Structural Analysis of a Drone Frame

Submitted in fulfilment of requirements of FEA course

Project by:

SHUBHAM MEHTA – 60005190107
TEJAS SRINIVAS – 60005190118
TATSAT BALDANIYA – 60005190117
TANMAY PAGARIYA – 60005200116

Faculty:
DR. VINAYAK KHATAWATE

Dwarkadas Jivanlal Sanghvi College of Engineering
Mechanical Engineering Department

Structural Analysis of Drone Frame

Vinayak H Khatawate[1], Shubham Mehta[2], Tatsat Baldaniya[3], Tejas Srinivas[4], Tanmay Pagariya[5],

[1,2,3,4,5] *SVKM's DJSCE, Vile Parle, Mumbai -400056*

ABSTRACT

In the present study, a structural analysis was implemented on a Drone Frame Arm. The software used for analysis was ANSYS. It was used to study the maximum displacement, stress and strain on the drone frame component. For the analysis the material was assumed to be isotropic throughout. A Finite Element Analysis (FEA) was employed to model the drone frame reaction to various loads. The frame was subjected to a single force, namely thrust. The model was also analyzed by considering the bending aspect of the thrust forces. The frame material used for the study is CF-ABS which has high tensile and bending strength with a higher strength to weight ratio.

Index Terms

Drone frame, carbon fibre, ABS, von mises stresses, Finite Element Analysis, Structural Analysis

I. INTRODUCTION

Drones are now being implemented in various fields from agriculture to defense. Every year the role of drones has been increasing. The areas where drones are being used are expanding due to ever-advancing electronics. Drones are also being used for surveillance and reconnaissance by defense forces. Drones are now being used to transport medical equipment to their required destination or to drop relief packages in disaster-hit or prone areas. Critical structures can be monitored such as industrial buildings or sewers to find exactly where damages or any repair works are required. In structural firefighting applications, drones can extinguish fires at places that are hazardous to humans, reaching places more effectively and quickly thus, the applications of drones are becoming very important in human life. Drones come in a lot of shapes and sizes such as X frame Quadcopters, H frame Quadcopters, and many more. We have adopted a simple drone frame for the analysis. (*Fig 1).*

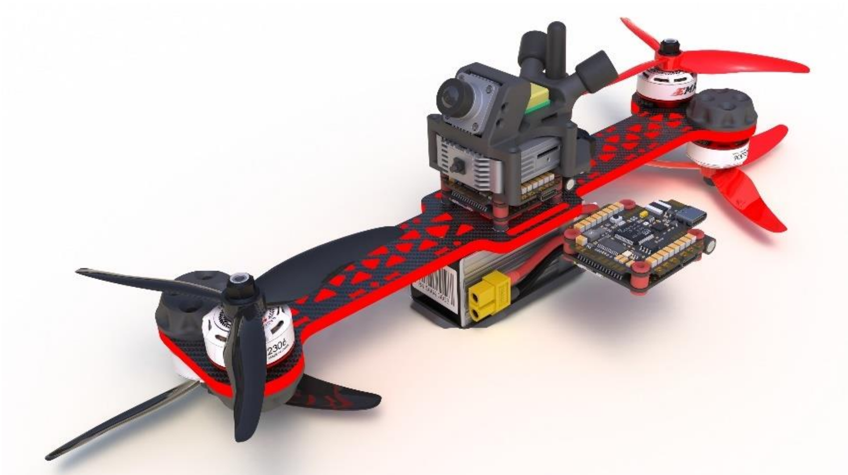

Image 1: Typical Drone Frame

II. LOADING CONDITIONS

Here, consider the drone body in static condition so the not considering yaw, roll and pitch movement on drone body, but the lift force and weight of drone cause the deformation on drone body. Need to estimate required thrust force and weight of drone.

$$Thrust = Lift$$
$$Thrust/Weight = 2.4$$

From above equation evaluate a total required thrust force of 48 N for drone and from the required thrust find required per motor thrust 12 N that is helps to choose rotor and propeller combo for drone.

