NUMERICAL INVESTIGATION ON THE FUSELAGE AIRFRAME OF LSU 05 NG

M. Hafid, Abian Nurrohmad[1], Redha Akbar Ramadhan
Aeronautics Technology Center, Indonesian National Institute of Aeronautics and Space (LAPAN)
[1]e-mail: m.hafid@lapan.go.id
Received: 17 April 2020; Revision: 6 may 2020; Accepted : 14 July 2020

ABSTRACT

In this paper, a numerical investigation on the fuselage structure of LSU 05 NG was carried out. This fuselage is designed to carry the payload up to 25 kg. Statical numerical analysis using the finite element method has been done by using Simulia Abaqus. The fuselage structure that has been designed which consists of frame, longeron, and skin that can also be called a semi-monocoque structure. This airframe uses combination of balsa and GFRP type of composite as the material. There are three load cases,i.e. take-off condition, cruise condition, and landing condition. Tsai-hill failure criteria is used to investigate the strength of the composite structure due to the load that is applied. The maximum stress from this calculation is 49.24 MPa on the ground condition (take-off and landing) while the maximum principle stress on the cruise condition is 52.76 MPa. The maximum Tsai-hill criterion is 0.95 at the cruise condition. With such simulation results, it can be said that the fuselage structure is still safe when operated and can also be optimized for several components so that the weight of the aircraft can be reduced.

Keywords: *finite element method, stress analysis, fuselage, unmanned aircraft.*

1 Introduction

LSU 05 NG is an unmanned aircraft developed by Pustekbang LAPAN. It is the new generation of LSU 05 with a wider fuselage to carry more payloads. The fuselage is reinforced with frame and longeron along in its skin structure which is called the semi-monocoque structure.

The aircraft developed from LSU 05 UAV (Unmanned Aircraft Vehicle) is a part of the Maritime Surveillance System Project. It has improved payload performance by optimizing the airfoil and the geometry of the internal structure. LSU 05 NG is designed to carry various missions. The main mission for LSU 05 NG is maritime surveillance with additional missions such as air logistics, search mission, and aerial photography. Every mission has its loading profile.

There are so many previous research about design and numerical simulation which has been done to build some unmanned aircraft. Chen and Lu have done the numerical simulation method of UAV fuselage compression under centrifugal load based on the discrete element analysis method. The purpose of this method is to calculate the centrifugal load stress spectrum of the surface of the UAV fuselage. By doing this method, the accuracy of the anti-compression numerical test of the UAV becomes higher (Chen & Lu). Ariyanto, et al has designed a low-cost fixed-wing unmanned aerial vehicle that can fly autonomously. This research is using the weighting factor to determine the best configuration of each design parameter (Ariyanto et al).

Chung et al also presents their report about design, manufacturing, and flight testing an experimental flying wing UAV. The design of the airframe is based on matching plot, weight estimation approach, and conventional aircraft design procedure (Chung et al). Velasco et al implementing UAV Design using five requirements such as maximum span, budget ceiling, payload and mission, weight, and cruise speed. The design meets the measurable design requirements even though it is using adjustments since the requirements is less specific and therefore is hard to assess (Velasco et al). Design and analysis of UAV fuselage is performed by using finite element analysis (Sighanart et al) which shows the maximum stress while load is applied (Dassault Systemes).

Raja et al has described the rivetted splice joint at the fuselage structure. Their calculation is performed by using cabin pressurization in aircraft fuselage. They use the global finite element using static analysis to get the benchmark results from the splice joint panel. This structure is a safe limit at this load case (Raja et al). Hadjez and Necib use the Patran-Nastran software to study the stress analysis on the aircraft fuselage with and without porthole. This work is to study the stress concentration in the aircraft fuselage due to the presence of porthole (Hadjez & Necib).

Material selection has a crucial role in the analysis of UAV Fuselage. To achieve the strength and lightweight structure, the composite material has been proven as an excellent material for lightweight structures (Abdurohman & Marta). The composite is ductile enough so that it can be used as a UAV fuselage material (Isna et al).

