

Comparing Stiffness of 3D Printed UAV Airfoil Prototypes using Finite Element Methods

Saul A. Lopez
Stanford University

6/9/15

Executive Summary:

Recent developments in additive manufacturing have prompted NASA to search for a novel production process that leverages 3D printing to improve the stiffness to weight ratio properties of 3D printed structures for production flexibility in flight certifiable Unmanned Aerial Vehicles (UAVs). The purpose of this analysis is to model a benchmark test specimen of an airfoil prototype - 3D printed in Polylactic Acid (PLA) thermoplastic with 50% infill - to within 5% accuracy compared to actual test results in order to use the model in future conceptual explorations to judge ideas at a faster pace and lower manufacturing resource cost.

Two models were developed using ANSYS APDL and Workbench. The APDL model produced results within 3.2% error using a quad mapped mesh of 1.524mm thick shell elements compared to the test results, assuming a linear elastic isotropic material with a Young's Modulus of 1.5 GPa and Poisson's Ratio of 0.36. Although the Workbench model had a much more complex mesh, it produced a solution with a 0.42% error using a tetrahedral solid mesh compared to the test results. Workbench has the benefit of interfacing well with a Computer Aided Design (CAD) package, and therefore a shortened time to complete an analysis. APDL, although more involved, allowed for a greater degree of control of the analysis details, but Workbench allowed for more model experimentation and easier modification.

Future conceptual explorations will utilize Workbench to compare the relative stiffness of different test specimens early in the conceptual design phase. Once a concept moves to a detailed development phase, the APDL environment will be favored.

Table of Contents

1. Problem Description	3
1.1 Background and Motivation	3
1.2 Analysis Objectives	3
1.3 Part to be Analyzed	4
1.4 Test Setup and Loading	4
2. FEA Code	6
3. Model Description and Assumptions	6
3.1 Engineering Assumptions	6
3.2 Finite Element Model	7
A. ANSYS APDL	7
3.2.1 APDL: Elements and Mesh	7
3.2.2 APDL: Boundary Conditions and Applied Loads	8
B. ANSYS Workbench	8
3.2.3 Workbench: Elements and Mesh	8
3.2.4 Workbench: Boundary Conditions and Applied Loads	9
4. Results and Analysis	10
4.1 Experimental Results	10
4.2 APDL Model	11
4.3 Workbench Model	13
4.4 Comparing Models	15
5. Conclusions and Recommendations	16
Appendix	17
Appendix A	17
Appendix B	18
Appendix C	20

1. Problem Description

1.1 Background and Motivation

In summer 2014, a project at NASA Ames Research Center demonstrated that the development time for small (Class G) science-mission Unmanned Aerial Vehicles (UAVs) could be reduced to weeks rather than months or years by using a combination of 3D printing and modular repurposed systems¹. Using 3D printed components with an internal truss-based design, they qualified and flew a UAV that would traditionally be manufactured using carbon fiber, saving weeks in production time.

The direction from the NASA project sponsor encouraged focusing on developing a manufacturing process that would improve the stiffness-to-weight ratio of the 3D printed components, since that metric is associated with improved flight time, payload capacity, and aspect ratio. Although aircraft designs with very high aspect ratios offer performance improvements, they encounter production difficulties making wings that are stiff enough for their flight conditions. These wings may be prone to failure from aeroelastic flutter effects or large wing deflections that lead to instabilities as experienced by the NASA Helios prototype².

This analysis supports a team project for ME317B at Stanford where the team established previously in ME317A that the real value 3D printing provides in UAV development is not in manufacturing cost savings, but time savings and specifically, producing flight-worthy parts while shortening the entire development cycle from conceptual development to manufacturing and testing. The team conducted various tests to determine the relative stiffness of 3D printed components using different processes and in varying orientations.

In ME317B, a concept exploration was conducted in attempts to produce parts with high stiffness to weight ratios that would prove to be a worthy alternative to manufacturing UAV parts with carbon fiber or Kevlar. Thirty-six test specimens were created and tested using 18 different manufacturing post-processing techniques. In the future, the team wishes to include virtual testing by means of Finite Element Analysis (FEA) to gauge early concept viability. For reference, the results of the testing is included in **Table A-1**, **Appendix A**. This study will focus on one of the test specimens, in particular, the benchmark that was used to compare the same design post-processed by nickel plating.

1.2 Analysis Objectives

The analysis objective is to build a finite element model that accurately represents the test results to within 5% in order to compare future test specimens virtually. The model will be used to incorporate FEA into future concept development spirals in order to complete them faster with reduced manufacturing costs.