III. DESIGN AND 3D MODELLING

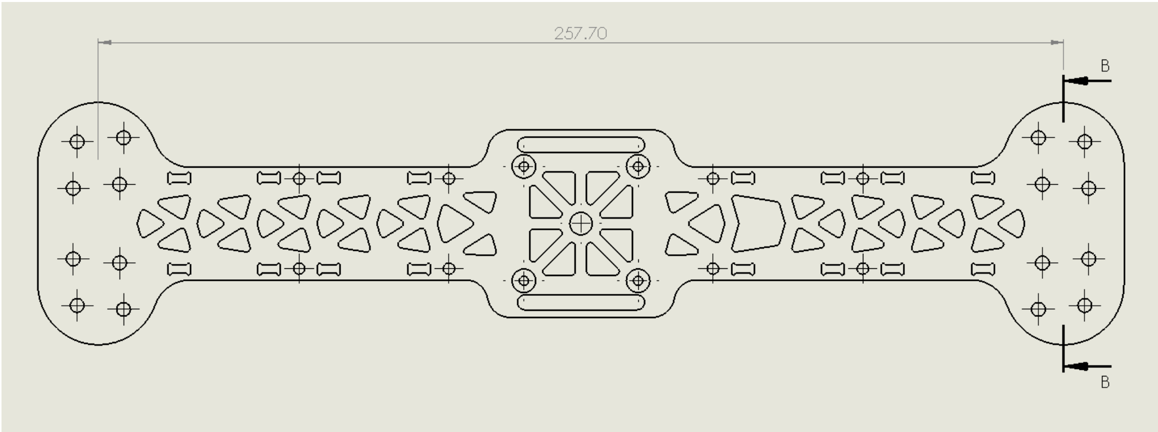

Image 2: Drone Frame Design

The drone body design was made by allotting the required space for fitting of electrical component such as flight controller, battery, Electronic Speed Controller and sensor. For this model, body frame size depends on rotor and propeller size that produces sufficient thrust for take-off of drone in free air. The propeller was chosen with 76.2 mm diameter that decides arm length that is 257.7 mm. The body frame is symmetric, and all electronic components placed at center of drone body frame for better weight distribution. Since, the arm thickness is known we used the analytical approach to calculate the bending stress according to the following formula,

$$[\sigma_b] = \frac{M \times y}{I}$$

(∵ taking value of y from the design)

Where, $[\sigma_b]$= Design bending stress

$M = Moment\ of\ Arm$

$y = distance\ from\ neutral\ axis$

$I = Moment\ of\ Inertia$

While assuming the cross section of the drone frame is rectangular throughout, treating the body as a cantilever beam as shown in the figure.

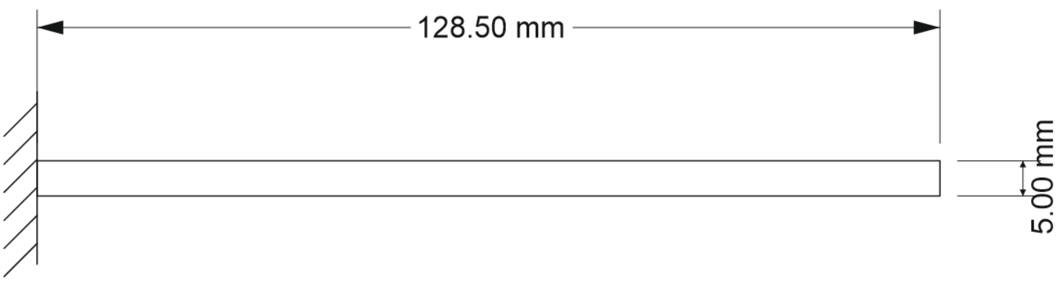

Image 3: Treating the arm as cantilever beam

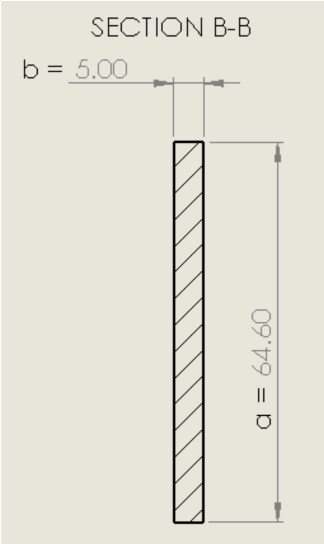

Image 4: Cross-section of beam/arm

$$I = \frac{a \times b^3}{12} = \frac{64.6 \times 5^3}{12} = 666.66\ mm^4 = 666.66 \times 10^{-12}\ m^4$$

