Finite Element Analysis on the Arms of Multirotor UAV

Abuzar Ulla Khan[1], Justus Shadrach J[2], Karan H [3], Vamshi B[4], Bhushan JK[5] & Nishanth P[6]

[1,2,3,4]Student, [5]Assistant Professor & [6]Professor
Department of Aeronautical Engineering
East West College of Engineering, Bengaluru, India
***DOI**: https://doi.org/10.34293/acsjse.v3i1.57*

Abstract - *As the development and applications of UAV is been increasing significantly in various fields. The main focus is towards the design and structural analysis of the UAV. As the initial step of initialization of any UAV prior condition is to design.*
The structural analysis of a UAV frame is important since it has to withstand the forces due to aerodynamic and gravitational forces due to the mounted weights. As design optimisation plays an important role for an efficient structure, a finite element analysis provides an efficient methodology to meet the desired objectives related to structural elements. This project mainly focuses on the structural loads and vibrations acting on the arms of multirotor UAV.
Finite element analysis is been carried out on the arms of multirotor UAV to understand the structural loads acting on the arms of the UAV. A small deformation occurred on the arms are safe and within the limit. Therefore, this study aimed to analyse the strength and durability of UAV arms during the in-flight conditions through the analysis.
***Keywords**: UAV, Finite Element Method, Composite Material*

I. INTRODUCTION

UAVs or Unmanned Aerial Vehicles, if described in simple words are aircrafts, where the pilot is replaced by computers and radio-links. But it is much more than “pilot-less” aircraft in real scenarios. Design and development of UAVs involves complex design process, so that it can be efficient enough to fly, with its aircrew on ground [1].

In recent years, there is a huge demand for unmanned aircrafts for both military and civilian applications. Due to its diverse nature, the application of UAVs ranges from monitoring a hostile war situation to spraying chemical on crops. In the last decade, there was a huge technological advancement which encouraged the manufacturers in massive investment in research and development to produce cost effective, feature rich and multi-functional UAVs [2]. Functioning of UAVs in dull, dangerous and dirty environment makes it more competent as compared to its manned counter-part. Since unmanned aircrafts offers low risk and high efficiency in its mission, there is a steady growth in the field of UAV and its components or sub-systems. With sizes of sensors getting smaller, there are endless opportunities to mount various payloads suitable with the mission. Though unmanned aircrafts have great advantages, the development of UAVs is complicated and time consuming [3].

For both manned and unmanned aircrafts, Airframe refers to the mechanical structure that houses the wings, fuselage, undercarriage, etc. for manned aircrafts and sensors, payloads, on-board computers, etc. for unmanned aircrafts. The airframe, for both manned and unmanned aircrafts is actually the basic structure excluding the propulsion unit [4]. The

UAV airframe consists of three categories: Fixed Wing, Helicopter and Multi-Copter. Based on mission profile and requirement, the airframe of any UAV is chosen. Since, the space is considerably low in case of UAVs, the design of airframe should be such that all the required sensors and payloads can be mounted on the airframe, without compromising the aerodynamics or flight performance. Hence, the airframe plays a major role in the development of unmanned aerial vehicles [5].

II LITERATURE STUDY

Analysis of UAV airframe and estimating its strength is of very much important in development of any UAV. Thrust forces generated due to power plant, gravitational forces due to the carried payload, impact load and load due to winds act on the structural element of UAVs. Under both dynamic and static loading condition, the airframe should establish its structural integrity, keeping the size and weight intact. In short, the desired airframe of any UAV should be light weight and strong [6].

Analysis of the airframe gives an insight about the different forces and stress produced by those forces on the airframe. It helps to determine the rigidity of the frame, the maximum weight that the UAV can carry at the time of take-off and the necessary propulsion unit that the UAV can be equipped. Based on the above investigated parameters, it will be convenient to find out the performance parameters and stability of the unmanned aircraft [7].

Finite Element Analysis (FEA) or Finite Element Method (FEM), are numerical simulations which are done extensively for structural analysis. In other words, FEA/FEM involves application of mathematics to understand and evaluate structural, fluidic, thermal and wave behaviour, apart from other applications. FEA employs Partial Differential Equations (PDE) to solve any problem [8]. These equations solve problems in engineering mechanics like evaluating admissible quantities of a structure (stress, strain, etc.). For any given problem, FEA gives approximate solution and this predicted result will give an idea how the physical quantity will behave in real world.