The model will be constructed both with ANSYS APDL and Workbench to compare efficacy and determine which would compliment the manufacturing process development better at different phases.

¹ <http://www.nasa.gov/ames/its-alive-ames-engineers-harvest-and-print-parts-for-new-breed-of-aircraft>

² <http://www.nasa.gov/centers/dryden/news/ResearchUpdate/Helios/>

1.3 Part to be Analyzed

The part to be analyzed is shown in **Figure 1**. The part was modeled as a scaled down version of a UAV airfoil wing and simplified for printing and testing purposes. Since the part is a test sample, it has no direct application of use besides gathering test data. It was made using a consumer grade desktop Fused Deposition Modeling (FDM) printer, Ultimaker 2. The material is Ultimaker-supplied bronze-colored Polylactic Acid (PLA) thermoplastic with 50% infill, made with code generated using Cura open-source software.

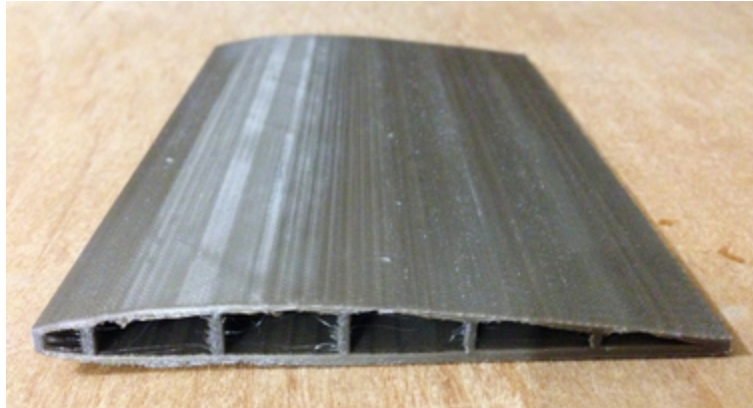


Figure 1: Testing fixture and load setup

The prototype test sample includes arbitrarily spaced I-beam walls (since only the relative stiffness to weight improvement is of concern) for added stiffness and has a uniform wall thickness of 1.524mm (0.060in), which is a rule-of-thumb for consumer grade FDM printers. Since this component is 3D printed layer by layer, it is anisotropic and the tensile strength will vary in different directions. To improve the loading response, the part was oriented such that the direction of the extruded filament is in the horizontal direction everywhere (rotated -90 degrees vertically from the above depiction). This orientation was chosen to prevent loads between layer interfaces where the part is not as strong.

The dimensions of the part are 152.4mm (6in) long by 97.73mm (3.85in) wide and 10.16mm (0.4in) tall.

1.4 Test Setup and Loading

The complete test fixture is shown **Figure 2** loaded with a different test sample. The test setup includes a digital depth gauge, CNC machined aluminum rear part clamp, and 3D printed front plastic clamp that facilitated applying the the load. A Flexbar digital depth gauge was used (model 16365) with 0.0005in resolution and 0.001in claimed accuracy. It has a spring-loaded rod, is clamped at one end, and held in position with a set screw to ensure that only vertical deflection data is captured.

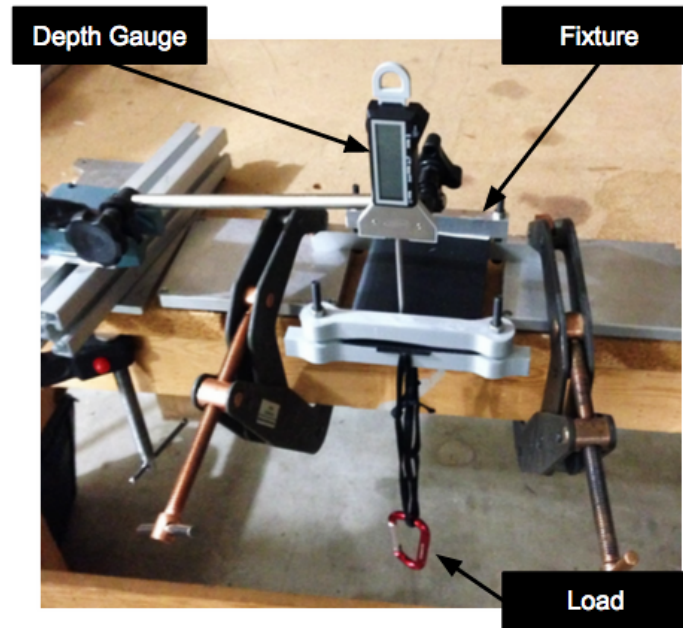


Figure 2: Testing fixture and load setup

Note that the depth gauge position is placed directly behind the 0.5in plastic clamp at the center of the part width. That is where the deflection data will come from in the models.