$$y = \frac{b}{2} = \frac{5}{2} = 2.5\ mm = 0.0025\ m$$

$$M = \frac{Thrust \times Arm\ length}{2} = \frac{24 \times 0.2577}{2} = 3.084\ Nm$$

(Arm length = 257.7 mm = 0.2577 m)

$$[\sigma_b] = \frac{M \times y}{I} = \frac{3.084 \times 0.0025}{666.66 \times 10^{-12}} = 1.156 \times 10^7\ Pa$$

And for design calculations considering Factor of Safety (FOS) = 3

$\sigma_b = 3 \times [\sigma_b] = 3 \times 1.156 \times 10^7 = 3.468 \times 10^7\ Pa$ …Eq. 1

Where, σ_b = Actual bending stress

Now, using the ANSYS approach,

From ANSYS, Actual normal stress (σ_n) = 2.256 x 10^7 Pa

We know that,
$\sigma_b = 1.6 \times \sigma_n = 1.6 \times 2.256 \times 10^7 = 3.6096 \times 10^7\ Pa$ …Eq. 2

Since, Eq. 1 and Eq. 2 are close to each other in terms of values, this gives us the indication that the design is safe.

IV. STATIC ANALYSIS

The drone is a single rigid body as shown in *Image 2*. The maximum loads on the drone body cause the deformation of drone arms. Static structural analysis was performed for making the body as rigid as possible and with optimized weight value. It is capable of withstanding loads which are applied by the thrust, weight of drone and payload. The load is applied on individual arms of drone body of 12 N and weight of drone including payload applied on base plate of 50 N. Analysis carried out with certain boundary conditions applied on drone body that is shown below,

The CF-ABS material choose for the analysis and material properties are given in Table 2 Fine and dense meshing applied on it for find appropriate results of this analysis. The meshing properties for this analysis given in Table 3 and meshing applied on drone body shown in Image 5.

Table 1: Material Properties

Properties	Values
Density (kg/m^3)	1110
Young's Modulus (MPa)	1940
Tensile Strength (MPa)	32.2
Poisson's Ratio	0.3
Shear Modulus (MPa)	857

Table 2: Meshing Properties

Property	Value
Mesh type	Solid Mesh
Jacobian points	16 Points
Element Size	2e-003 m
Tolerance	0.0001
Total nodes	208684
Total elements	126202

The outcomes of this analysis are in maximum stress, displacement, and strain that value is 22.5 MPa, 0.1024 µmm and 0.000118 respectively. The following images show the Maximum stress, Displacement, and Strain respectively. After the simulation of drone body, it is found that mostly the stress was generated at y section area of arm. Here, von mises stresses were generated all-over the drone frame. The maximum stress value is below the ultimate strength of material of 22.5 MPa. This stress value gave FOS (factor of safety) 3 that was fulfilled by our desired limit of Factor of Safety which was between the range 2.5 to 3.

