International Journal of Smart Energy Technology and Environmental Engineering
Volume 2, Issue 1, March 2023
http://globalpublisher.org/journals-1007/

Modeling of Drone Structures Based on Composite Materials

Murthadha Kamil Abed Aljanabi [1,*]
1- Director of Al-Nisoor office for Engineering Consultations Dhi Qar Governorate, Nasiriyah
*Corresponding Author: murtadhakamil1985@gmail.com

ABSTRACT

The focus of this study pertains to the modeling of Unmanned Aerial Vehicle (UAV) structures utilizing composite materials. Composite materials are frequently employed in the fabrication of high-impact components owing to their favorable dynamic attributes and superior mechanical properties, including a commendable strength-to-weight ratio, notable stiffness and resistance to fatigue, as well as exceptional corrosion resistance. The initial phase of the analysis entails conducting a three-point bending test on specimens composed of composite materials. The test is subsequently simulated using the Samcef software. The congruence between empirical tests and the finite element model yields the inference that the material properties furnished by the manufacturer are accurate and faithfully depict the material's actual performance. The subsequent phase involves the formulation and construction of a finite element model encompassing the entirety of the aircraft. The model's validity is assessed through the execution of a preliminary bending test on the UAV. A comparison is made between the displacements observed in the model and those obtained in the actual experiments. The findings indicate that the model exhibits a marginal increase in rigidity compared to the actual aircraft, as anticipated. This outcome facilitates the verification of the finite element model's results. The ultimate phase entails the simulation of a verisimilar flight configuration wherein the aircraft is exposed to substantial aerodynamic forces. The designated role pertains to the resource allocation associated with the aircraft's descent for retrieval purposes. Aerodynamic loads are computed using the PanAir software and subsequently incorporated into the finite element model. The findings indicate that the structural integrity of the system is capable of withstanding the applied loads. However, it is recommended to explore potential material optimization strategies, as certain components exhibit stress levels that exceed their yield strength.

Keywords: Unmanned aircraft, composite, bending, strength, aerodynamics.

1. INTRODUCTION

The increasing prevalence of unmanned aerial vehicles (UAVs) necessitates engineers to efficiently develop and produce these products within constrained timeframes and at cost-effective prices. The feasibility of achieving this outcome can be attributed to the extensive accessibility of contemporary composite materials and advancements in computer systems. The CAD environment provides the opportunity to generate a virtual model. Furthermore, by employing computer-aided engineering (CAE) tools, it becomes feasible to replicate numerous characteristics and actions exhibited by tangible entities. In numerous instances, it is imperative to juxtapose the outcomes of numerical analysis with empirical data. The utilization of computer-aided design and manufacturing (CAx) tools allows designers to significantly reduce the cost associated with the implementation of a new product [1, 2].

The utilization of composite materials, particularly polymer matrix composites reinforced with continuous fibers, is deemed the most suitable option for modern unmanned aerial vehicles (UAVs) due to their ability to meet the demanding requirements of high strength and low weight [3]. These materials exhibit a Young's modulus that is twice as high as that of aluminum alloys, while maintaining a weight that is two times lower. The challenges associated with the utilization of composite materials are primarily attributed to their anisotropic/orthotropic nature. The utilization of these materials in the field of numerical analysis necessitates a comprehensive understanding of the material constants and mechanical properties in the principal axes. The assurance of genuine structural behavior and the reliability of calculation results can only be achieved through the utilization of fully defined composite materials [4].

The process of developing wings for unmanned aerial vehicles (UAVs) is a complex and multifaceted endeavor that encompasses several stages. These stages involve the careful selection of airfoils, meticulous geometrical calculations, precise structural design, thoughtful materials selection, rigorous numerical analysis, and meticulous manufacturing techniques. The primary objective of the wing development process is to conceive a structural design that exhibits a combination of superior strength and minimal weight.