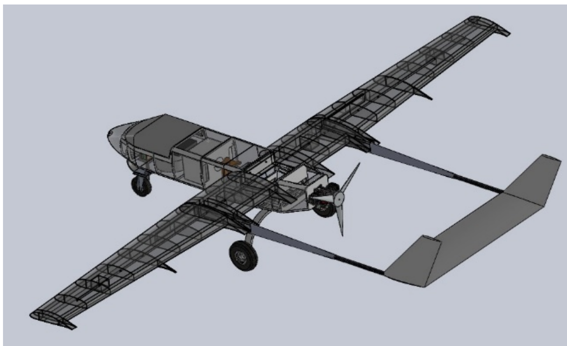

Figure 1-1: Detail Design of LSU-05 NG

In this research, there are three loading configurations on LSU-05NG's fuselage. The loading is simulated as static load as the result of combination of various loading such as wing and empennage loading. The loading configurations consists of take-off condition, cruise condition, and landing condition. The prediction of structure strength will be conducted by numerical simulation. Numerical simulation will be done with commercial finite element software Abaqus. The result of simulation will be evaluated by the Von Mises and Tsai-Hill failure criteria hence the strength of structure can be determined.

2 Methodology

The method used in this research is to start by design the fuselage of UAV using CAD (Computer Aided Design) software. In this case, the software used is Solidwork 2017. After the design is complete, the next step is exporting the drawing model in to the finite element software and set up the finite element model. The finite element model consists of main component that divided into small partitions that called finite elements. The boundary conditions and material databases need to be inputted into the model, so the simulation could yield results to be analyzed. The workflow of this research is shown in Figure 2-1.

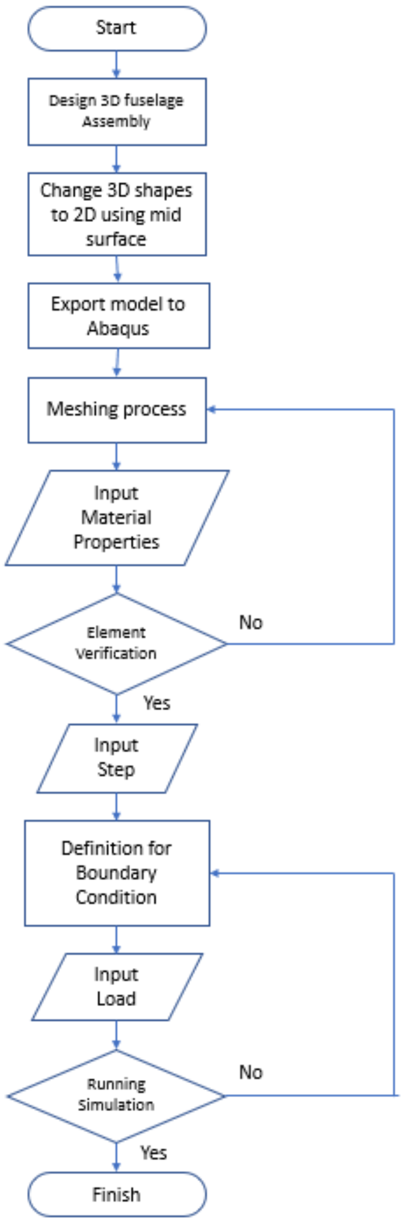

Figure 2-1: Workflow of Research

The detail design of the UAV fuselage airframe has been done by using Solidwork 2017. The detail drawing can be seen on Figure 2-2. Commonly the fuselage of aircraft consists of frame, skin, longeron, and bulkhead. The frame is dedicated to give the shape of the fuselage. The longeron together with the frame carry the bending load and torsion in the fuselage due to its own weight and the weight of payload inside. The bulkhead is the reinforced frame that dedicates to carry heavier payload. Usually the bulkhead is placed in the joint between wing and fuselage and fuselage with landing gear. The skin structure is effective to carry tension load and is dedicated to be the shield of system in the fuselage during the mission of this aircraft.