The plastic clamp was 3D printed to obtain the correct airfoil profile. A notch was then cut into it to ensure that the rope carrying the weight would sit flush with the part being tested.

The clamps that fixed the rear end of the test specimen relative to the test bed are shown in **Figure 3**. They were CNC machined out of aluminum to obtain the desired airfoil shape with a much higher stiffness than any of the components being tested. After conducting test trials, sandpaper was added to the clamps using double-sided tape in order to prevent the prototype parts from slipping.



Figure 3: Part clamping fixture showing sandpaper adhered to airfoil geometry

Since the part had only a thin layer of material on the outer edges, foam was added in between the clamping region to prevent deformations due to tightening the clamp screws. The parts were only tested in their elastic region, but due to some inadvertent failures, the failure mode was confirmed to be in tension at the top side of the test specimen near the clamping region as expected.

The test setup was used to determine the stiffness of the different specimens through a force-deflection test. Using Hooke's Law as the governing equation:

$$F = -kx, \quad (\text{Eq.1})$$

the structural stiffness can be found by rewriting Eq. 1 in terms of deflection as follows in Eq. 2.

$$k = \frac{F}{\delta} \quad (\text{Eq.2})$$

Then to find the stiffness to weight, k would simply be divided by the mass. Since the force and mass is constant, the analysis will focus on finding the vertical displacement, δ . It will be compared to experimental results.

2. FEA Code

Two ANSYS codes were used to run the analysis: Mechanical APDL R15.0 Academic and Workbench 15.0. The analysis was done on a MacBook Pro using BootCamp with a replaced Samsung 850 EVO SATA III Internal SSD (MZ-75E500B/AM) with sequential read speed of 550MB/s, sequential write speed 520MB/s and with 8GB of RAM.

3. Model Description and Assumptions

3.1 Engineering Assumptions

For the sake of the analysis, linear elastic material behavior is expected. It is assumed to be isotropic with a Young's Modulus of 1.5GPa based on experimental data for PLA printed with 50% infill³. This assumption is justified since due to the way the material was oriented during printing, all the strands are in the same orientation and the loading condition concerns forces applied in only one direction. Since the material was printed with 50% infill, it is porous. The Young's Modulus uses the data from my3dmatter.com that assumes the voids are all part of the material. Furthermore, a Poisson's ratio of 0.36⁴ is used from literature.

The part was assumed to be fixed at the clamping end since there was no observable slipping after the sandpaper was added and the clamps were tightened. The compression loads caused by the clamping are ignored at both ends since there were no observable geometric deformations.

The load applied to the forward end of the test specimen is assumed to be distributed along the length of the wing due to the matching profile clamping of the plastic fixture.

³ <http://my3dmatter.com/influence-infill-layer-height-pattern/>

⁴ <http://onlinelibrary.wiley.com/doi/10.1111/j.1541-4337.2010.00126.x/full>

3.2 Finite Element Model

A. ANSYS APDL

The geometry of the model was created from scratch in ANSYS APDL to obtain the midside planes for use with shell elements. Lines were created from keypoints as shown in **Figure B-1**, in **Appendix B**. The geometry and location data for the keypoints was obtained from SolidWorks as shown in **Figure B-2**. The lines were then extruded to obtain the desired areas and geometry shown in **Figure B-3**. Finally, since the I-beam walls had to be extruded in a separate operation from surrounding walls, adjacent areas were glued together using a boolean operation.

3.2.1 APDL: Elements and Mesh

The model uses SHELL181 elements, which are four noded elements with six degrees of freedom at each node including three translations and three rotations. For the element stiffness, the “bending and membrane” were chosen in Keyopt 1. The shell elements have a thickness of 1.524mm for the entire structure. A quad mapped mesh was used on areas with line subdivisions to obtain the mesh shown in **Figure 4**.

