FEM Wing Analysis

Kiril Yampolsky
San Diego State University, San Diego, CA, 92182, USA

This report analyzed the Finite Element Analysis of a wing box. Using ABAQUS finite element software, a static, buckling and vibration analysis was done. The wing geometry as well as the force of lift was given in order to complete the analysis. The results demonstrated that the wing, even under its maximum stress nodes, would have a stress less than the yield stress of aluminum, meaning it could withstand the pressures of lift and gravity. On top of that, the static analysis was able to display the tip displacement of the wing under the loads to get a better understanding of how the wing would act. The buckling analysis was able to demonstrate how the wing would act under different buckling modes. The static analysis showed the wings behavior given the vibration.

I. Introduction

The motive of this document is to perform a static, buckling, and vibration analysis and be able to understand the results to confirm they make sense. By creating the wing geometry and placing it under the certain conditions, these results could be found. A static analysis is able to give insight into the stress and displacement the wing will be under when certain loads and boundary conditions are applied. This will give a better understanding of the maximum stresses the wing will encounter when in flight and comparing the stress values to the yield stress demonstrates if the wing material will yield due to the applied stresses. The buckling modal analysis is done in order to estimate the locations of critical buckling loads, and try to get solutions on how to prevent them from occurring. A frequency vibration analysis is done to get a better understanding of how the wing will react to external forces to better understand the behavior and structural integrity under the stresses. It is very useful for failure prediction and understanding where the weak points of the structure are.

.

II. Theory

The first step to being able to perform the static, buckling, and vibration analysis is building the geometry of the wing box. This was done using the finite element analysis (FEA) software ABAQUS. The way to create the figure is by creating a datum planes along the span of the wing box and then revolving it. This same process can then be used to create the ribs and middle spar. The geometry is shown in figure 1 below and the thickness values can be shown in table 1 below. The wing skin, spar webs, and ribs utilize shell elements. The spar caps are made as wires and can be modeled as truss elements. In the meshing process, quadrilateral S4 elements were used to mesh the wing box.

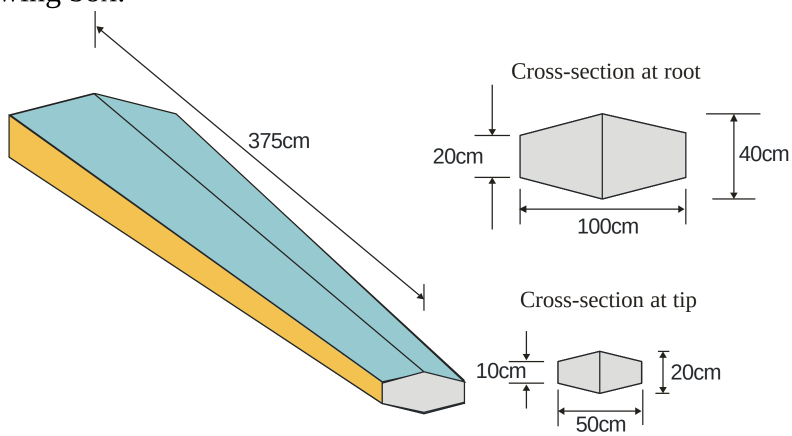

Figure 1: Tapered Unswept Hexagonal Wing box Geometry

Component	Values
Top/Bottom skin at root, t_{root}	4 mm
Top/Bottom skin at center, t_{center}	2 mm
Top/Bottom skin at tip, t_{tip}	1 mm
Spar web thickness, t_{web}	3 mm
Rib thickness, t_{rib}	2 mm
Spar cap (area), A_{spar}	500 mm^2

Table 1: Wing Thickness values

After that, applying the thickness values from table 1 towards the aluminum properties shown below, and assembling it all together, the geometry is complete.

Density($\frac{kg}{m^3}$)	Youngs Modulus(GPa)	Poisson Ratio	Yield Stress(MPa)
2810	71.7	0.33	469.56

Table 2: Aluminum 7075 T-6 Property Values

In order to calculate the gravitational force applied to the wing, multiplying the density of aluminum by the wing box volume using equation 1 below, the mass can be found. Assuming the wing is under earth's gravitational constant, the gravitational acceleration of 9.81 can be used. Equation 2, below shows how to calculate the constant force of gravity applied on the wing. The full calculations can be found in the appendix.

$$m = \rho V \tag{1}$$

$$F = ma = mg \tag{2}$$

The lift force is given and dividing the lift force at the certain areas, the lift pressure can be calculated. The pressure calculations are done in the appendix.