Composite materials are comprised of multiple phases that exhibit distinct properties, both in terms of their physical and chemical characteristics. When these components are amalgamated, they give rise to a composite material that exhibits distinct properties in comparison to its constituent elements. Each phase remains distinct and independent within the final structure. The constituents of composites can be categorized into two main components based on their respective roles: filler, also known as reinforcement, and matrix. The matrices materials commonly employed in various applications encompass polymers, metals, ceramics, and carbon. Conversely, fillers materials, which serve to enhance the properties of the matrices, include glass, carbon, aramid, and boron.

Reinforcement manifests itself in various forms, including continuous fibers, short fibers, and particles [4].

2. METHODOLOGY

This research demonstrates practicality from a subjective perspective. The design of this approach is rooted in a quantitative methodology. Regarding research tools, it possesses a descriptive nature. The utilization of the research library method, along with the incorporation of industry experts' opinions, will be employed as a means of data collection due to financial constraints and inadequate resources for constructing a conceptual model.

In terms of the model's design, we will utilize the CATIA Engineering application software, which facilitates 3D design. Additionally, if the necessary resources are available, including the provision of funds for creating a conceptual model, the construction of a drone prototype may also be feasible.

A conceptual model can be constructed based on the dependent variable and the independent variable.

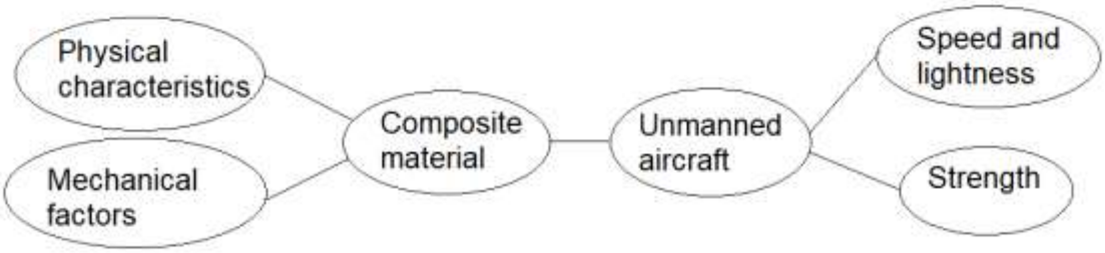

Fig. 1. Conceptual model of the plan

3. SIMULATION

The initial stage of finite element model development involves acquiring accurate geometries that yield extensive meshes, thereby ensuring their fidelity to real-world conditions.

Refiteh employed a three-dimensional model in order to facilitate the design of the mold that was utilized for shaping the composite sheets. The model is depicted in Figure 2. The original intention was to implement this model in Samcef due to its numerous advantages. The process of developing a new model is a time-consuming endeavor, and in this particular instance, the model under consideration closely resembled the existing drone geometry. Nevertheless, it was necessary to undergo modifications due to various factors.