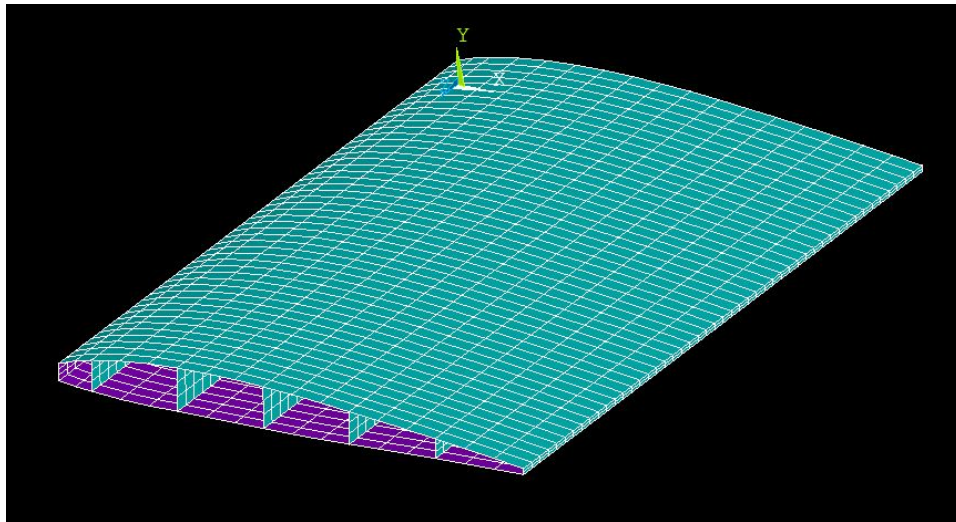


Figure 4: Meshed model showing shell elements

The model is made up of 2592 elements with 2401 nodes. There are 17 elements along the width and 48 elements along the length of the model. All of the vertical walls have three elements through the height except the smallest wall at the trailing end of the wing, which has only two.

Care was taken to ensure that no mesh discontinuities were present between the I-beam walls and the surrounding walls. An approximate number of elements was chosen to achieve the relative resolution necessary and the elements in the mesh are sized in 3.175mm increments along the length to facilitate ease of applying boundary conditions and loads, and gathering nodal data at important locations.

3.2.2 APDL: Boundary Conditions and Applied Loads

The weight of the clamp and fixture that was used to apply the load was 106g and it is included in the analysis since it was accounted for in the digital depth gauge before it was zeroed. The depth gauge spring mechanism contributed an added 140g, and in total the fixture loading with no weight was 2.413N. The actual part was clamped with a 12.7mm (0.5in) fixture, and so the model constrains (zero displacement for all degrees of freedom) all of the nodes that cover the width of the 12.7mm section at the rear end as shown in **Figure 5**.

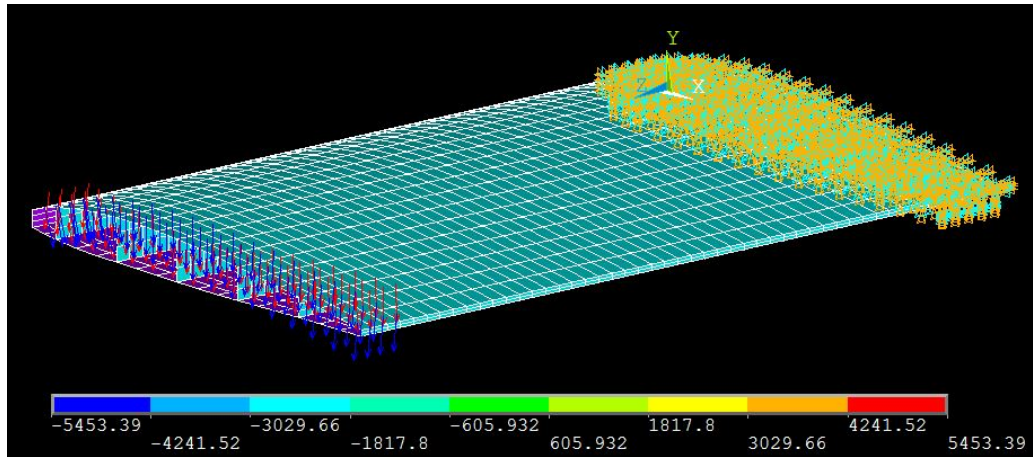


Figure 5: Meshed model showing boundary conditions and applied loads in APDL

Initially, the 13.54N force that includes the fixture and 1.5lb weight was applied as a pressure to the top face only. Using a cross-sectional area of 1241.171mm^2 (the width, 3.85in multiplied by the fixture thickness, 0.5in) results in a pressure of 10.907 kPa applied downward normal to the face. The pressure load was then split between the top and bottom face evenly at 5.4535 kPa, leading to a difference of less than 0.1% in the results. The analysis was completed using the split pressure loading as shown in **Figure 5**.