$$P = \frac{F}{A} \tag{3}$$

III. Results and Discussion

Part A: Static Analysis

	Converge 1:	Converge 2:	Converge 3	Converge 4
Number of Elements	852	3408	5400	7548
S,mises max	$1.109 * 10^8$	$1.605 * 10^8$	$1.832 * 10^8$	$1.901 * 10^8$
u deformation	$2.338 * 10^{-2}$	$2.494 * 10^{-2}$	$2.536 * 10^{-2}$	$2.538 * 10^{-2}$
% Error		6%	1.7%	<1%

Table 3: Convergence Mesh Analysis

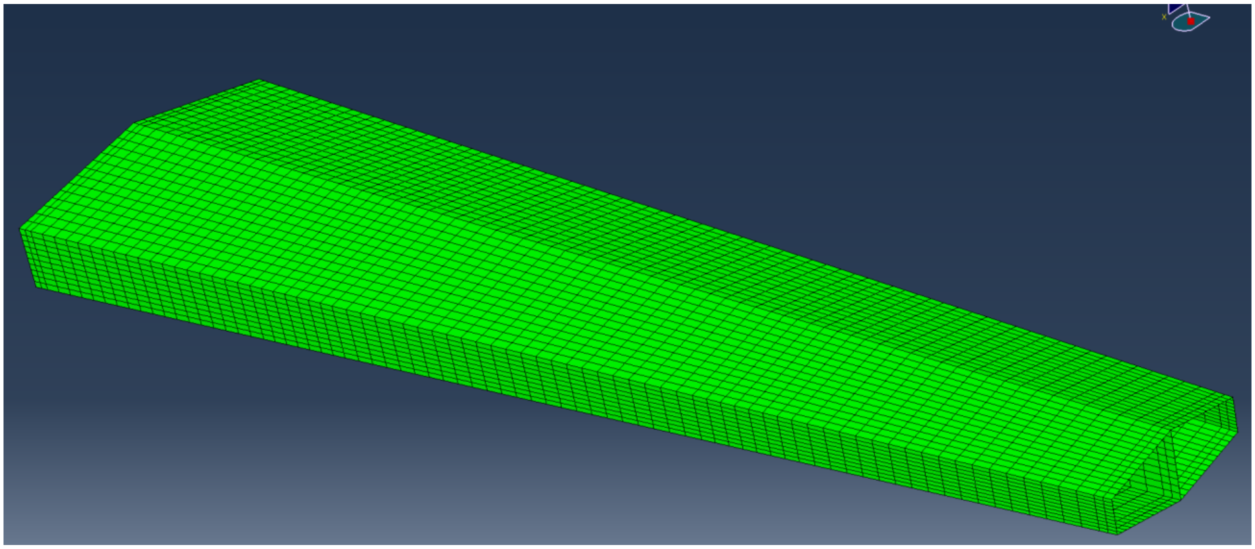

Figure 2: Fully Converged Mesh bottom skin u magnitude displacement

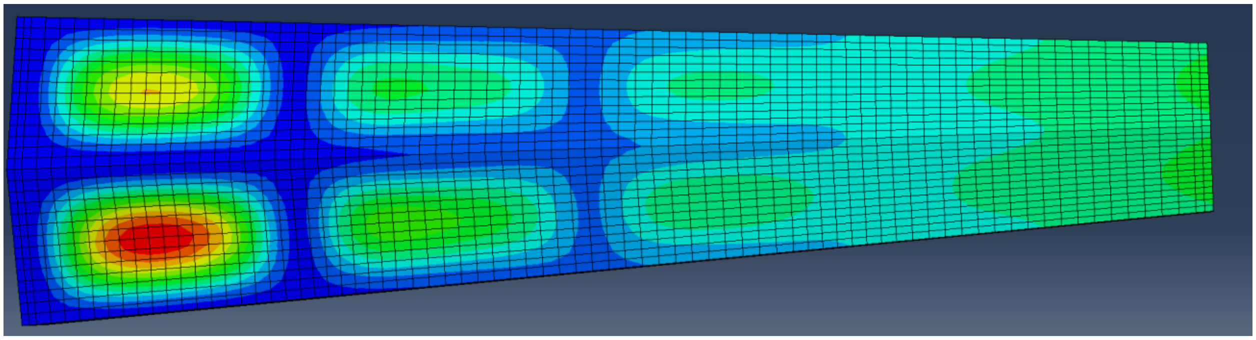

Figure 3: Fully Converged Mesh bottom skin u magnitude displacement

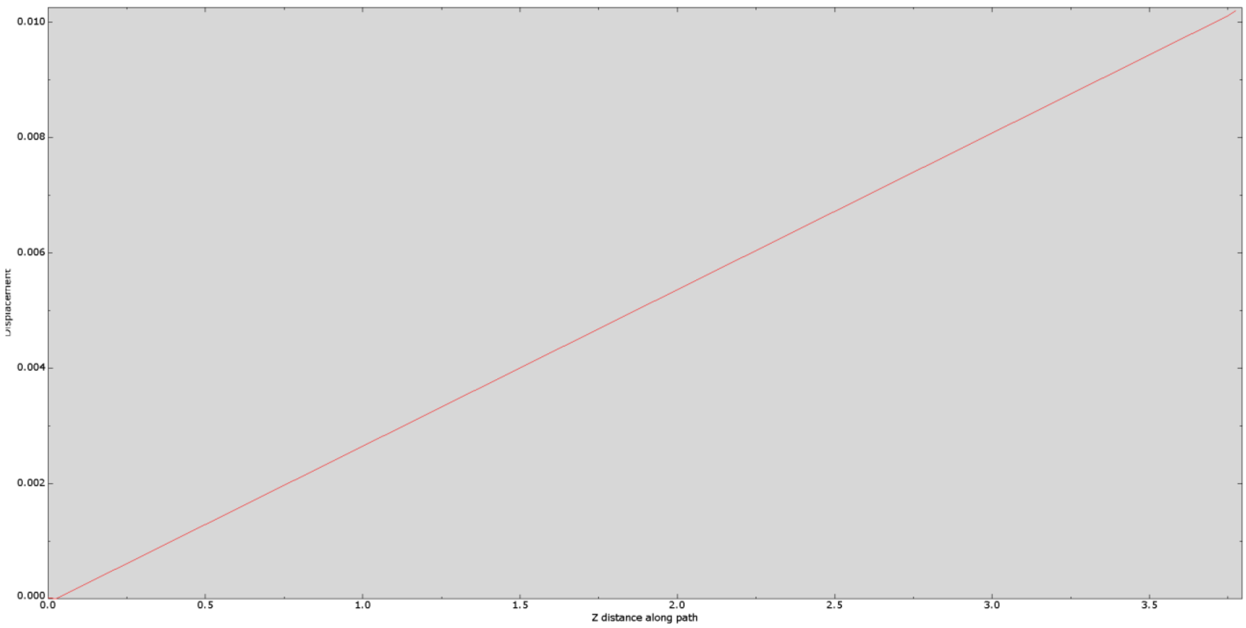

Figure 4: Bottom Skin v Span Length Plot After Deformation

	Top Skin	Bottom Skin	Spar Webs
Max Von Mises Stress	$3.646 * 10^7$	$1.901 * 10^8$	$8.852 * 10^7$

Table 4: Maximum Von Misses Stress

Max Principal Stress on Bottom Skin
$1.336 * 10^8$

Table 5: Maximum Principle Stress on Wing

After creating the figure and getting a baseline mesh, a convergence analysis can be performed by refining the mesh until the values start to converge. The purpose for refining and performing a convergence analysis is to be more confident in results and getting more accurate nodal data along the figure. Taking a look at table 3, the displacement values percent error can be used to show convergence. The percent error got to being below 1% which is the threshold for convergence. Figure 2 represents the fully converged mesh and pre deformation of the figure. After that, taking a look at figure 3, the bottom skin contour is shown to demonstrate the sections where the maximum u deformation and maximum stress is on the wing. This data lines up with the pressure forces applied on the wing. The maximum pressure is applied on the middle skin closest to the root of the wing, and given the wing is very thin, the high pressure values would affect the wing in those areas the most. While the tip of the wing is a lot thinner than the root, it makes sense that it experiences less stress as the pressure applied to the tip is a lot lower than to the root. Taking a look at table 4, the maximum von misses stress is at its peak on the bottom skin, as that is where the lift force is applied. The top skin and spar webs experience less stress than bottom skin as they do not have the lift pressure applied to them. The maximum principal stress on the bottom skin is found in table 5. This is the point where the maximum normal stress occurs and is usually the point where the material is most likely to yield. Comparing the maximum principal stress to the yield stress of aluminum shown in table 2, the bottom skin will not yield. Figure 4 demonstrates the u deformation of the wing along the span of the wing. This plot should be linear, and while the root has 0 deformation, the wing tip has the highest u deformation.