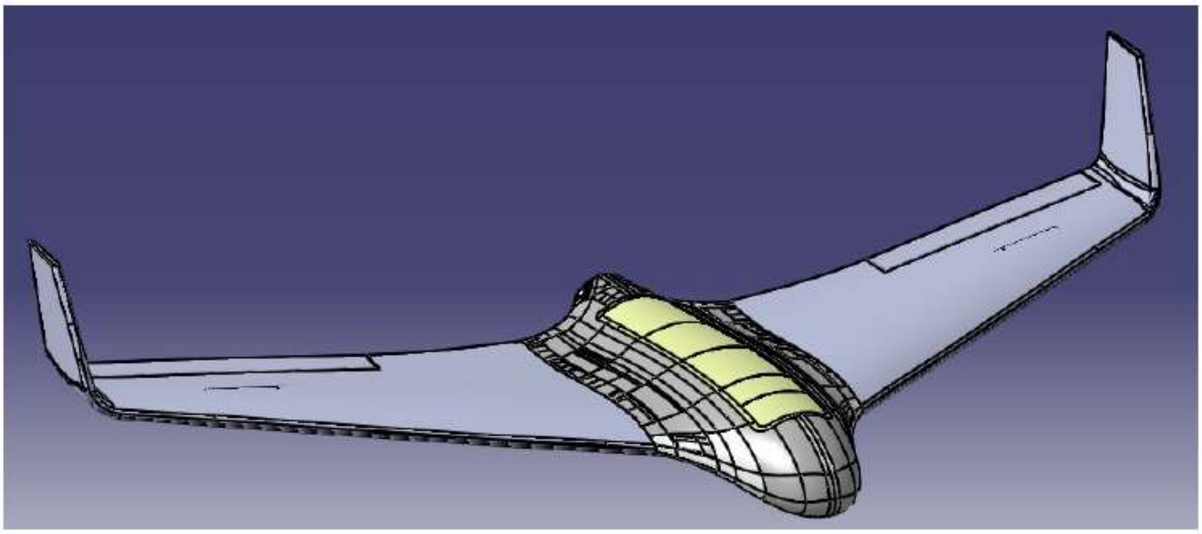

Fig. 2. UAV modeling in CATIA

The initial issue arose as a result of the methodology employed in constructing the model. The model was created by the community utilizing a three-dimensional scanner. The efficacy of this approach lies in its expeditiousness and precision, albeit yielding a model characterized by numerous intricate facets and edges, particularly along the leading edge of the wing and the fuselage. This is illustrated in Figure 3. The process of meshing the structure in Samsef is associated with various challenges. In order to ensure proper meshing, it is necessary for Samcef to allocate a minimum of two nodes along each edge, with one node positioned at each respective end. The irregular and disordered nature of the mesh was a result of the presence of significantly smaller edges in comparison to the average inter-nodal distance.

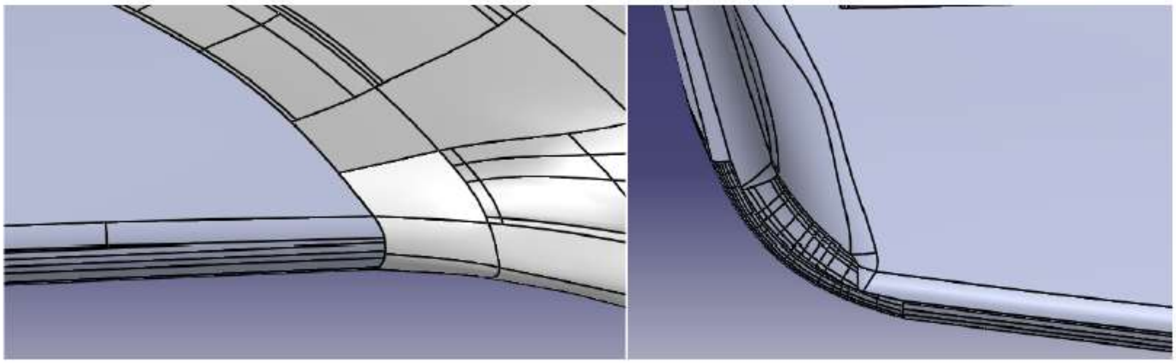

Fig. 3. Complexity of the CATIA model

The second issue pertained to the composition of the model, which consisted of disconnected surfaces upon import. In Samcef, the establishment of connections between them was feasible through the implementation of appropriate boundary conditions. However, due to the substantial number of faces involved, this process would be time-consuming and labor-intensive. The resolution of this issue can be facilitated through the utilization of a solid model imported from Ketia, thereby circumventing the potential errors that may arise from manually configuring the boundary conditions.

The geometry within CATIA has been modified to reduce its surface area and create a more refined mesh. Simpler surfaces have been implemented to replace the leading and trailing edges of the wings, while also streamlining the connection between the wing and the fuselage. The alterations made do not alter the drone's geometry; rather, they significantly streamline the model's intricacy. The novel geometric configuration is visually depicted in Figure 4.

