounted to the frame. The importance of 15 kg is the electrical equipment, including two batteries weighing 2.5 kg each and a pair of outriggers weighing 5 kg each. In order to better reflect actual conditions, the simulation will also use standard earth gravity. The red region with "A", represents the load for the components, "B" the storage tank, and "C" the skimmer roller. Blue arrows indicate the location of the Fixed Support features on the base plate, which is where the frame and hull join. Additionally, the 5 kg oil skimmer roller that is located in front of the specified area exerts stress on the structure.

Before the solution of simulation, meshing must be applied to the design. Meshing is a geometric shape formed over the design to help define the physical shape. The higher the detail of meshing, the higher the data accuracy produced. The parameter set for the meshing in Ansys is shown in Table 4. Figure 6 displays the applied mesh on the frame, which has 202881 nodes and 101378 elements. A Tetrahedral element meshing was utilized in this simulation.

Table 4: Data of a mesh for the aluminium frame

Element Order	Program Controlled
Element Size	5.0 mm
Resolution	3
Transition	Fast
Span Angle Centre	Fine
Smoothing	High
Nodes	202881
Elements	101378

Before the solution of simulation, meshing must be applied to the design. Meshing is a geometric shape formed over the design to help define the physical shape. The higher the detail of meshing, the higher the data accuracy produced. The parameter set for the meshing in Ansys is shown in Table 4. Figure 7 displays the applied mesh on the frame, which has 202881 nodes and 101378 elements. A Tetrahedral element meshing was utilized in this simulation.

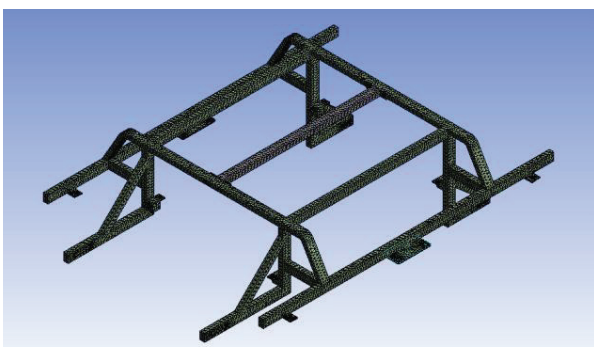

Figure 6: Applied mesh on the frame

3.0 RESULT AND DISCUSSION

3.1 Equivalent Stress Simulation

The simulation is conducted with a total static loading of 21 kg, or 206.01 N, including the weight of the skimmer 5 kg and two thrusters of 0.5 kg each. The result shows the maximum equivalent stress value of 16.851 MPa, as shown in Figure 5. Therefore, this frame design is safe as the stress value presented is lower than the yield strength of the material of 280 MPa. A similar statement can be observed in a paper, where the researcher stated that any material would fail when the specific yield strength is exceeded [17].

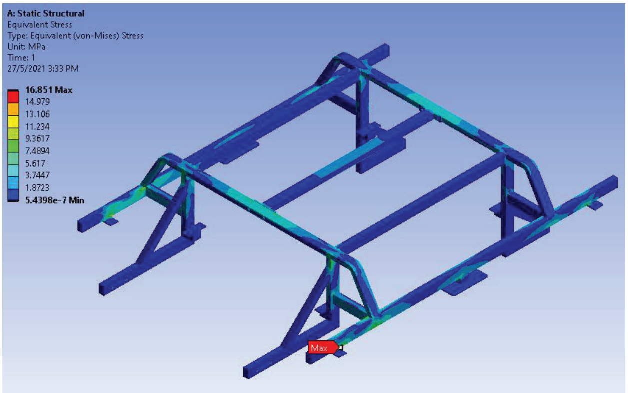

Figure 7: Equivalent stress simulation with contour view

3.2 Equivalent Strain Simulation

Given the static loads of 21 kg, or 206.01 N, the findings of the stress-strain analysis simulation indicate the maximum equivalent strain value to be 0.25896e-3 mm/mm. The outcome of the strain simulation is depicted in Figure 8.

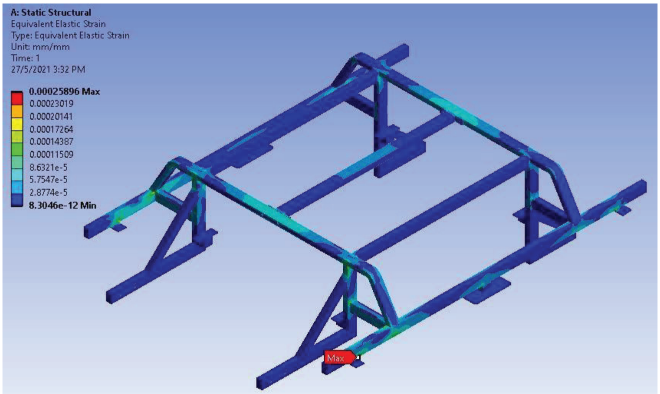

Figure 8: Equivalent strain simulation with contour view

3.3 Total Deformation

Given a static load of 21 kg (206.01 N), the findings of the stress-strain analysis simulation indicate a maximum deformation of 0.3742 mm. The red indicates the maximum stress, and the blue shows the minimum pressure generated on the frame. The upper middle bar of the frame experiences the most deformation, as seen in Figure 9, however, the amount is too little to be observed by the naked eye. Therefore, the Portable Oil Spill Skimmer frame is rigid and will not distort. Thus, the structure is capable of withstanding 100kg of load before the deformation could be visually noticed.