Part B: Buckling Analysis

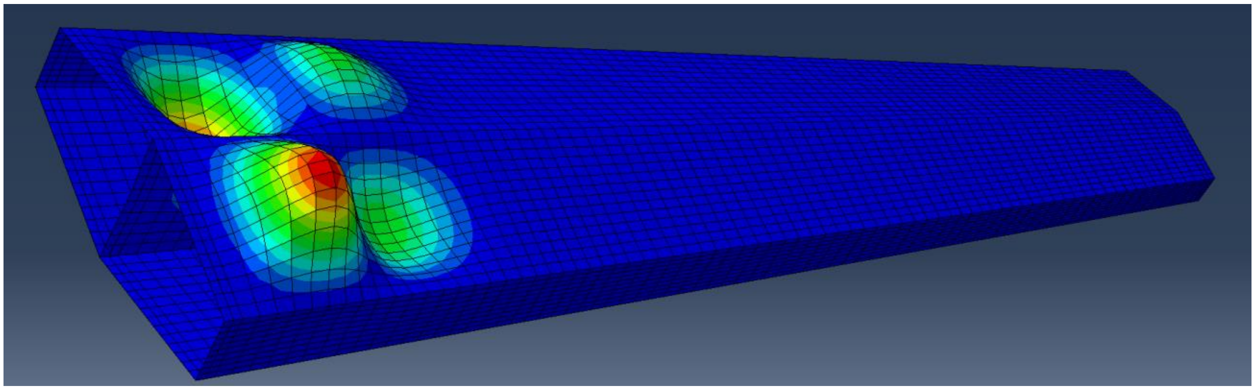

Figure 5: Mode 1 Buckling Analysis

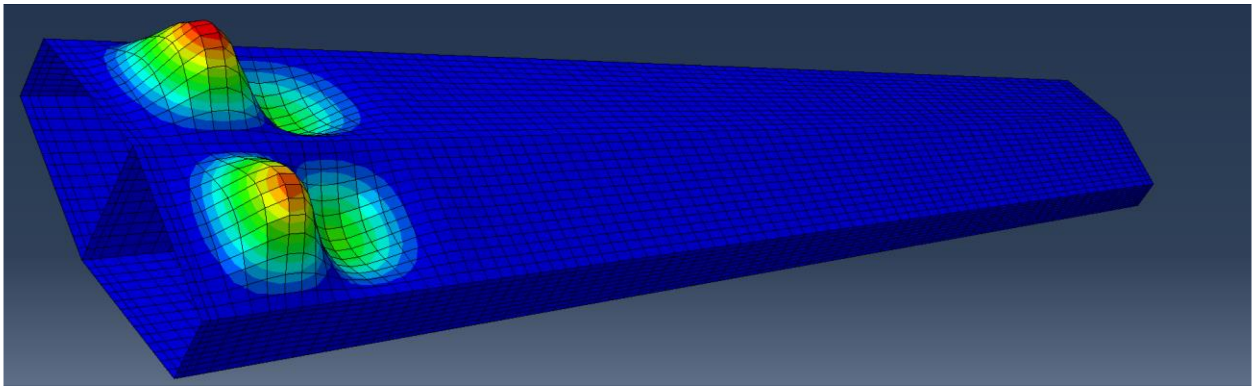

Figure 6: Mode 2 Buckling Analysis

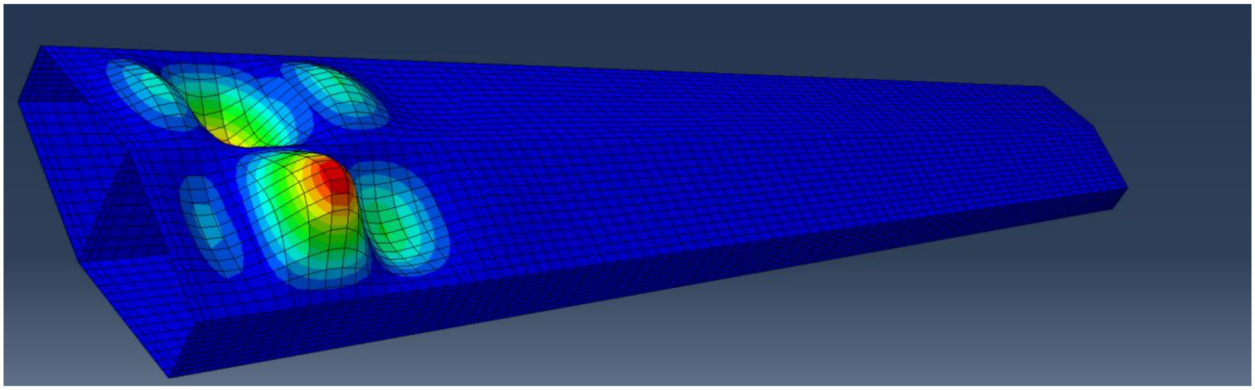

Figure 7: Mode 3 Buckling Analysis

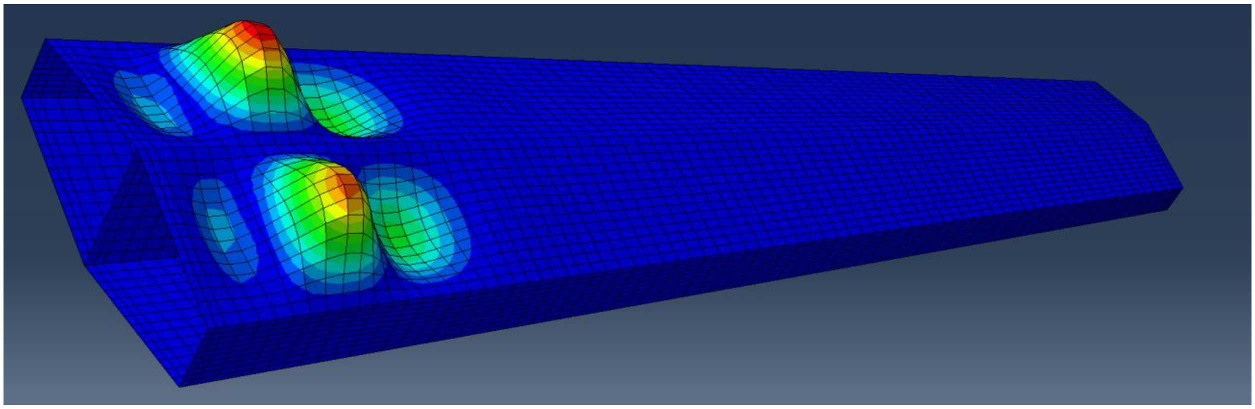

Figure 8: Mode 4 Buckling Analysis

	Mode 1	Mode 2	Mode 3	Mode 4
Load Eigenvalue	0.356	0.394	0.401	0.436

Table 6: Buckling Load Values

Figures 5-8 go over the buckling analysis of the wing box. The buckling analysis is used to investigate the stability of a structure under applied loads. The load eigenvalues indicate the magnitude of loads that would cause the wing box to buckle. The load eigenvalues found in table 6 are very small which means that the wing would not be able to support much additional weight before it buckles. The mode shapes shown in the contour plots of figures 5-8 describe the shape the structure would deform under the loads applied. As seen, the local buckling occurs between the first rib and the top skin wing root. What this entails is that to better support more weight, the local thickness would need to be increased at this section to be able to counteract the load force applied. The buckling analysis is extremely important as it gives the sections of the wing box that are most vulnerable to buckling and under what loads that would occur. The different mode shapes also give insight into how the shape of the wing will react when the applied force is shown.