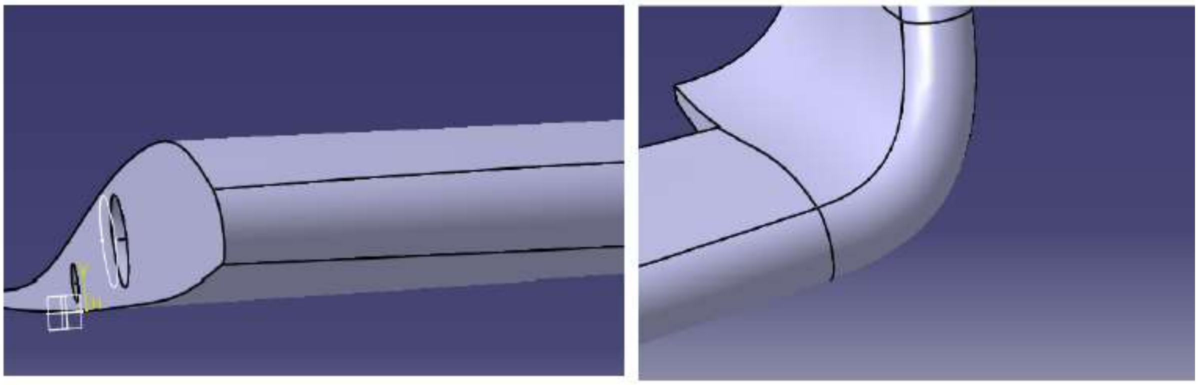

Fig. 4. Simplified CATIA model

The final aspect to be modified is not a genuine issue, but rather a resourceful approach to problem-solving. Given the inherent symmetry of the drone with respect to the Y-Z plane, it becomes feasible to analyze solely one half of the drone. By employing suitable boundary conditions at the midpoint of the fuselage, it becomes possible to simulate the symmetrical behavior of the entire drone. The increased number of nodes within the mesh contributes to enhanced computational efficiency, resulting in faster calculations. Simulating symmetry can be achieved by restricting displacements along the Y axis. In the event that an aircraft is operating with a slip angle, it is important to note that the assumption of a symmetrical configuration becomes invalid. Under these circumstances, the aerodynamic forces acting on the right and left sides of the drone exhibit asymmetry. Nevertheless, the current analysis did not consider this particular flight configuration, and it is possible to hypothesize that symmetric airflow and symmetric load distribution were assumed.

The model can be enhanced by incorporating simplifications that facilitate the simulation process. The initial aspect pertains to ailerons. The ailerons are affixed to the wing through the use of a slender membrane. It can be posited that the interaction between the wing and the airfoil is minimal, thus allowing for their separation. The wing pockets housing the servos are enveloped by a slender layer of carbon fiber composite material. They may also be eliminated based on the assumption of zero effort required. This hypothesis is also employed for the purpose of eliminating the body covering.

The construction of the model necessitates the fabrication of multiple components, which are subsequently manufactured separately and subsequently integrated together within Samsaf. The essential components required for the modeling process encompass a distinct body structure, a wing element, an internal support mechanism within the body to facilitate wing attachment, and ultimately, a connecting rod that links the wing to the body.

The provision of support was not facilitated by Refitech, thus rendering the unavailability of a 3D model. The manual modeling process was conducted using Samcef software. The support structure consists of a carbon fiber composite shell in the shape of an L, with an additional aluminum alloy cylinder integrated within the body. The model utilized for demonstration is depicted in Figure 5.