B. ANSYS Workbench

The geometry for the Workbench model was obtained directly from the CAD shown in **Figure B-4** in **Appendix B**. Before importing the model into Workbench, it was adjusted to be 12.7mm (0.5in) less in length due to the way the clamping constraints were applied at the fixed end. The CAD model was saved as a *.IGES file and imported into the Workbench environment, then linked to a static structural study.

3.2.3 Workbench: Elements and Mesh

A number of mesh strategies were attempted in Workbench. Notably, the automatic mesh, a hex dominant mesh, and a hex dominant mesh with refinements through the thickness of the part height. The automatic mesh was found to overpredict the stiffness by about 24%. The Hex Dominant mesh was an improvement and resulted in the mapped mesh shown in **Figure 6**.

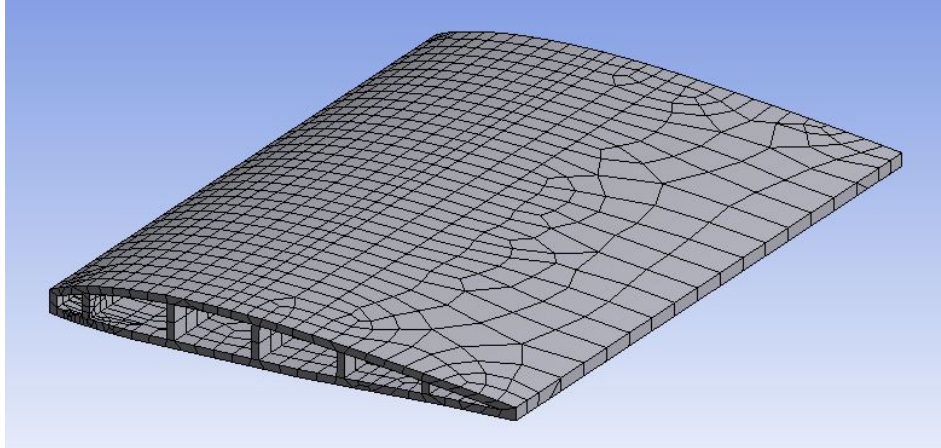


Figure 6: Mesh generated in Workbench using Hex Dominant method with no refinements

Although the mesh seems similar to the one generated using shell elements and looks like it would yield favorable results, it still overpredicts the stiffness by about 12%. Once the Hex Dominant mesh is refined to add more elements through the thickness of the part height as shown in **Figure 7**, the results begin to resemble the asymmetrical deflection as in the APDL model. The mesh below was used to carry forward the analysis and includes 16298 elements with 28729 nodes and midside nodes set to be program controlled.

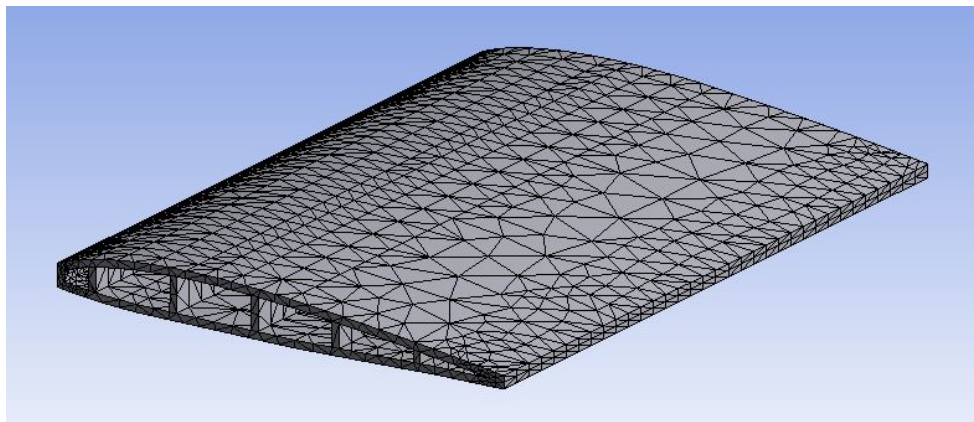


Figure 7: Mesh generated in Workbench using Hex Dominant method with refinements through the part height

The reason for the improvement is that the mesh is made up of SOLID187 elements which are 10-node tetrahedral solids with 3 degrees of freedom only in translation. If there are not enough elements through the thickness, the model will not properly account for rotational effects. The model has also generated contact elements in the form of CONTA174 and TARGE170 elements as it meshed the imported geometry.