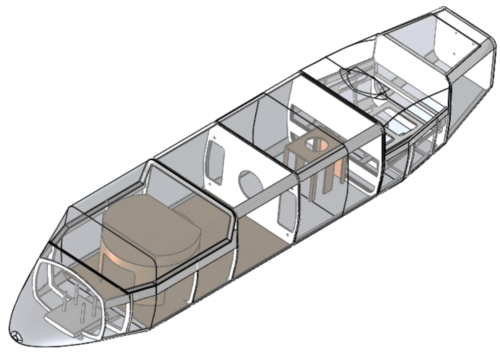

Figure 2-2: Fuselage 3D Detail Drawing

After the 3D model is complete, the next step is to build the finite element model. In this case, the structure can be modelled as shell element, to simplify the simulation process while maintaining the consistency of the model. The simplification has been done with CAD software Solidwork 2017 by simplifying solid parts into surface. The surface model is dedicated to build the finite element model using shell element instead of solid element. The benefit of shell element in this case is the running time is faster and the results is still pretty accurate.

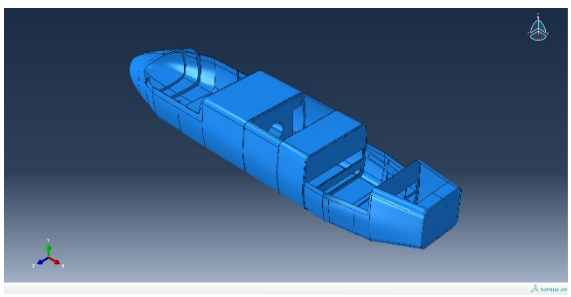

Figure 2-3: Fuselage Surface Model

The skin structure is simplified from a 3D model into a surface model by generating from its middle surface. The skin of this fuselage using composite as material properties. In this skin there are three openings which function for the entry and exit of the payloads and fuel tank and attachments with wing. The detail surface model of the fuselage skin can be seen in Figure 2-4 below.

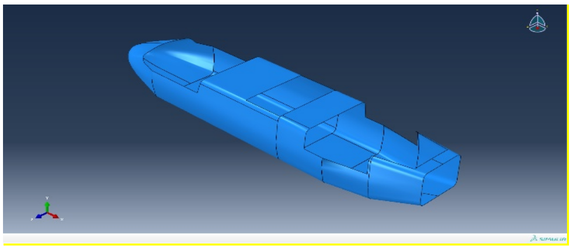

Figure 2-4: Fuselage Skin Model

The frame bulkhead and the longeron is also converted into the surface model. The frame bulkhead and the longeron also use composite for material properties. The detail surface model of the bulkhead, frame and longeron can be seen in Figure 2-5.

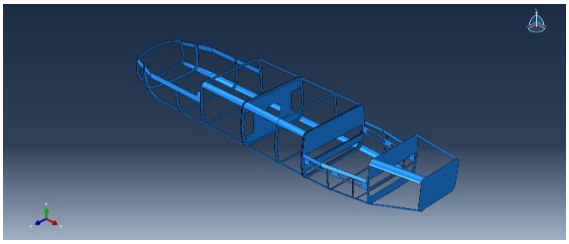

Figure 2-5: Fuselage Frame and Longeron Model

One of the important process on structural stress analysis is meshing process. This process dividing a part into several element. The element acts as finite element where the reaction and stress acts at each corner of the elements. The quality of the mesh will determine the simulation. Finer mesh will yield more accurate results than coarse mesh but with increased computation time. The element that is used in this model is quad shell element type. This element have a square shape. The quad element is used with plane stress assumption. This assumption can be used accurately for thin relative structure. The benefit of using quad shell element instead of solid element is faster in the computational time. The shell element assumption can not used if the concern is in the trough thickness structure. The finite elemet model with quad dominated shell element can be seen in Figure 2-6.

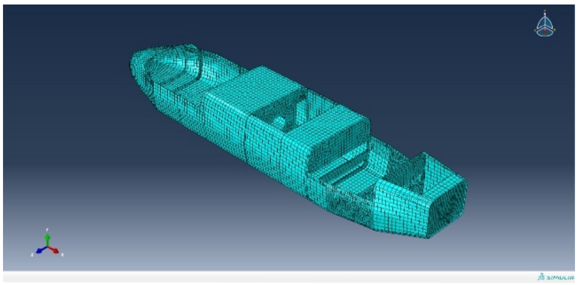

Figure 26: Meshed Geometry

The skin structure of fuselage is using composite material. A composite material is often used because it has several advantages including being lighter and possessing high strength. Designed material for LSU 05 NG's fuselage structure is using E-Glass EW185/Epoxy composite, with 4 ply. The properties of the composite is shown below in table 2-1. The main reason to use E-glass in fuselage structure is about its good interaction with avionic component. It is because the main avionic device like control unit, battery, antenna are located in the fuselage.