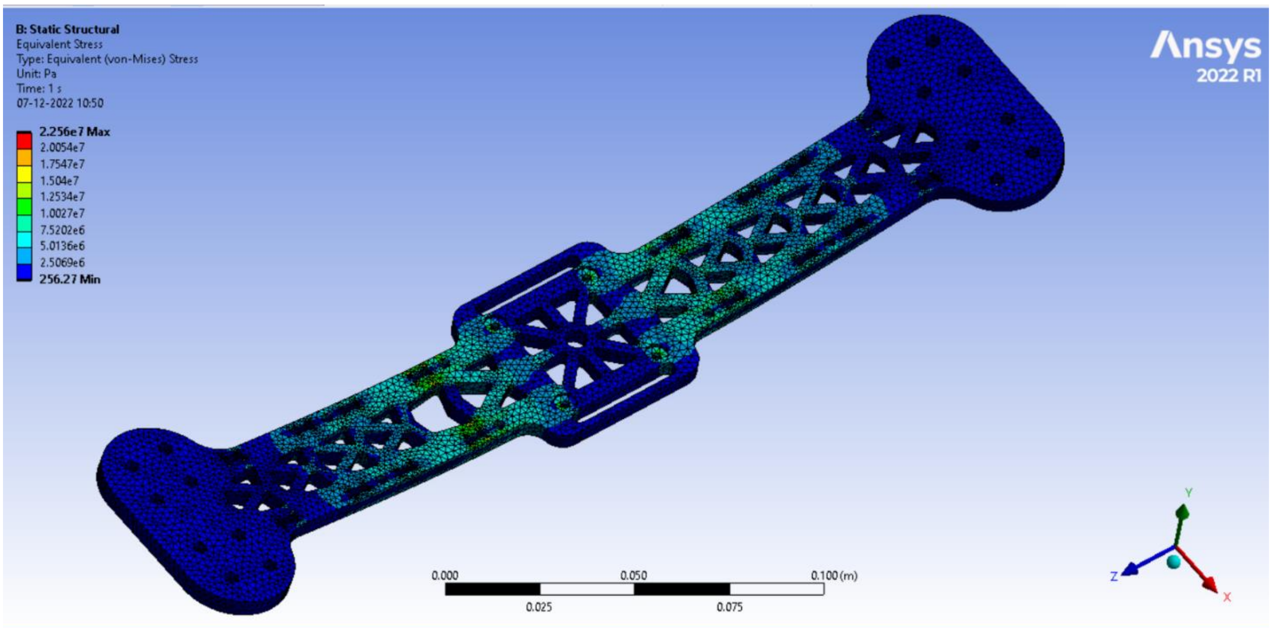

Image 5: Static analysis stress (Isometric view)