3.2.4 Workbench: Boundary Conditions and Applied Loads

With the modified geometry imported into Workbench, the clamp was ignored at the rear and a fixed support was applied to the face as shown in **Figure 8**. The part has been rotated for clarity.

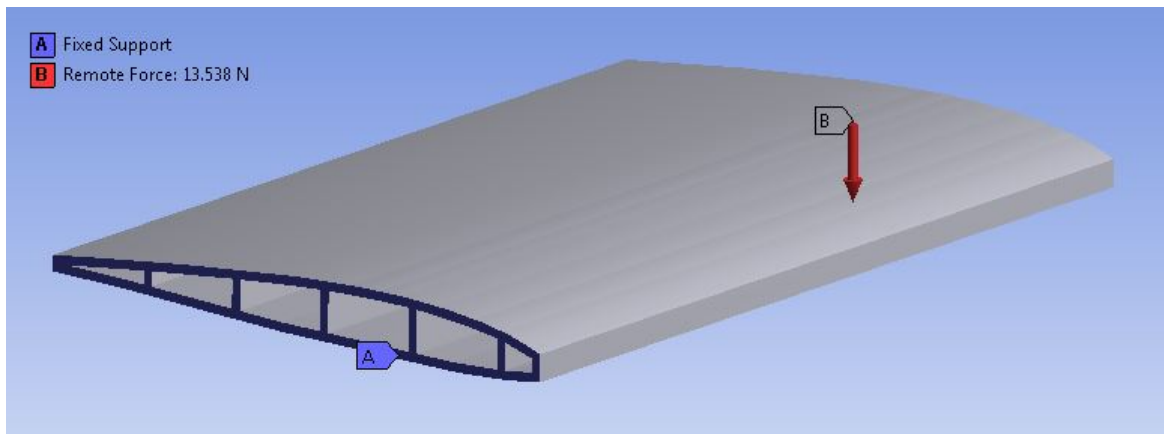


Figure 8: Applied loads and boundary conditions in Workbench showing rotated part

Multiple loading conditions were also attempted. Applying the load at the face of the forward end resulted in an underprediction of stiffness by about 17% due to shear effects. The final loading condition was modeled as a point load of 13.538N applied at the center of the part, halfway through where the clamp would be, or 0.25in from the forward end.

4. Results and Analysis

4.1 Experimental Results

The experimental test results are shown in **Figure 9**. The results show that the linear elastic material behavior assumption is valid.

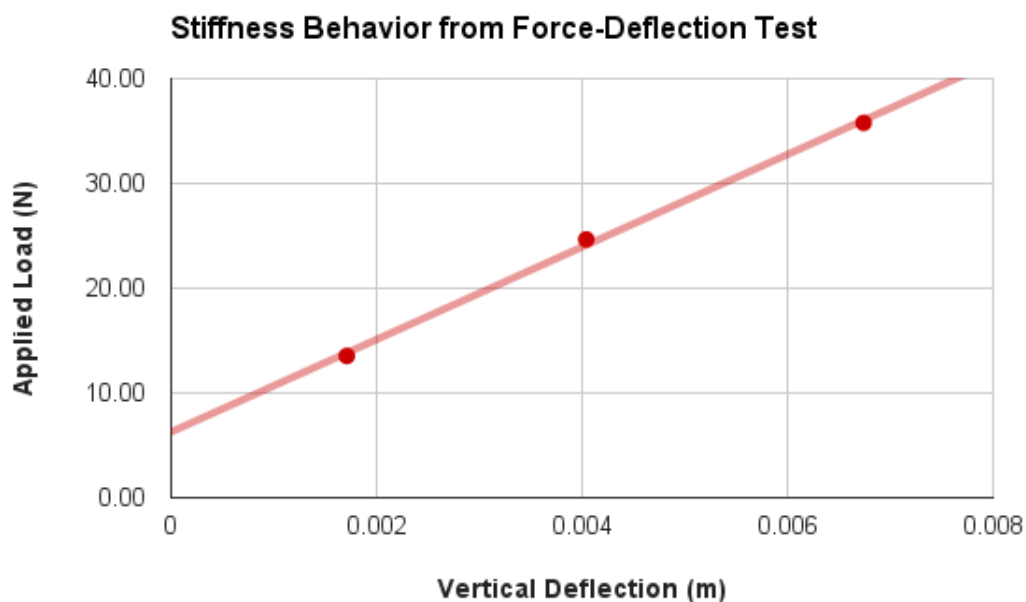


Figure 9: Force-deflection test results data showing linear elastic behavior

The focus of this study is on matching the deflection for the 13.538N load (2.5 lb + fixture). The actual test result data are listed in **Table 1**.