The values from table 2-1 are obtained from tensile testing of composite materials using 4 layers of E-Glass fiber ±45^{o} which has a density of 2.2 g/ cm^{3} the results of the test are greatly affected by material, method, specimen conditions, preparation, and also the percentage of reinforcing material. (Abdul-Hamead et al).

Table 2-1: Material Properties

Properties	Value
E1	8090 MPa
E2	8090 MPa
υ_{12}	0.1
G12	1270 MPa
G13	1270 MPa
G23	1270 MPa
$\sigma_{max\_tension\_1}$	261 MPa
$\sigma_{max\_tension\_2}$	261 MPa
$\sigma_{max\_comp\_1}$	63.66 MPa
$\sigma_{max\_comp\_2}$	63.66 MPa
σ_{max12}	17.3 MPa

To assess the damage in the composite structures, Tsai-Hill failure criterion will be used. Tsai-Hill criterion predicts the failure of anisotropic materials by comparing the value of working stresses with maximum stresses of the material. The equation used in Tsai-Hill criterion is shown below,

$$\left(\frac{\sigma_{11}}{X_{11}}\right)^2 - \left(\frac{\sigma_{11}\sigma_{22}}{X_{11}^2}\right) + \left(\frac{\sigma_{22}}{X_{22}}\right)^2 + \left(\frac{\sigma_{12}}{X_{12}}\right)^2 \leq 1 \qquad (2\text{-}1)$$

In equation above, σ and τ refers to working stresses on material. X refers to maximum stresses of material. Subscripts in front of X indicates the direction of maximum stress. The material will fail when the result of equation exceeds 1.

Table 2-2: Concentrated Loads

Loads	Value
Engine Weight	44.64 N
Payload Weight	119.76 N
Wing Weight	174 N
Empennage Weight	81.96 N
Fuel Tank (Full Tank)	164.42
Engine Thrust	360 N
Fuselage & Landing Gear Weight	133.22 N

In this research, there are six loads in form of concentrated loads. This load is subjected to represents the real load that happens in the fuselage structure when the fuselage at the ground condition or the flight condition. The loads and their values are shown below in table 2-2.

In the process of analyzing the load that applied in fuselage, the load is considered as a concentrated force. Concentrated force is the load that will be located specifically at a particular location since weight/force and location of the load is known. Many of load are simplified to six loads that have been tabulated by table 2-2. Boundary conditions are defined as fixes on landing gear attachments, center fuselage connections with nose, and wing attachment. The fuselage boundary conditions are given in table 2-3.

Table 2-3: Fuselage's Boundary Condition

BC#	BC 1	BC 2	BC 3
U1	0	0	0
U2	0	0	0
U3	0	0	0
UR1	0	0	1
UR2	0	0	0
UR3	0	0	0

The load and boundary condition application is given by Figure 2-7.

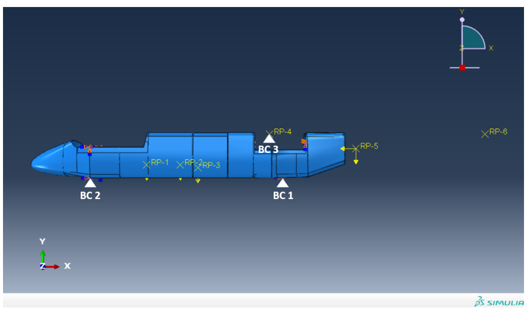

Figure 2-7: Definition for boundary condition and loads

The static load on LSU 05 NG will be simulated with three load case. These three load case represents critical loading scenario on fuselage. Take-off and landing condition is part of ground load. This condition is critical for fuselage structure because the ground reaction at the landing gear station. While the flight condition is critical in the joint between fuselage and wing structure. The configurations is shown in table 2-4.