III METHODOLOGY

Finite Element Method (FEM) is a numerical analysis technique used to solve complex engineering problems [12]. The methodology of FEM analysis can be summarized as follows:

- **Problem Definition:** The first step in FEM analysis is to define the problem and specify the boundary conditions. This involves defining the geometry, material properties, and the loads applied to the structure.
- **Discretization:** The next step is to divide the structure into smaller elements or subdomains. This process is known as discretization, and it involves breaking down the structure into simpler elements that can be easily analysed. This step is crucial in FEM analysis, as the accuracy of the analysis depends on the quality of the discretization.
- **Formulation:** Once the structure is discretised, the governing equations that describe the behaviour of the structure are formulated. These equations are typically

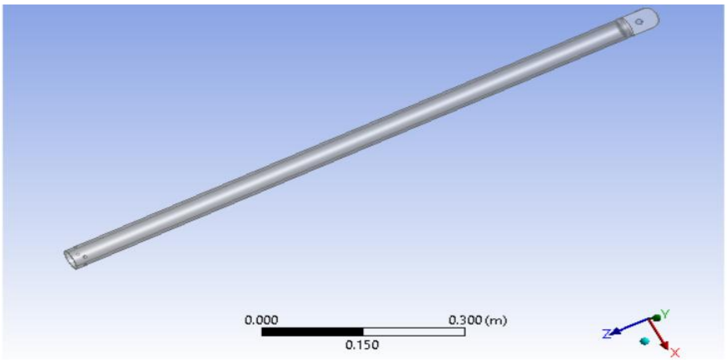
differential equations that describe the relationships between the various variables that affect the behaviour of the structure.

- **Assembly:** In this step, the individual element equations are assembled into a global system of equations. The resulting system of equations is a set of algebraic equations that can be solved using numerical methods.
- **Solution:** The global system of equations is solved using numerical techniques to obtain the values of the unknown variables. The solution provides information about the behaviour of the structure under the specified loads and boundary conditions.
- **Post-processing:** Once the solution is obtained, post-processing is carried out to analyse and interpret the results. This involves visualizing the results using graphical tools and extracting relevant information from the results.
- **Verification and validation:** The final step is to verify and validate the results obtained from the FEM analysis. Verification involves checking that the analysis has been carried out correctly, while validation involves comparing the results obtained from the analysis with experimental data or other analytical method [13].

IV EXPERIMENTAL DETAILS

Design of a Multirotor UAV

Design of the multirotor UAV arm is done by using CATIA software where the length of the multirotor UAV arm is 1000mm, outer diameter- 38mm, inner diameter-34mm.



Design of Multirotor UAV Arm

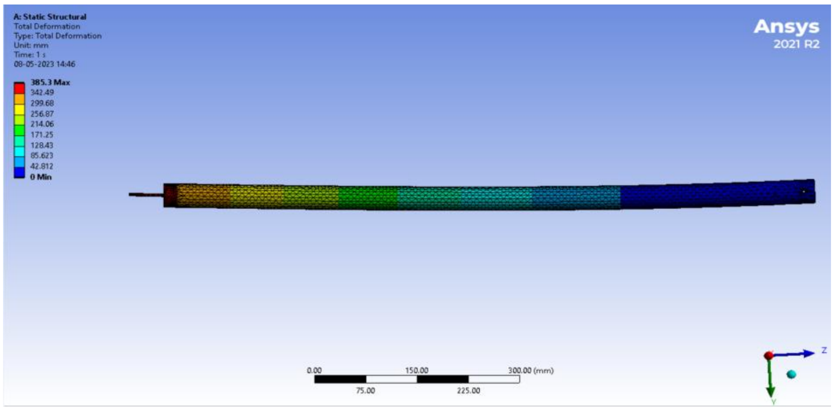
Finite Element Method Analysis

With the progress of unmanned aircraft technologies, the manufacturing techniques for the same have also improved. Though manufacturing techniques like Electrical Discharge Machining, Computer Numerical Control, etc were successful in manufacturing UAVs and their components, they were time consuming and were limited in manufacturing complex UAV structure [9].

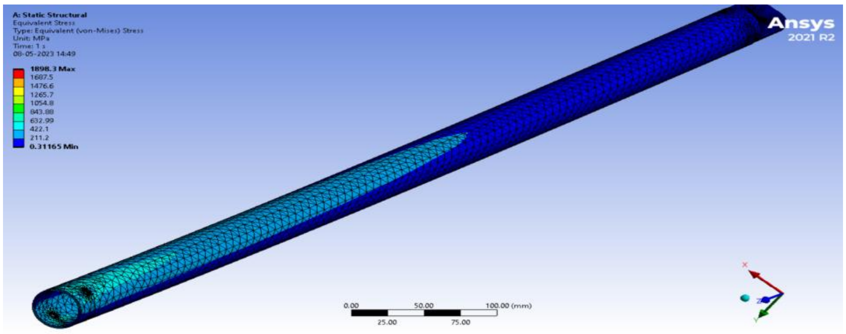
The material properties were analysed by Finite Element Analysis, for optimizing the stiffness and reducing the overall weight. The material selected to carry out the FEM analysis on the arms of multirotor UAV is CARBON FIBRE.