Part C: Nodal Analysis

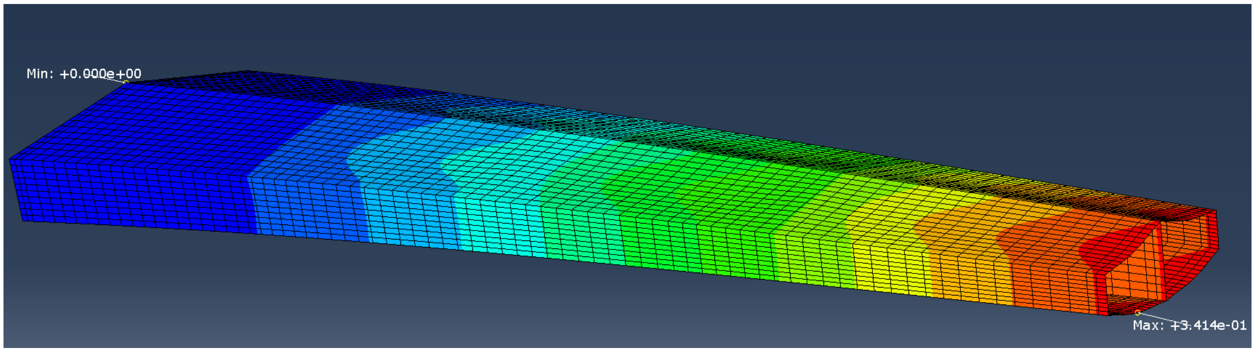

Figure 9: Mode 1 Vibration Analysis Frequency 28.26

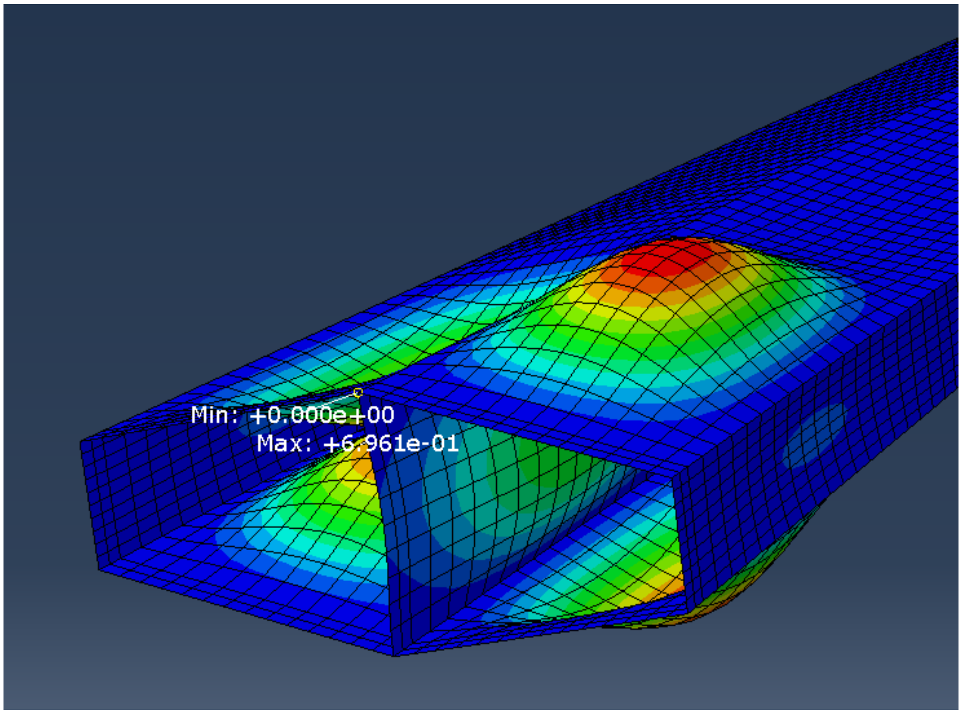

Figure 10: Mode 2 Vibration Analysis Frequency 42.14

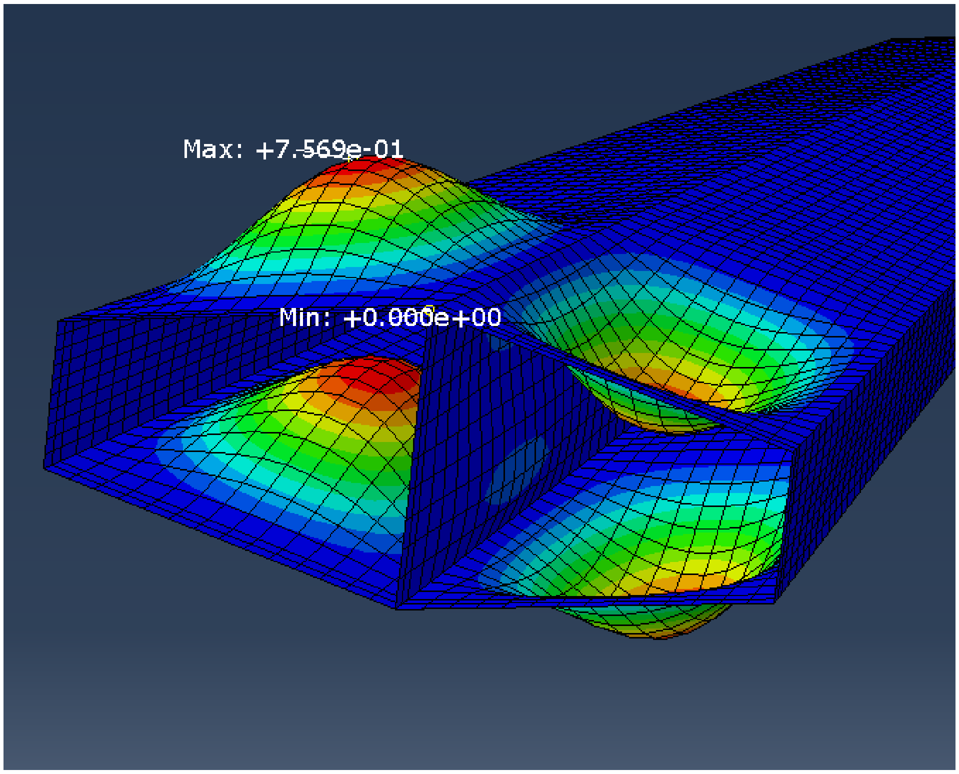

Figure 11: Mode 3 Vibration Analysis Frequency 48.21

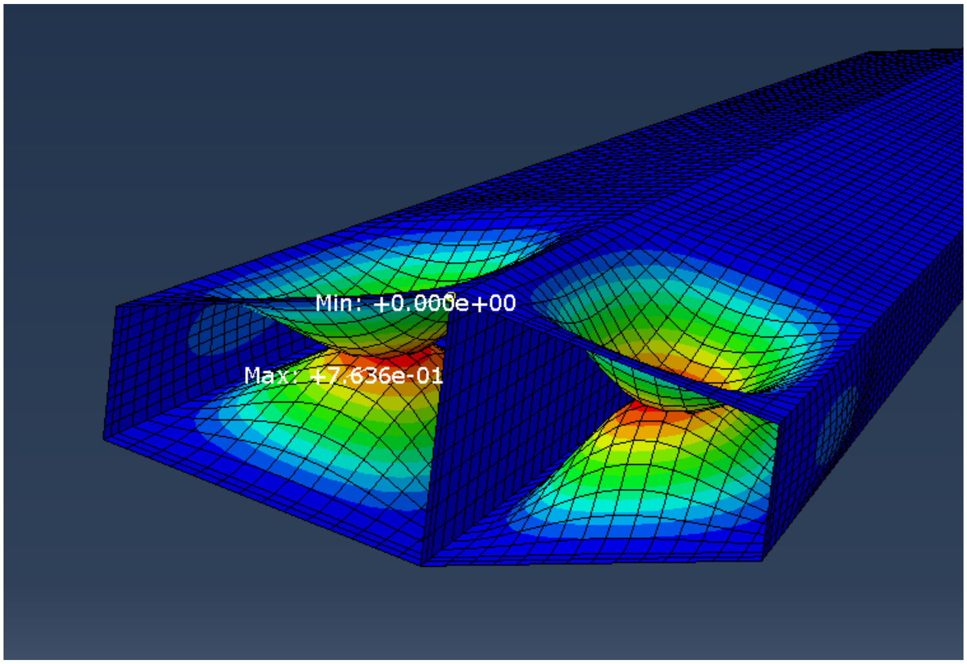

Figure 12: Mode 4 Vibration Analysis Frequency 50.44

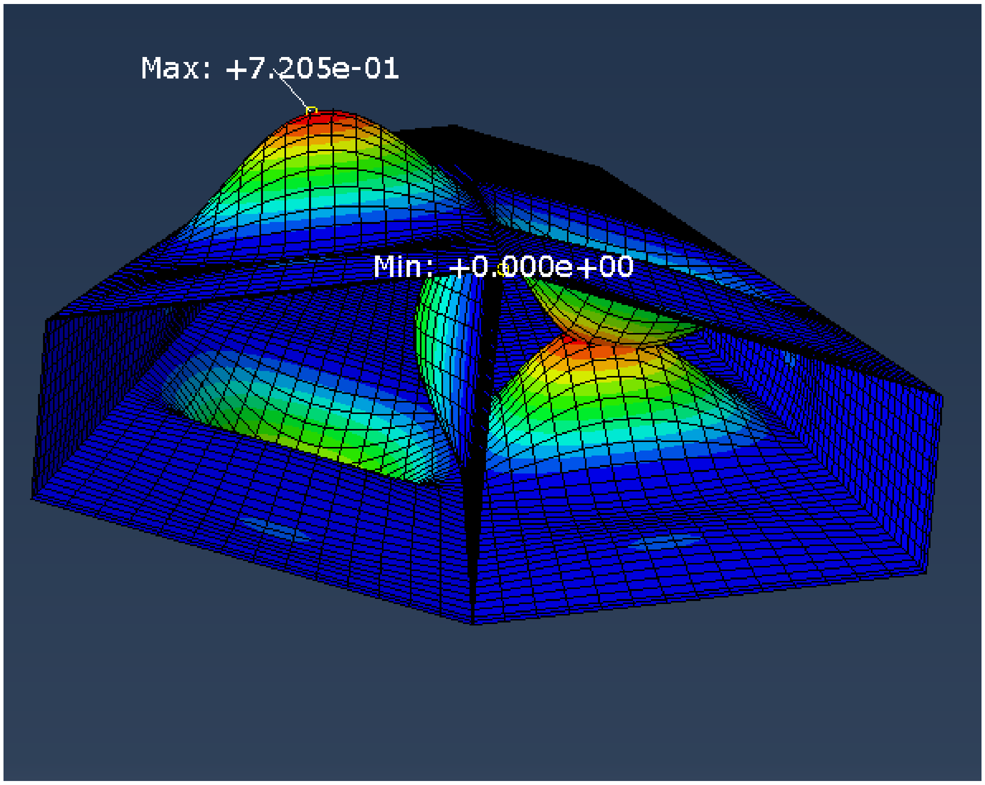

Figure 13: Mode 5 Vibration Analysis Frequency 52.03

	Bending Mode(1)	Torsion Mode(4)
Frequency	28.2	50.44

Table 7: Pure Bending and Torsion Mode Frequencies

The frequency analysis is based on 0 load condition applied to the wing. The frequency analysis gives the natural oscillation of the wing given different vibration values. Taking a look at figure 9, the first mode was pure vertical bending of the structure. The wing tip experiences the most deformation at this point. Figures 10-13 only show the local reactions near the root of the wing. This is due to the wing skin thickness not being able to withstand the magnitude of the vibration and causes the structure to fail before it can deform. Mode 2 demonstrates pure bending again, but this time horizontally. Figure 10 shows the top skin of the wing failing near the root and not being able to move properly to adjust to the bending. Mode 3 demonstrates vertical bending a well as compression of the wing box. Mode 4 represents the wing under pure torsion. Figure 12 shows the wing again not being able to