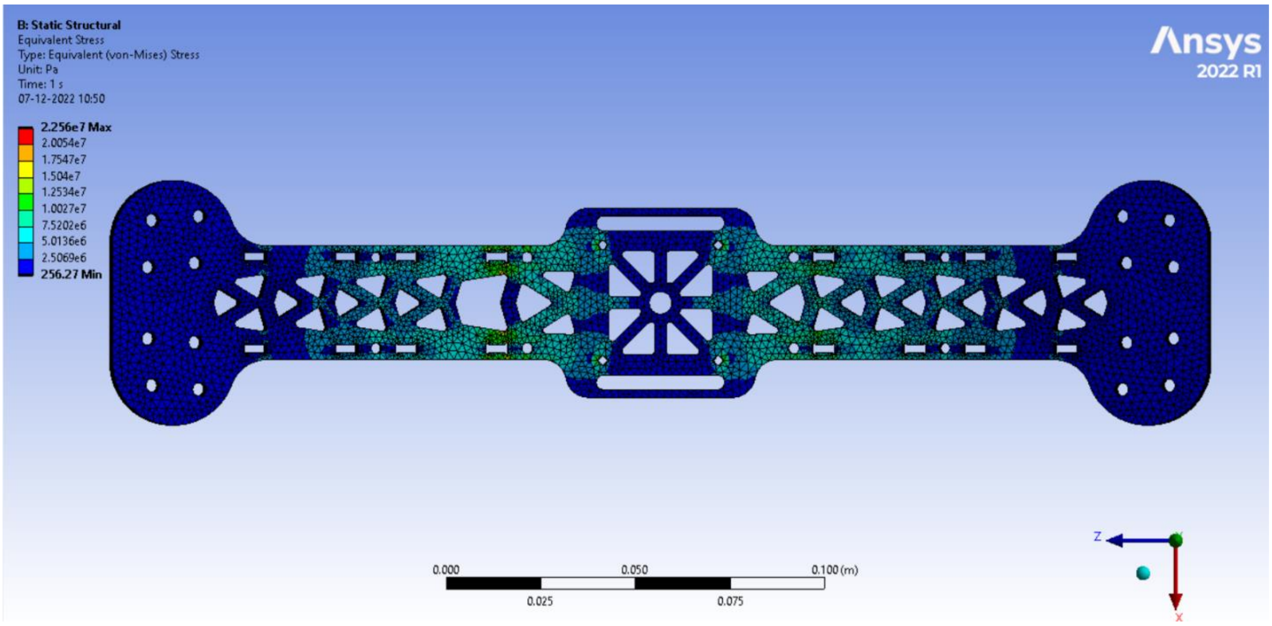

Image 6: Static stress analysis (Top view)

	Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1	1.	256.27	2.256e+007	1.9577e+006

Image 7: Static stress analysis table

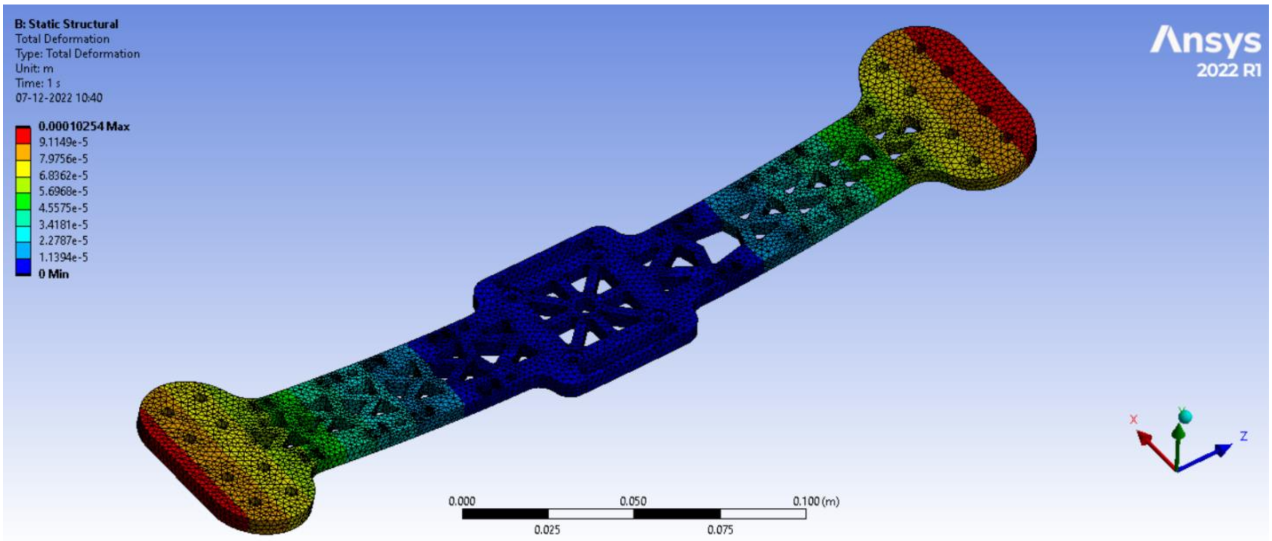

Image 8: Static analysis displacement (Isometric view)