FEM analysis is carried out using ANSYS and COMSOL by taking boundary conditions as $q_1=0$(fixed end) and $q_2=1000$(free end). The forces acting on the multirotor UAV arm is considered to be 1000N [14].

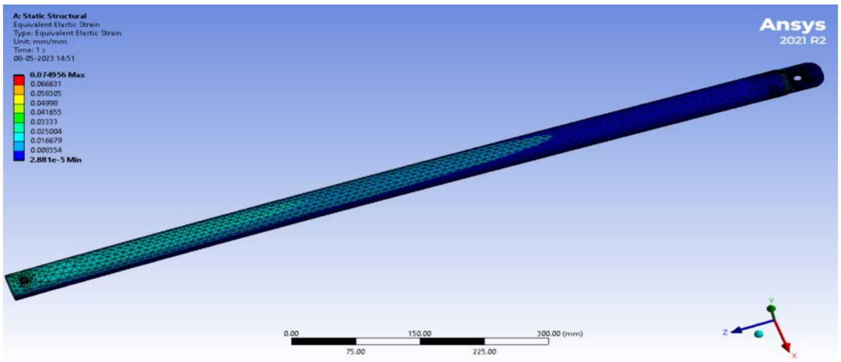
FEM Anlysis using ANSYS



Total Deformation of Multirotor UAV Arm

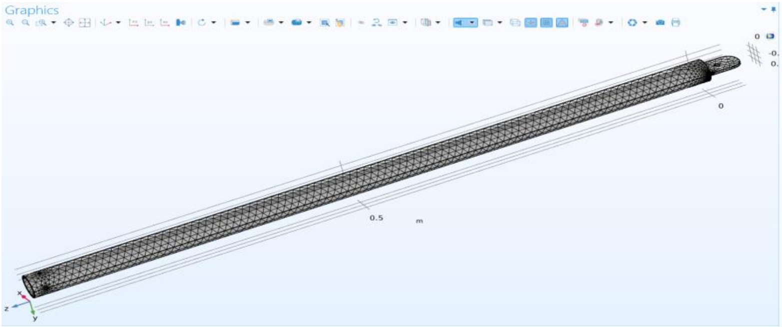


Total Stress Acting on the Arm of Multirotor UAV

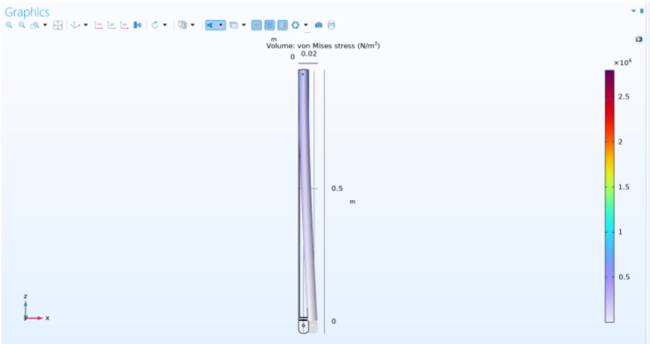


Strain Acting on the Arm of Multirotor UAV

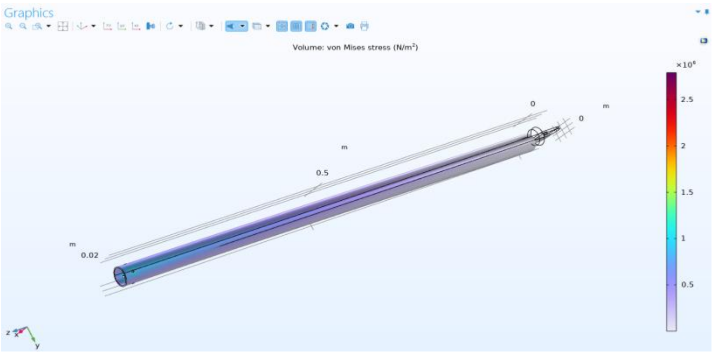
FEM Analysis using COMSOL



Meshing



Deformation of Multirotor UAV Arm



Stress Acting on the Arm of Multirotor UAV

Theoretical Calculations

Data

Young's modulus: $E=230\times10^3$ N/mm^2

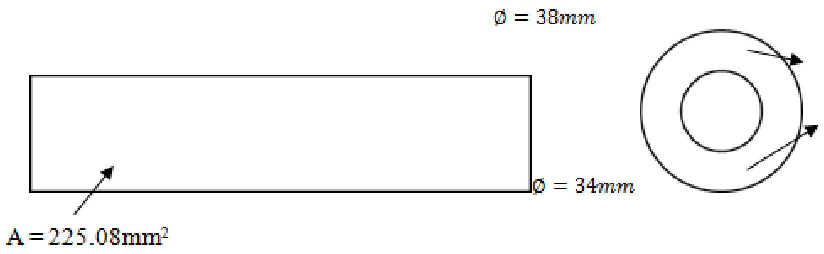
Outer diameter = 38mm

Inner diameter = 34mm

Thickness = 4mm

Length = 1000mm

Force(F) = 1kn



Area of UAV rotor arm

$A = \pi/4(D^2 - d^2)$

$A = \pi/4(38^2 - 34^2)$

$A = 225.08mm^2$

Element Matrix

$[K_e] = \frac{AE}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

$[K_1] = 5.17\times 10^4\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

Global Stiffness Matrix

$[K] = [K_1] = 10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}$

Global Nodal Displacement Vector

$(Q) = \begin{pmatrix} q_1 \\ q_2 \end{pmatrix}$

Global Load Vector

$(F) = \begin{pmatrix} F_1 \\ F_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 1000 \end{pmatrix}$

Equilibrium Condition

[K] (Q) = (F)

$10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}\begin{pmatrix} q_1 \\ q_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 1000 \end{pmatrix}$