Table 1: Experimental results data for stiffness/weight ratio tests

Deflection (mm)	Applied Load (N)	Stiffness (N/m)	Stiffness/Weight (N/kg-m)
0.00171	13.538	7916.96	136.90
0.00404	24.662	6104.46	105.56
0.00674	35.786	5309.50	91.81
	Average	6443.64	111.42

Only one test specimen of this type is being considered for this study and only one point. It is understood that this number poses limitations and that the repeatability of the results may have errors, however, this consideration will not detract from the usefulness of the model should it accurately represent the testing behavior.

4.2 APDL Model

The von Mises contour plot stress results obtained from the APDL model are shown in **Figure 10**. As expected, the plot shows the highest stress point occurring in tension near the fixed end of the test specimen.

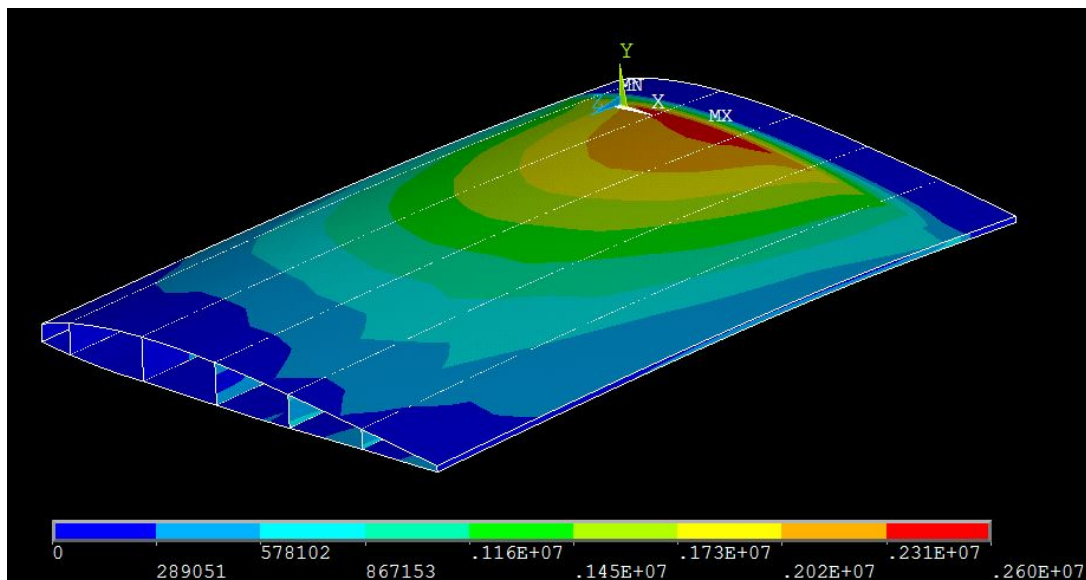


Figure 10: Von Mises stress contour plot from APDL model

Furthermore, the maximum stress value is 2.6015 MPa as shown in **Table 2**. Since the Yield stress for PLA printed with 50% infill is 17 MPa, it once again assures that the loading occurs in the elastic deformation region of the material.

Table 2: Maximum von Mises Stress nodal output taken from APDL model

Nod e	X Position (mm)	Y Position (mm)	Z Position (mm)	Max von Mises Stress (MPa)
1320	36.448	8.587	15.875	2.6015

Figure 11 illustrates an example of the actual failure mode of a different prototype. The photo shows tensile failure of the material, but interestingly, it did not occur along a filament layer interface. One explanation is that the material is printed with 100% infill and not 50% as in the benchmark prototype.

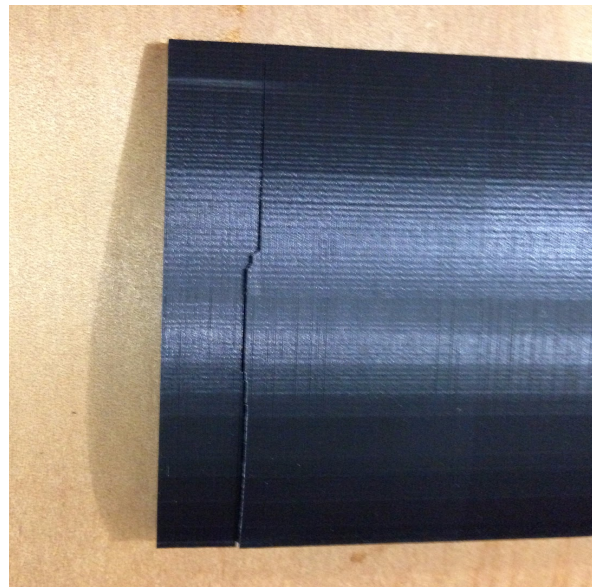


Figure 11: Actual failure location for a different test sample

Figure 12 shows the displacement contour plot for the test sample. Interestingly, the sample does not deflect perfectly flat as imagined. The model results make sense however, because the area of the thin wing section will be less stiff due to a smaller distance to the neutral axis.