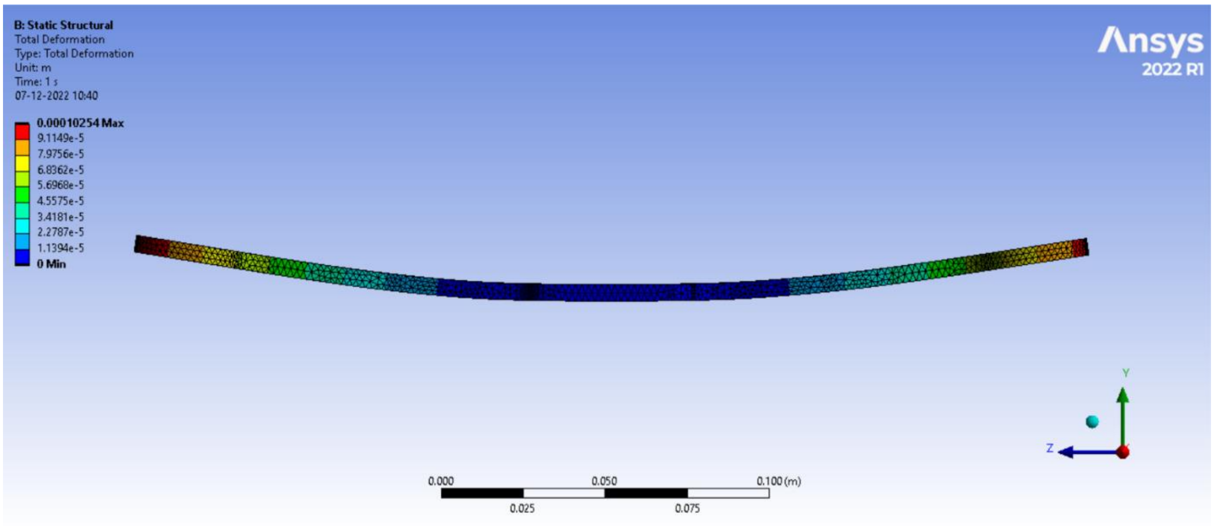

Image 9: Static analysis displacement (Side view)

	Time [s]	Minimum [m]	Maximum [m]	Average [m]
1	1.	0.	1.0254e-004	3.563e-005

Image 10: Static analysis displacement Table

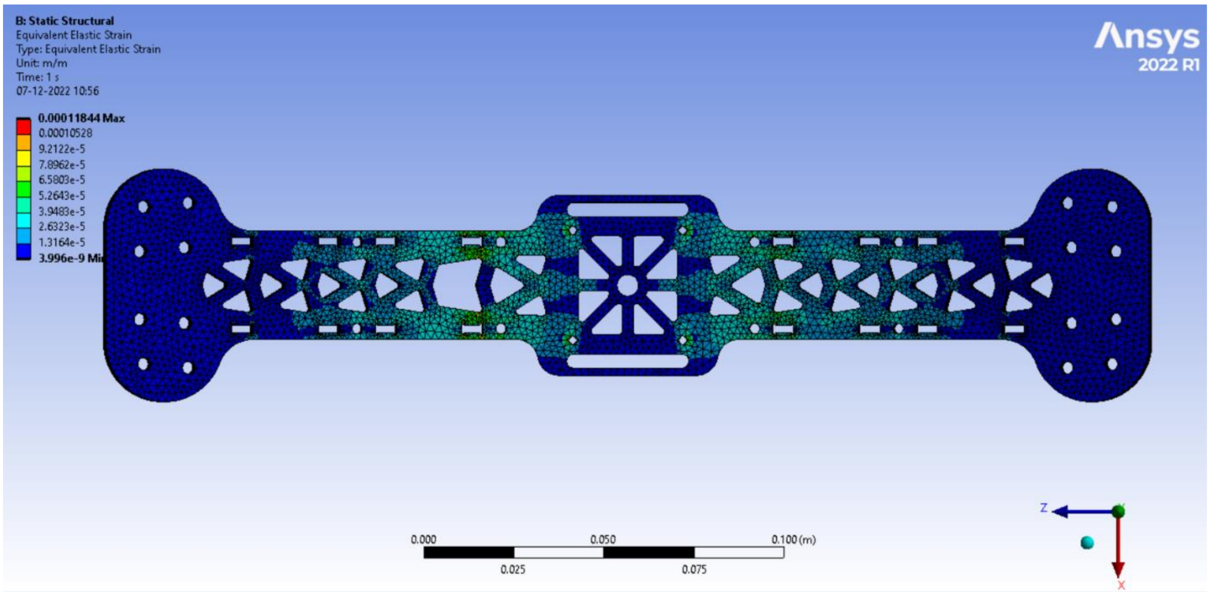

Image 11: Static strain analysis

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
1.	3.996e-009	1.1844e-004	1.0583e-005

Image 12: Static strain analysis table

.