Table 2-4: Fuselage Static Loading Configuration

Loading Configura-tion	Scenario	Loadings	Boundary Conditions
#1	Take-Off Condition	Engine, Payload, Wing, Fuel Tank, Empennage, and Engine Thrust	BC 1, BC 2
#2	Cruise Condition (Load Factor 2G)	Engine, Payload, Fuel tank, Fuselage and Landing gear, and Engine Thrust	BC 3
#3	Landing Condition	Engine, Payload, Wing, Fuselage and Landing Gear, Empennage, Engine Thrust (50%), and fuel tank	BC 1, BC 2

(50%)

In Abaqus, when 2D components are imported from solidwork it consists of parts that have not been connected to another. So that constraint process is needed to enable modeling kinematic relationship between components chosen is tie constraint because it can be used to make translational and rotational motion and all other degrees of freedom equal for a pair of surfaces. In working with a tie constraints, one surface or region node in the constrint is defined as the master and the other surface or region node is determined as a slave. Nodes are tied only where the surfaces are close to one another. The process of selecting surfaces as master and slave can deterine the result of numerical computation. Each node can only be used as a slave once and if more than one then simulation will run an error.

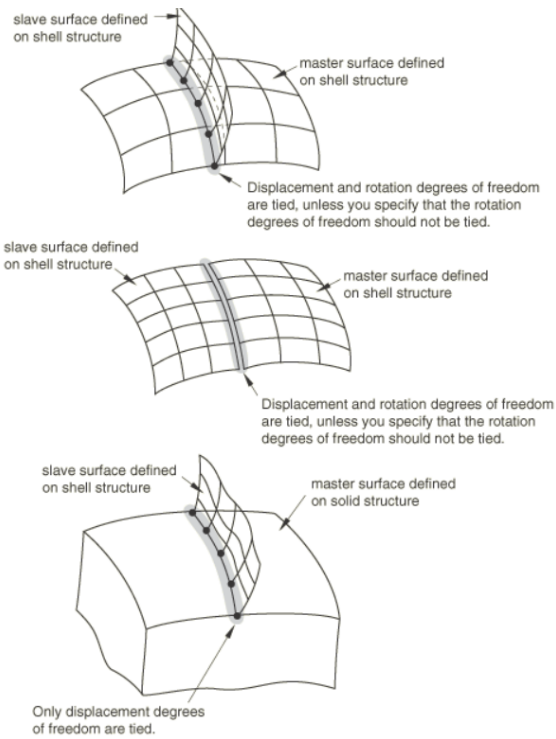

Figure 2-8: Surface-based Tie Algorithm (https://abaqus-docs.mit.edu)

Tie constrain is suitable while two components are surfaces. Displacements and rotation degrees of freedom are tied, unless the designer specify that the rotation degrees of freedom should not be tied. But if master surface defined on solid structure, only displacement degrees of freedom are tied. For interaction between load and fuselage, constraint coupling is used because it is possible to limit some of the nodes in the fuselage to the reference node. The reference node here is the point where load is applied.

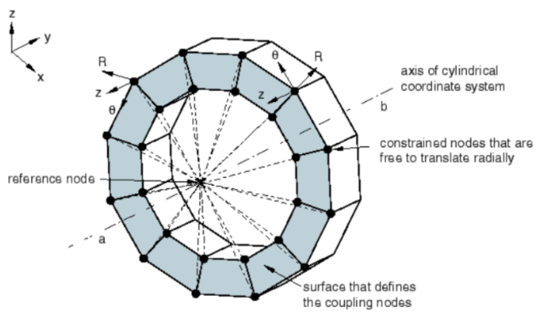

Figure 2-9: Kinematic Coupling Constraint (https: // abaqus-docs.mit.edu)

The coupling constraint is useful when coupling nodes on fuselage structure is constrained to the rigid body motion of single node. This coupling constraint effectively used in application of load or boundary conditions to a model.

3 Result and Analysis

To design composite fuselage, a stress analysis is performed finite element using Abaqus Software. Abaqus simulation allows to analyze the strength of composite material that used to fuselage structure. Property manager allows to set required number of layers, thickness, orientations and layer materials. This tool shows laminate coordinate system and direction of each individual layers. In considered case numbers of layers was set to 4. Thickness of composite layers amounted 0.125 mm on skin so that total thickness of skin is 0.5 mm. the frame bulkhead is made thicker with 1.25 mm so that total thickness of frame bulkhead is 5 mm. The results of stress analysis on the LSU 05 NG fuselage can be seen from Figure 3-1 to Figure 3-6. The stress analysis is presents in maximum principle stress to evaluate which stress is maximum in each direction. The Tsai-Hill failure criterion was used to check whether the structure is fail or not in all direction. The structure can be claimed to be safe and has no failure is the value of Tsai-Hill criteria is below 1. The stress value is presented to give some illustration about the stress distribution in the fuselage. From the stress distribution can be seen the location of maximum or critical condition that occured in the fuselage structure.