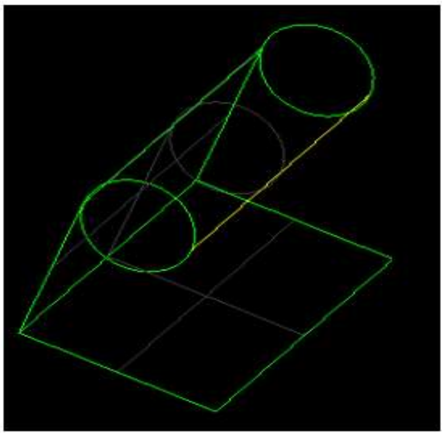

Fig. 5. The support model used in Samcef

The attachment of the cylinder to the composite shell is facilitated by the utilization of two rivets. Nevertheless, the finite element model encounters difficulties in accurately representing the behavior of rivets, resulting in the simplification of their contact as linear. However, it is crucial to acknowledge that the actual stress levels in this region are expected to be higher due to the presence of rivets, which induce stress concentration.

An additional component should be constructed to accommodate the wing structure. It is important to note that the wing is internally filled with polyurethane foam, making it impossible to visually depict both the outer skin and the foam within a single entity. The skin model is subsequently replicated and transformed into a solid element possessing the characteristics of the foam material. The simulation then proceeds to examine the interactions between the foam and the skin.

In summary, the different entities can be identified as follows and are interconnected through the specified boundary conditions:

The wing composite skin is a structural component that emulates a carbon fiber composite skin.

The wing solid model is employed for the purpose of simulating the foam material located within the wing structure. The attachment of the object to the skin is achieved through the process of affixing skin elements onto it.

- The composite skin is represented using shell elements in the body.
- Support: The support is securely attached to the body through the utilization of the "stick" mechanism.

The carbon fiber composite strip is represented by a shell element in the model. The rod is affixed to the wing through the use of foam, which is inserted into the wing's solid model to replicate the attachment. The connection between the two entities is established through the inclusion of a "contact" constraint.

It is imperative to acknowledge that a fundamental distinction exists between the act of bonding elements together and the process of simulating contact between them. The process of gluing refers to the act of connecting elements together, resulting in the nodes at the interface of the two objects exhibiting identical displacements. The term "contact" refers to the condition in which two elements are in physical proximity to each other and have the potential for interaction. Interaction occurs only when the first element exerts pressure on the second. Conversely, if tension is applied between two elements that are in contact, no interaction takes place. The utilization of the contact constraint is highly effective in simulating the interaction between the support and the bar. Conversely, the application of the glue option may result in inaccurate constraints and displacements.

Several additional boundary conditions have been incorporated into the model in order to enhance its realism. A contact constraint is introduced to establish a connection between the lateral surface of the fuselage and the tip of the wing. Indeed, the wing and fuselage are not solely linked by the rod, but rather they are in direct contact with each other. Consequently, this region is subject to numerous limitations. Given that the drone was still in the developmental phase and at the prototype stage during the time of analysis, it can be inferred that the wing and fuselage were in direct physical contact with each other. The utilization of a joint in the commercialized model serves the purpose of mitigating direct contact between composite parts. The rivet located in front of the wing is not accounted for in the rivet model, yet it is necessary to include in the modeling process. The wing-to-body connection is a fixed joint that restricts wing rotation during flight. The model is constructed by adhering certain nodes of the wing to the body.

The ultimate model employed is depicted in Figure 6.

Fig. 6. The complete model of the UAV in Samcef

4. RESULTS AND DISCUSSIONS

validation

A preliminary pilot study is conducted in order to assess the reliability and accuracy of the model. In the initial phase, it is recommended to incorporate the procedure of sealing the fuselage and subsequently subjecting the wing tip to a vertical force. Nevertheless, the closure of the body presents a challenge, leading to the selection of a more straightforward test.

The drone is structurally reinforced by roller supports positioned at both ends. Weights are strategically positioned within the fuselage in close proximity to the center of gravity, and the resulting displacement is quantified.