V. MODAL ANALYSIS

The Modal analysis was also done on the design model. The results obtained are presented below.

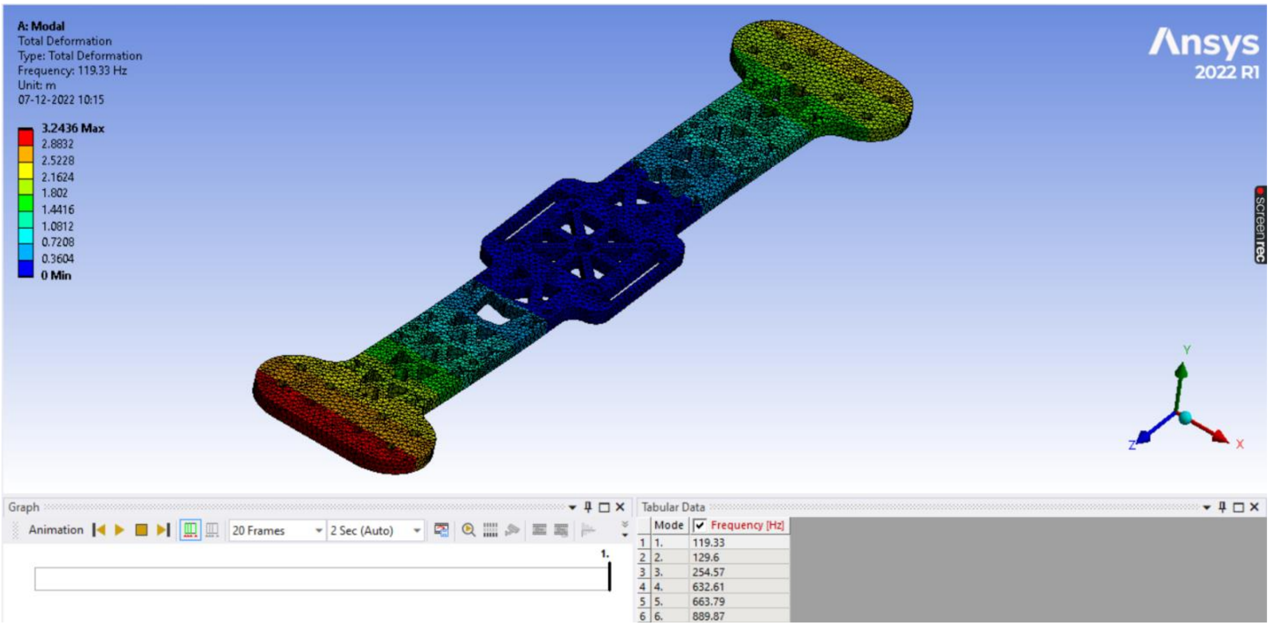

Image 13: Modal Analysis (i)

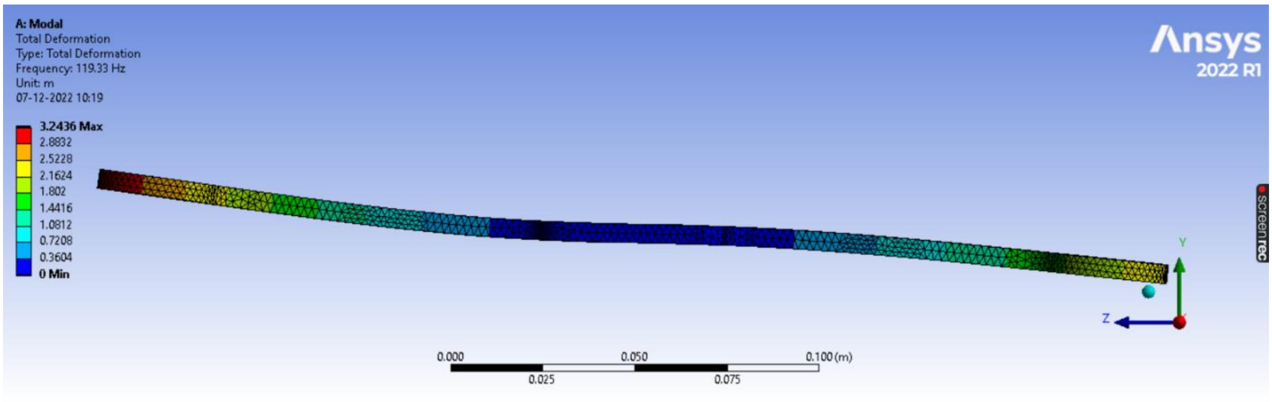

Image 14: Modal analysis (ii)