withstand the vibrational frequency of the torsion. Mode 5 as seen in figure 13 represents the wing when horizontal bending as well as compressive frequencies are applied to it. Again, the wing is not able to survive the vibration and will fail on the top skin near the root. The vibration analysis provided insight to the wing not being able to bend in certain directions without the structural integrity failing. A way to counteracting this is by increasing the thickness of the wing box, or by adding in more spars, spar caps, and ribs in order to reinforce the structure of the wing better.

IV. Conclusion

A proper investigation of the hexagonal wing box was able to be conducted given the initial conditions. The most important part of using a finite element software is not only being able to create the parts, but to be able to understand how the parts should act in order to confirm the results make sense. There is no point in getting results if they are not properly analyzed and understood. Taking a look at the results of the static wing analysis, the maximum von mises stress was less than aluminum yield stress which means the material would not yield. On top of that, the spanwise deformation showed it was at a maximum near the tip which means the structure deforms the most at the tip. When it came to buckling and vibration analysis, the wing structure buckled along the root. This means that in order to be able to carry heavier loads, the structure would need more reinforcement near the root in order to buckle under higher load values. While the wing was able to maintain its shape during a pure bending vibration analysis, given the other 4 mode cases the wing structure would fail to keep its shape. Some improvements to counteract this would be to increase the overall thickness of the wing, or to add more reinforcements to the wing skin to ensure it would be able to withstand the bending, tension, and compression vibrations applied to it.

V. Appendix

MATLAB Code

```
%Hexagonal Wing Root Values
x1=-.25; x2=0; x3=.25; y1=-.05; y2=.05; y3=.1; y4=.05; y5=-.05; y6=-.1; theta =
atand(.1/3.75);

%Wing Volume Calculations
skinroot_length = sqrt((.1)^2+(.5)^2);
skintip_length = sqrt((.05)^2+(.25)^2);
trap_area = ((skintip_length+skinroot_length)/2)*(3.75/cosd(theta));
trap_Volume= trap_area*.002;
trap_volume=trap_Volume*4;
skinarea = ((.2+.1)/2)*3.75;
skin_volume = skinarea*.003;
spar_volume = skin_volume*3;
ribs_area1=(.1*.2)*2;
ribs_area2 = .1*.2;
ribsavg_area = (ribs_area1+ribs_area2)/2;
ribs_area = 3*ribsavg_area;
ribs_volume = ribs_area*.002;
total_volume = ribs_volume+skin_volume+trap_volume;

%Gravitational Force Calculations
mass = 2810*total_volume;
force = mass*9.81

%Sectioned Pressure Calculations
press_force = 95000;
press_area = press_force/(2*trap_area);
PressTot = 95000/(2*trap_area);
x=1+.7+.3+.24+.56+.8;

WA1 = (1/x)*PressTot;
WA2 = (0.7/x)*PressTot;
WA3= (.3/x)*PressTot;
WA4 = (.8/x)*PressTot;
WA5 = (.56/x)*PressTot;
WA6 = (.24/x)*PressTot;
WA_TOT = WA1+WA2+WA3+WA4+WA5+WA6;
```