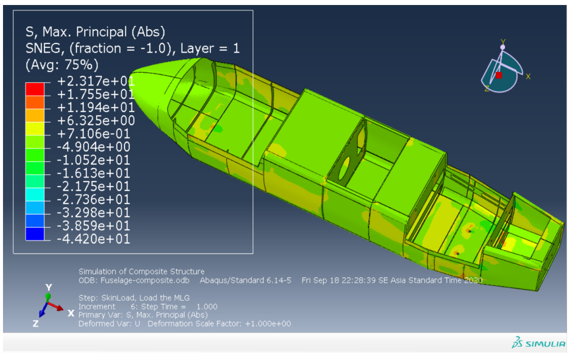

Figure 3-1: Max. Principal Stress for Configuration 1

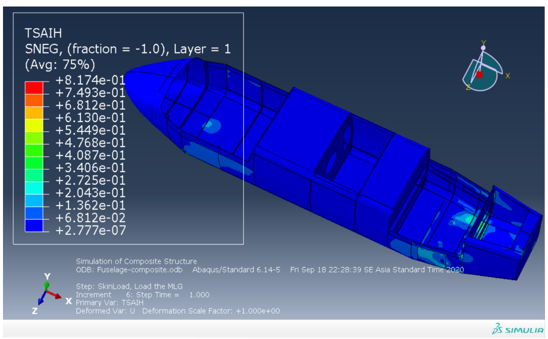

Figure 3-2: Tsai-Hill Criterion Result for Configuration 1

Figure 3-1 shows the stress distribution on the fuselage at take-off condition. The stress distribution that presented is maximum principle stress. The maximum tension stress is 23.17 MPa while the maximum compression stress is 44.2 MPa. The Tsai-Hill value can be seen on Figure 3-2. The maximum Tsai-Hill value is 0.81 at the main landing gear bulkhead.

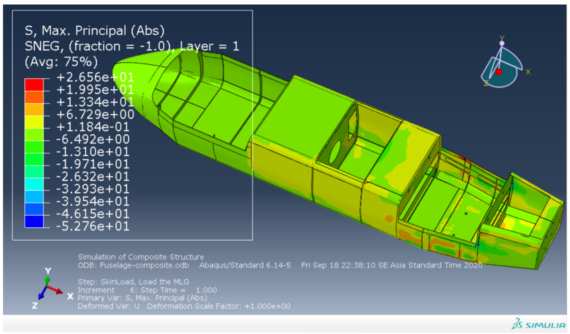

Figure 3-3: Max. Principal Stress for Configuration 2

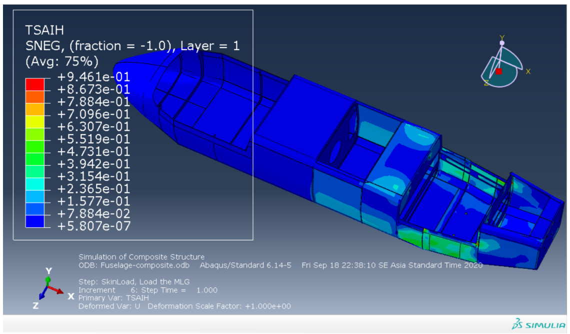

Figure 3-4: Tsai-Hill Criterion Result for Configuration 2

Figure 3-3 shows the stress distribution on the fuselage at cruise condition. The load that applied at this condition is assume by 2G load factor due to its maneuver. The stress distribution that presented is maximum principle stress. The maximum tension stress is 26.5 MPa while the maximum compression stress is 52.76 MPa in the negative sign. To investigate its failure the Tsai-Hill failure criterion is used. Figure 3-4 shows the Tsai-Hill distribution along the fuselage. The maximum Tsai-Hill value is 0.946 at the wing joint, so it is the critical point that failure might happen.