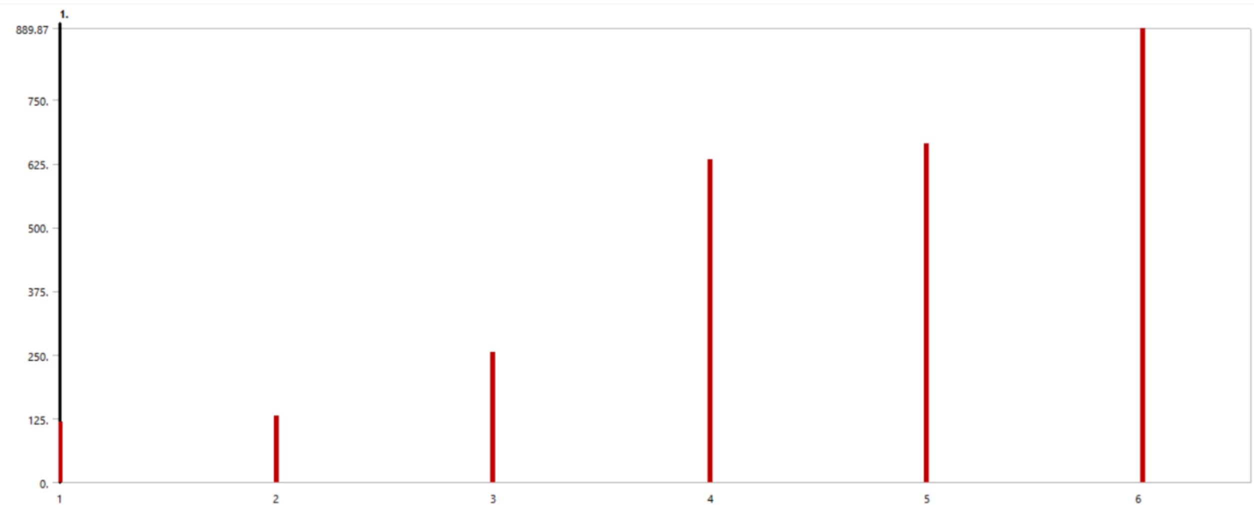

Image 15: Modal analysis Graph

Tabular Data

	Mode	Frequency [Hz]
1	1.	119.33
2	2.	129.6
3	3.	254.57
4	4.	632.61
5	5.	663.79
6	6.	889.87

Image 16: Modal analysis tabular data

VI. CONCLUSION

It was concluded that the drone frame arm design was feasible as the theoretical stress values (refer to point III) were consistent with the values obtained from the FEA. Also, the maximum stress was found to be less than the ultimate strength of the material by FEA analysis. Similar results were obtained for the maximum strain value as well.

VII. REFERENCES

[1] Brijesh Patel, R. P. Sukhija, J. V. Sai Prassana Kumar, "Structural Analysis of Arm of Multicopter with Various Loads", International Journal of Emerging Technology and Advanced Engineering, Volume 8, Issue 4, 2018.

[2] Yangguang.2CZ. Mechanical analysis of 21 type sugarcane machine frame and optimization [J]. Agricultural Machinery, 2007, 38(1).

[3] Pan Wei, ZiJian Yang , Quanzi Wang, "The Design of Quadcopter Frame Based On Finite Element Analysis", 3rd International Conference on Mechatronics, Robotics and Automation (ICMRA 2015), DOI: https://dx.doi.org/10.2991/icmra-15.2015.260