The force-deflection values obtained from the experimental tests and the finite element model can be overlaid in order to assess their congruence. The graphics identified are presented in Table 1.

Table 1. Comparison of experimental behavior and finite hybrid model results

crosshed displacement [mm]	0	2	4	6	8	10	12	14	16	18	20
force [N]											
experimental tests	0	2	43	79	98	116	118	124	110	100	85
finite element model	0	2	43	80	100	120	118	125	114	-	-

It is evident that the slope of the linear segment of the curves exhibits a near equivalence, with the actual samples displaying only a marginal increase in stiffness. The discrepancy in the displacements of the finite element model is approximately 2%. The correctness of the Young's modulus employed in the finite element model can be inferred.

It is worth noting that a slight disparity exists between the experimental curves and the finite element model, which can be attributed to the limited nonlinear range exhibited by the actual samples.

In the finite element model, a red cross is placed when the value of the Tsai-Hill criterion is equal to 1. The value of Tsai-Hill approaches 1 as the linear behavior of real samples nears its conclusion. This implies that the yield strength values employed correspond to the point at which the material's linear elastic response ceases, rather than indicating its failure. Next, the validity of the tensile and shear strength values is assessed. The acquisition of nonlinear behavior in real samples occurred at a slightly later stage. This implies that the resistance values presented by the manufacturer were slightly conservative, yet it offers a substantial safety buffer.

The mechanical properties of T700S carbon fiber composite have been verified through these experimental tests.

Finite element model results

In the context of our analysis using a basic finite element model, the focus lies on the holistic response of the drone. Consequently, the system's behavior can be predominantly characterized as linear. To construct the force-displacement diagram, we employ three distinct configurations. The observed behavior exhibits non-linearity due to the numerous interactions among elements, joints, and rivets. The practice of determining deformations for multiple weights using the finite element method lacks coherence as it fails to provide an accurate representation of reality.

The force-displacement diagram is depicted for three different magnitudes of force: 0 newtons, 78 newtons, and 157 newtons. The findings are depicted in Figure 7.

A graphical representation can be employed to contrast the disparities in slope between the experimental and simulated curves. As previously elucidated, the significance of the slope of the curve surpasses that of the deviation in question. The curves obtained are transformed within the finite element model in order to facilitate the comparison of their slopes. The findings are presented in Table 2.

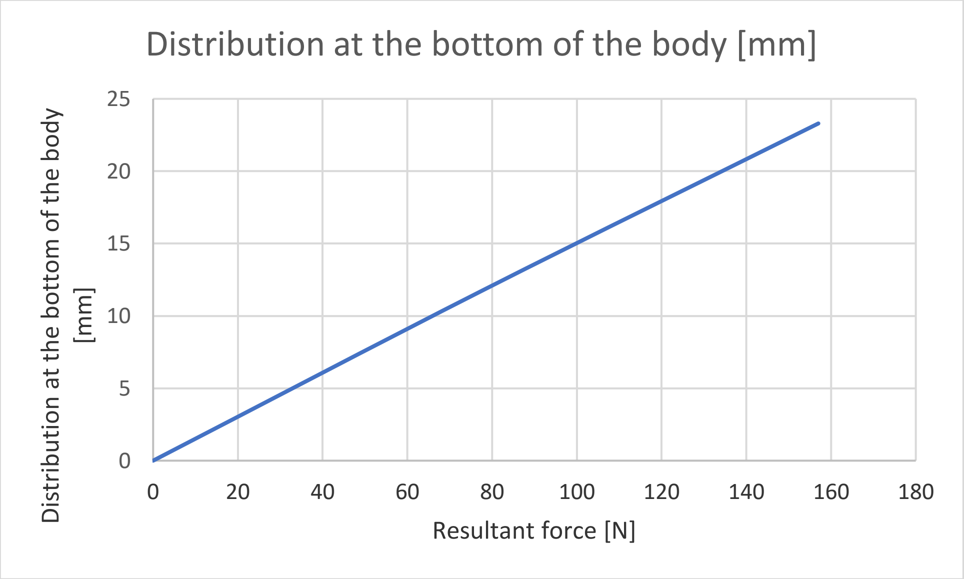

Fig. 7. Shifts in the intention element model

Table 2. Comparison of force-displacement

Displacement of the point [mm]	0	5	10	15	20	25
force applied [N]						
experimental tests	30	60	95	118	139	160
finite element model	30	62	100	130	165	188