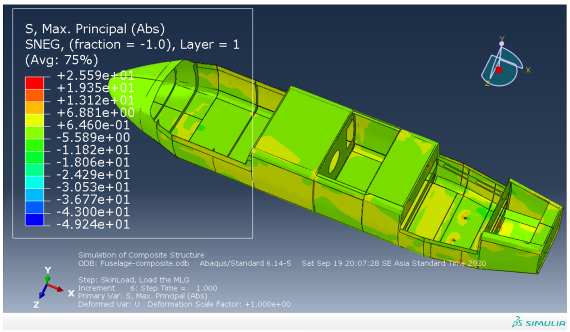

Figure 3-5: Max. Principal Stress for Configuration 3

Figure 3-5 shows the stress distribution on the fuselage at landing condition. The stress distribution that presented is maximum principle stress. The maximum tension stress is 25.59 MPa while the maximum compression stress is 49.24 MPa. The Tsai-Hill value can be seen on Figure 3-6. The maximum Tsai-Hill value is 0.92 at the main landing gear bulkhead. It can be concluded that landing condition is more critical in terms of structure rather than take-off condition. The skin structure near the landing gear bulkhead might fail due to the stress flow from the landing gear impact.

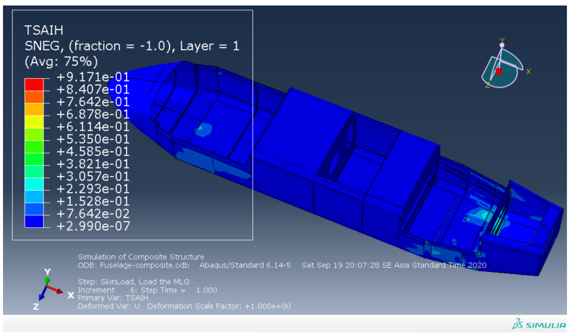

Figure 3-6: Tsai-Hill Criterion Result for Configuration 2

The maximum values of stress from all scenario are tabulated into table 3.1. the overall results is dominated with blue color that indicated the minimum stress is occured in the majority of the structure. It is one point that can be the reason to do the optimization to minimize the weight of the fuselage in the next research. The maximum stress occured in the main landing gear attachment or on the boundary condition since this area is critical.

Table 3-1: Maximum Values for Von Mises Stress and Tsai-Hill Criterion

Loading Configuration **#**	Max. Principal Stress (Tension) [MPa]	Max. Principal Stress (Compression) [MPa]	Maximum Tsai Hill Criterion
#1	23.17	44.20	0.82
#2	26.56	52.76	0.95
#3	25.59	49.24	0.92

The maximum principal stress on fuselage is 52.76 MPa (compression), which occurs at cruising configuration. The value of Tsai-Hill Criterion is 0.95, less than 1. The working stress value is lower than maximum stresses of the material. It can be concluded that fuselage of LSU-05 NG is capable to endure loading at its operations. The result of numerical analysis can be used as a reference for next iteration design so that the fuselage structure is obtained that satisfies the requirements specified in the LSU 05 NG Project, such as weight reduction.

Development of fuselage for unmanned aerial vehicle is a multistage task with includes: geometrical calculations, structural design, materials selection, numerical analysis, and elaboration of technology. Numerical investigation plays an important role because it can predict the structural strength of a fuselage when the load applied. it still needs validation with an experimental investigation so the design results of the LSU 05 NG's fuselage are completely able to do flight tests.

4 Conclusions

From the results of simulation and analysis, it can be concluded that,

a. E-Glass WR185/Epoxy 4 ply composite can be used as material for fuselage structures to endure loadings in critical conditions.
b. Maximum principal stress occurred at loading configuration #2, with value 52.76 MPa (compression).
c. Maximum Tsai-Hill criterion is 0.95. The value is less than 1, so the structure is considered safe at the loading configuration.
d. The fuselage can be optimized to minimizing concentrated stress at boundary condition
e. Several frame bulkheads can be optimized to reduce the weight of the LSU-05 NG's aircraft.

Reinforcement at any critical area is needed to make sure the airframe is not fail while in its operation. The thickness can be increase by add-up some plies at the skin or bulkhead. It's will be better by design sandwhich structure or combine the laminate using balsa to increase its moment inertia so the maximum stress can be reduced.