Therefore, the nodal vector displacement is,

$(Q) = \begin{pmatrix} 0 \\ 0.019 \end{pmatrix}$

Stress in Each Element

$\sigma = \frac{E_e}{l_e}[-1\ 1]\begin{pmatrix} q_i \\ q_{i+1} \end{pmatrix}$

Where i = e and e is a element number

$\sigma = \frac{230\times 10^4}{1000}[-1\ 1]\begin{pmatrix} 0 \\ 0.019 \end{pmatrix}$

$\sigma = 43.7N/mm^2$

Uniformly Distributed Load at Fixed Support of UAV Rotor Arm

Element Matrix

$[K_e] = \frac{AE}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

$[K] = [K_1] = 10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}$

Global Nodal Displacement Vector

$(Q) = \binom{q_1}{q_2}$

Load Vector

$(F) = \binom{\frac{P_0}{2}}{} = \binom{1}{1}$

$(F) = \binom{50}{50}10^4$

Equilibrium Condition [10]

[K] (Q) = (F)

$10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}\binom{q_1}{q_2} = \binom{50}{50}10^4$

Therefore, the nodal vector displacement is,

$(Q) = \binom{0}{9.67}$

Stress in Each Element

$\sigma = \frac{E_e}{l_e}[-1\ 1]\binom{q_i}{q_{i+1}}$

Where i = e and e are an element number

$\sigma = \frac{230\times10^3}{1000}[-1\ 1]\binom{0}{9.67}$

$\sigma = 2.22 \times 10\text{^}3\ N/mm^2$

V RESULTS AND DISCUSSION

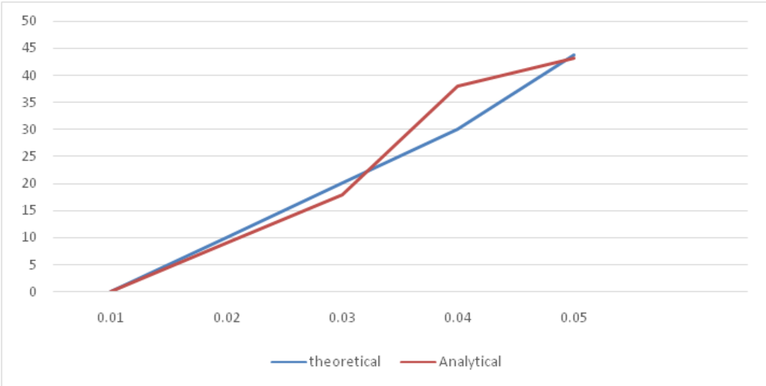
Calculated Results

Parameter	Point load	Uniformly Distributed Load
Elemental stiffness[K]	$10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}$	$10^4\begin{bmatrix} 5.17 & -5.17 \\ -5.17 & 5.17 \end{bmatrix}$
Global Nodal Displacement[Q]	$\binom{0}{0.019}$	$\binom{0}{9.67}$
Global Load Vector [F]	$\binom{0}{1000}$	$\binom{50}{50}10^4$
Stress (σ)	43.7N/mm^2	2.22× 10^3N/mm^2