The table illustrates that the actual unmanned aerial vehicle (UAV) exhibits greater flexibility due to the relatively higher magnitude of deformation observed. This phenomenon can be attributed to the omission of rivets and joints in the finite element model. Moreover, as demonstrated in the preceding chapter, the characteristics of the finite element model result in a composite material that exhibits a slightly higher stiffness compared to the actual material.

Upon analyzing the displacements observed on the Samsef, it becomes evident that a minor roll of the fuselage is taking place. This is indicated by the discrepancy in displacement between the nose of the fuselage and the lower section near the aircraft's center of gravity. The obtained results appear to validate the previously posited hypothesis regarding the relocation of the fuselage nose.

5. CONCLUSION

it is intriguing to examine the constraints of the finite element model. The investigation focuses on determining the limits associated with an applied force of 157 N, as this particular configuration places the highest demands on the unmanned aerial vehicle (UAV).

The observation of the wing tip reveals an intriguing phenomenon: despite the absence of anticipated stress, the skin is subjected to considerable constraints, which deviates from the expected value of near-zero.

This issue arises due to the implementation of the finite element method. The software aims to maintain continuity of tension and kinematic constraints at the boundary between the foam and the skin. The resolution of this problem is deemed unattainable, with the software generating a heightened level of tensions in its attempt to address it. The elevated stresses experienced at the wing's trailing edge are, in fact, artificially induced. This issue can be resolved by eliminating the constraints associated with attaching the surface to the wing, thereby transforming it into a direct contact between the skin and the foam material. The simulation involves replicating the interaction between the foam and the shell at the front edge. These modifications have no impact on the outcomes, except for the elimination of imposed limitations. The visibility of limits has been enhanced due to their scalability in relation to the maximum stress level.

Another noteworthy aspect to observe is the region where the support is connected to the body. There is a limited degree of tension concentration. The limitations observed in this mechanism are a result of the upward force exerted by the composite rod on the support, which in turn induces a tendency for the support to pull the body upwards. Nevertheless, it is imperative to acknowledge that in actuality, the demands placed upon the support system ought to be more substantial. The support exhibits a lack of secure attachment to both the body and the aluminum rod. Rivets are employed in various applications, resulting in a notable accumulation of stress concentrations. When dealing with composite materials, it is crucial to acknowledge that even minor imperfections occurring during the process of drilling rivet holes can lead to a substantial amplification of stresses at those specific locations when subjected to heavy loads. Due to this rationale, the experimental testing is constrained to a maximum force of 157 Newtons, thereby facilitating the observation of deformations while maintaining a safe distance from the material's threshold.

The aluminum cylinder can be subject to investigation due to its isotropic nature. The equivalent von Mises stress is employed for analysis, with a maximum value of 54 MPa observed, while the yield strength is estimated to be approximately 241 MPa. The observed properties of this material align with the anticipated elastic behavior.

REFERENCES

1. K. Panas, A. Łukaszewicz, Machine Dynamics Research 2013, 37, 45.
2. Łukaszewicz, Z. Koruba (ed.), Scientific aspects of unmanned mobile vehicle. Polish Society of Theoretical and Applied Mechanics, 145, 2010.
3. Ch. Kassapoglou, Design and Analysis of Composite Structures with Applications to Aerospace Structures, Wiley, 2013.
4. D.D.L. Chung, Composite Materials – Science and Application, Springer 2010.