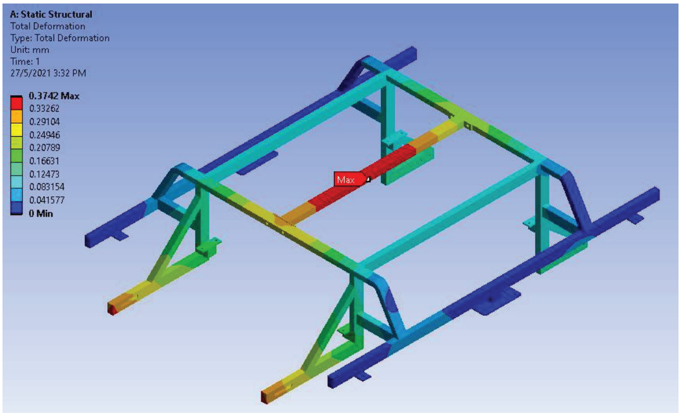

Figure 9: Total deformation with contour view

3.4 Variable Load of Storage Tank

The simulation is performed on the frame when different loads are applied on the mounting surface of the storage tank to replicate when the tank is filled with liquid. According to the storage tank's design, it may hold up to 18kg of liquid. Thus, a series of loads begin from 3 kg, 6 kg, 9 kg, 12 kg, 15 kg, to 18 kg, and the weight of other components is applied to the frame. This is to analyze whether the portable oil spill skimmer frame can uphold the weight of the oil and additional equipment during the oil recovery process.

Table 5: Simulation result of the stress of different loads applied

Loading of the tank (kg)	Static loading (kg)	Total loading (Tank + Static)	Stress value (MPa)
0	21	21	16.851
3	21	24	18.610
6	21	27	20.369
9	21	30	22.128

12	21	33	23.888
15	21	36	25.647
18	21	39	27.406

The results from the simulation show an additional 3 kg of load up to 18 kg is tabulated in Table 5. The results of the stress simulation on the variable load are shown in Figure 8. The finding reveals a consistent increase in the stress value, indicating that the portable oil spill skimmer may withstand the maximum weight of 39 kg without failure. Moreover, the frame has a factor of safety of 3.019, which indicates that the structure is safe to use, as shown in Figure 10. A safety factor of precisely 1.0 means that the actual stress equals the material strength limit, so the design is on the verge of failure. It is also stated that having a higher safety factor is important to sustain the loading condition thus making the design safe to be used [18].

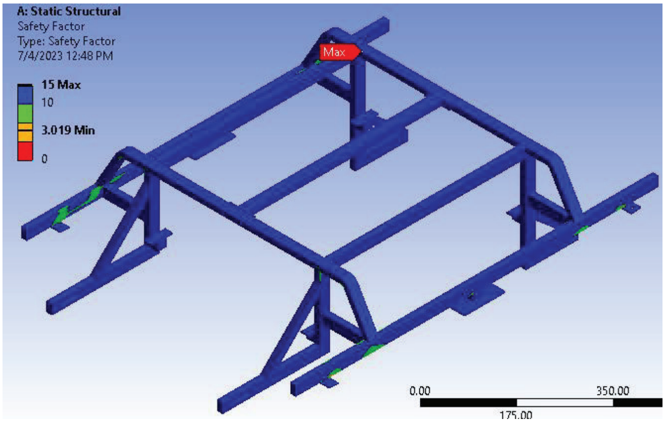

Figure 10: The factor of safety of the portable oil spills skimmer frame

4.0 CONCLUSION

In this paper, a new and improved portable oil spill skimmer frame has been designed and analyzed with Finite Element Analysis (FEA). Stress-strain analysis, total deformation, and safety factors were investigated on the structure. Aluminum hollow is the best material for the portable oil spill skimmer frame. The results show the frame could withstand 39 kg of weight without failure. Furthermore, the safety factor of 3.019 indicates that the structure may withstand three times the weight of 39 kg before structural failure. Moreover, the stress and strain acted upon the skimmer are 16.851 MPa and 0.25896e-3 mm/mm, respectively. In addition, the total deformation of the skimmer was 0.3742 when the weight was applied. In conclusion, aluminium hollow is selected to be fabricated according to the design; thus, the objective has been achieved.

ACKNOWLEDGMENTS

This paper and its research would not have been possible without the exceptional support from the UTeM Short Term Grant no (PJP/2020/FKP/PP/S01775). Furthermore, we are eternally grateful for the generous support from the Center of Research Innovation Management (CRIM) Universiti Teknikal Malaysia Melaka for the publication of this paper.