This Paper has presented a composite structure used in LSU 05 NG's fuselage and configure the load applied on fuselage. It has been found that Abaqus/Explicit is able to predict failure by tsai-hill criterion and von mises stress. Further, experimental investigation is needed to gain confidence about the strength of fuselage structure.

Acknowledgements

The authors would like to acknowledge the support of Mr. Gunawan Prabowo as the Head of the Aeronautics Technology Center – Indonesian National Institute of Aeronautics and Space (LAPAN) and Mr. Atik Bintoro as the Program Director of Maritime Surveillance System (MSS). We also thank the entire LSU 05 NG team which helped the research process.

Contributorship Statements

MH and AN conceived of the presented idea. MH, AN, and RA compiled post-processing method. MH performed numerical simulations, data analysis, and interpretation. AN and RA contributed to the final version of the manuscript.

References

ABAQUS/Explicit: Advanced Topics, (2005). *Lecture 7: Constraints and Connections*, https://www.researchgate.net/profile/Mehdi_Ebadi_Jamkhaneh/post/How_we_apply_tie_material_between_two_layers_of_different_material_components_in_abacus_software/attachment/5a408fae4cde266d587d8007/AS%3A575328980566016%401514180526271/download/l7-connections.pdf

Abdul-Hamead. A.A, Kasim. T, Mohammed. A. A, (2014). *Mechanical Properties for Polyester resin Reinforce with Fe Weave Wire*. International Journal of Application or Innovation in Engineering & Management (IJAIEM) ISSN: 2319-4847 Vol. 3, Issue 7

Abdurohman. K, Marta. A, (2016). *An Experimental Study of Polyester Composite Tensile Properties Reinforced Undirectional Carbon Fiber Manufactured by Vacuum Infusion for LSU Materials*. Jurnal Teknologi Dirgantara Vol. 14 No. 1 Juni 2019 :61-72

Ariyanto. M, Setiawan J. D, Prabowo. T, Haryanto. I, Munadi, (2018). *Design of a Low-Cost Fixed Wing UAV*. IJCAET & ISAMPE 2017. MATEC Web of Conferences 159, 02045 (2018)

Chen. Y, Lu. X, (2018). *Numerical simulation of unmanned aerial vehicle under centrifugal load and optimization of milling ang planning*. AIP Conference Proceedings 1967, 040062 (2018)

Chung. P. H, Ma. D. M, Shiau. J. K, 2019. *Design, Manufacturing, and Flight Testing of an Experimental Flying Wing UAV*. Applied Sciences

Dassault Systemes, (2010). *Introduction to Abaqus*. Abaqus User Documentations.

Grodzki. W, Lukaszewicz. A, (2015). *Design and Manufacture of Unmanned Aerial Vehicles (UAV) Wing Structure using composite materials*. Mat.-wiss. U. Werkstofftch, 46, No. 3

Hadjez. F. And Necib. B, (20150. *Stress Analysis of an Aircraft Fuselage with and without Portholes using CAD/CAE Process*. Journal of Aeronautics & Aerospace Engineering Volume. 4 Issue 1

Hibbeler. R, (2011). *Mechanics of Materials*. Pearson Prentice Hall.

Isna. L. R, Abdurohman. K, Ula. N. M, Wijaya. Y. G, (2019). *Tensile properties E-Glass ±45° Fiber with Matriks Polyesther Composite as Structure of LSU*. Iptek Penerbangan dan Antariksa: Progres Litbangyasa Roket, Satelit dan Penerbangan 2018

Raja. D. B. P, et.al, (2018). *Stress analysis of Al2014-T6 riveted splice joint fuselage structure using finite element analysis*. IOP Conf. Series: Materials Science and Engineering 455 (2018) 012138

Singhanart. T, Srimontok. C, Pisitpan. N, Chitimaworaphan. S, Mongkhonchaiwiwat. W, (2012). *Design and Analysis of UAV Fuselage*. Applied Mechanics and Materials ISSN: 1662-7482 Vol. 225, pp 305-309

Velasco. M. F, Stampfli. E, Fardoun. H, Balboa. M. D, (2016). *Mechanical Design Project: Conceptual and Preliminary Design of an Unmanned Aerial Vehicle*. Universidad Europea de Madrid.