Comparison between Theoretical and Analytical Results

Parameter	Theoretical Results	Analytical Results
Equivalent Stress	43.7N/mm^2	43.375N/mm^2
Total Deformation	0.022mm	0.05mm

Graph



Stress v/s Deformation

VI CONCLUSION

Finite element analysis on the arm of multirotor UAV has been carried out and the factors such as deformation and stress acting on the arm is calculated experimentally and theoretically and the results are for point load and uniformly distributed load.

VII REFERENCES

[1] Klippstein H., Hassanin H., Sanchez A. D., Zweiri Y., and Seneviratne L. (2018), “Additive Manufacturing of Porous Structures for Unmanned Aerial Vehicles Applications”, Advance Engineering Materials, Volume. 9, Issue 10.

[2] Balasubramanian E., Sagar N., Chandrasekhar U., and Salunkhe Y. (2019), “Development of Light Weight Multi-Rotor UAV Structures through Synergistic Application of Design Analysis and Fused Deposition Modelling”, International Journal of Materials and Product Technology, Volume. 9, Issue. 3, pp. 229-238.

[3] Ding, Y., Zhou, Z., Wang, Z., Liu, H., and Wang, K., (2019), “Bionic Stiffener Layout Optimization with a Flexible Plate in Solar-Powered UAV Surface Structure Design”, Applied Sciences, Volume. 9, Issue. 23.

[4] Chen R., Zhu X., Zhou Z., Wang Z., and Zhang, T. (2018), “Study of Structure Design of a Solar Powered UAV”, IEEE International Conference on Prognostics and Health Management, Washington, USA.

[5] Rumayshah K. K., Prayoga A., and Moelyadi I. M. A. (2018), “Design of High-Altitude Long Endurance UAV: Structural Analysis of Composite Wing using Finite Element Method”, IOP Conf. Series: Journal of Physics, Volume. 1005, Issue. 1.

[6] Santoso D. W., and Putra A. N. (2018), “Design and Analysing the Strength of the Structure Fixed Wing Vertical Take Off-Landing UAV”, Proceedings of the National Seminar on Information Technology and Aerospace, Volume. IV.

[7] Kim H., Kim S. J., Kim S. and Kim T. U. (2018), "Structural Analysis of Fasteners in the Aircraft Structure of the High-Altitude Long Endurance UAV", Journal of Aerospace System Engineering, Volume. 12, Issue 1, pp. 35-41.

[8] Zahran M., and Abdelwahab M. (2019), "Crash Analysis of UAV Hybrid Composite Fuselage Structure under Different Impact Conditions", Materials Science Forum, Volume. 953, pp- 88-94.

[9] Park S. W., Shin J. W., and Kim T. U., (2018), "Development of the Main Wing Structure of a High-Altitude Long Endurance UAV", International Journal of Aeronautical & Space Sciences, Volume. 19, pp- 59-71.

[10] Vasic, Z., Maksimovic, S., and Georgijević, D. (2017), "Applied Integrated Design in Composite UAV Development", Applied Composite Materials, Volume. 25, pp. 221-236.

[11] Madeva Nagaral, Auradi, V and Ravishankar, MK, "Mechanical Behaviour of Aluminium 6061 Alloy Reinforced with Al2O3 & Graphite Particulate Hybrid Metal Matrix Composites", International Journal of Research in Engineering & Technology (IJRET), Volume. 1, Issue. 2, pp. 193-198

[12] Mishra, A.A.D.Y.A., Pal, S.O.U.R.A.V., Malhi, G., and Singh, P.R.A.B.H.A.T. (2020), "Structural Analysis of UAV Airframe by using FEM Techniques: A Review", International Journal of Mechanical and Production, ISSN: 2249-6890.

[13] Rajesh Prabha, N., Edwin Raja Dhas, J and Ramanan, G (2017), "Finite Element Structural Analysis of Connecting Rod of AA7075-Tic Composite using ANSYS", International Journal of Mechanical Engineering and Technology, Volume. 8, Issue. 7.

[14] Salleh, M. M. B.H., Salleh, Z., Rosdi, M. S.and Sapuan, S. M, "Mechanical Properties of Coconut Carbon Fibre/Epoxy Composite Material", International Journal of Mechanical Engineering (IJME), Volume. 2, Issue. 3, pp. 55-62.