REFERENCES

[1] Q. Pan, H. Yu, P. S. Daling, Y. Zhang, M. Reed, Z. Wang, Y. Li, X. Wang, L. Wu, Z. Zhang, H. Yu, and Y. Zou, "Fate and behavior of Sanchi oil spill transported by the Kuroshio during January-February 2018", *Marine Pollution Bulletin*, vol. 152, 2020.

[2] B. Doshi, M. Sillanpää, and S. Kalliola, "A review of bio-based materials for oil spill treatment", *Water Research*, vol. 135, pp. 262–277, 2018.

[3] A. T. Hoang, V. V. Pham, and D. N. Nguyen, "A Report of Oil Spill Recovery Technologies", *International Journal of Applied Engineering Research*, vol. 13, no. 7, pp. 4915–4928, 2018.

[4] P. E. Ndimele, A. O. Saba, D. O. Ojo, C. C. Ndimele, M. A. Anetekhai, and E. S. Erondu, "Remediation of Crude Oil Spillage", *The Political Ecology of Oil and Gas Activities in the Nigerian Aquatic Ecosystem*, pp. 369–384, 2018.

[5] L. Abdullah, K. Santuso, Z. Jamaludin, R. Zamri, and M. N. Maslan, "Design and analyses of semi-automated portable oil spill skimmer for water treatment application", *Lecture Notes in Mechanical Engineering*, 2020, pp. 372–384.

[6] L. Abdullah, Z. Jamaludin, T. H. Chiew, N. A. Rafan, and M. S. S. Mohamed, "System identification of XY table ballscrew drive using parametric and non parametric frequency domain estimation via deterministic approach", *Procedia Engineering*, vol. 41, pp. 567–574, 2012.

[7] K. Christodoulou, M. Vozinidis, A. Karanatsios, E. Karipidis, F. Katsanevakis, and Z. Vlahostergios, "Aerodynamic Analysis of a Quadcopter Drone Propeller with the Use of Computational Fluid Dynamics", *Chemical Engineering Transactions*, vol. 76, pp. 181–186, 2019.

[8] M. Ali, N. Idayu, M. Salleh, S. Sivaraos, and M. Yamaguchi, "Optimization of Injection Machine Parameters on High Tensile Strength of Polypropylene/Fibreglass", *Journal of Advanced Manufacturing Technology*, vol. 16, no. 3, pp. 1985–3157, 2022.

[9] H. Venkatesan, P. R. Roy, V. Hariram, M. Subramanian, and P. R. Roy, "Accelerating the Product Development of a Commercial Vehicle Radiator using Finite Element Analysis", *International Journal of Vehicle Structures & Systems*, vol. 8, no. 5, pp. 269–272, 2017.

[10] M. Phyoe Thu and N. Lin Min, "Stress Analysis on Spur Gears Using ANSYS Workbench 16.0", *International Journal of Science and Engineering Applications*, vol. 7, pp. 208–213, 2018.

[11] C. Li, S. Fan, and M. Shi, "Preparation of CAD model for finite element analysis", in *2010 International Conference on Computer, Mechatronics, Control and Electronic Engineering, CMCE 2010*, pp. 491–494. 2010.

[12] A. Srirekha and K. Bashetty, "Infinite to finite: An overview of finite element analysis," *Indian Journal of Dental Research*, vol. 21, no. 3. pp. 425–432, 2010.

[13] A. Ivan, M. Ivan, and I. Both, "Comparison of FEA and experimental results for a steel frame connection", *WSEAS Transactions on Applied and Theoretical Mechanics*, vol. 5, no. 3, pp. 187–196, 2010.

[14] Rupesh Patil, "FEA Analysis of FSAE Chassis", *International Journal of Engineering Research and Technology*, vol. 9, no. 7, pp. 290-295, 2020.

[15] H. S. Kim, Y. S. Lee, S. M. Yang, and H. Y. Kang, "Structural analysis on variable characteristics of automotive seat frame by FEA", *International Journal of Precision Engineering and Manufacturing-Green Technology,* vol. 3, no. 1, pp. 75–79, 2016.

[16] R. Ramanathan, L. Abdullah, and M. S. S. Mohamed, "The Utilisation of Kansei Engineering in Designing Conceptual Design of Oil Spill Skimmer", *Lecture Notes in Mechanical Engineering*, vol. 25, pp. 434–447, 2021.

[17] T. Heitkamp, S. Girnth, S. Kuschmitz, N. Waldt, G. Klawitter, and T. Vietor, "Experimental and Numerical Investigation of the Mechanical Properties of 3D-Printed Hybrid and Non-Hybrid Composites", *Polymers*, Vol. 15, Page 1164, vol. 15, no. 5, p. 1164, 2023.

[18] R. K. Mohanty, R. C. Mohanty, and S. K. Sabut, "Finite element analysis and experimental validation of polycentric prosthetic knee", *Materials Today: Proceedings,* vol. 63, pp. 207–214, 2022.