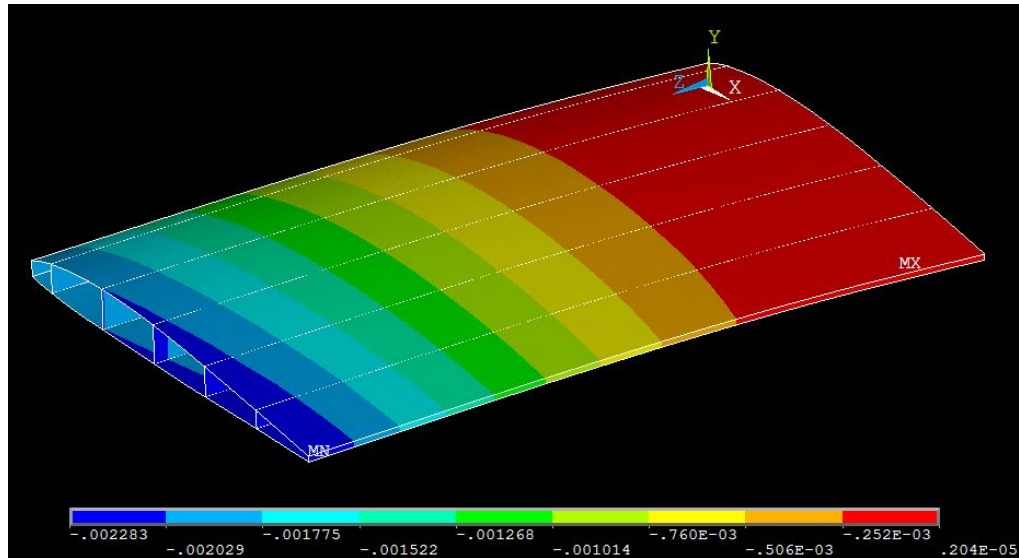


Figure 12: Displacement contour plot from APDL model

The nodal displacement data for the APDL model is listed in **Table C-1** in **Appendix C**. To account for the 0.25in rod diameter on the depth gauge and for shifts between test samples, elements were chosen as in **Figure 13** and averaged to determine the displacement.

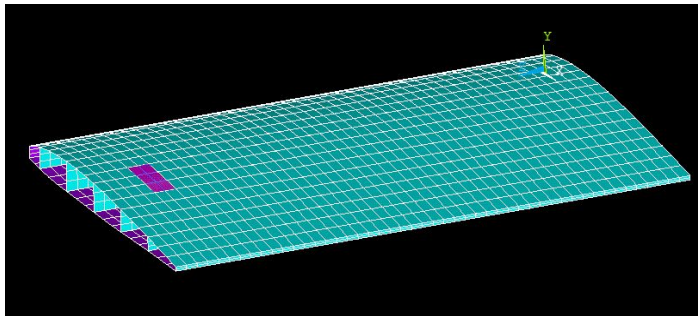


Figure 13-a

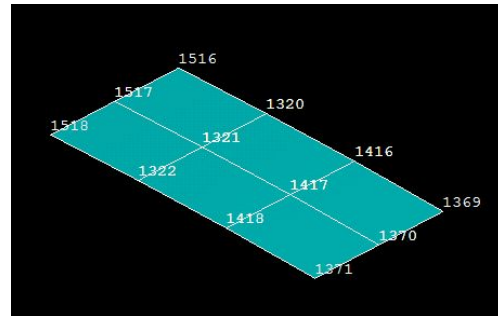


Figure 13-b

Figure 13: (a) Model showing region of deflection reading and selected elements of interest. (b) Selected elements and associated nodes.

The model predicts the deflection and so the stiffness at that loading condition to within a 3.2% error. The average displacement is -1.765mm compared to the actual measured displacement of -1.71mm.

4.3 Workbench Model

Figure 11 depicts the von Mises stress contour plot for the Workbench model. It is very similar to the plot produced for the APDL model created with shell elements.

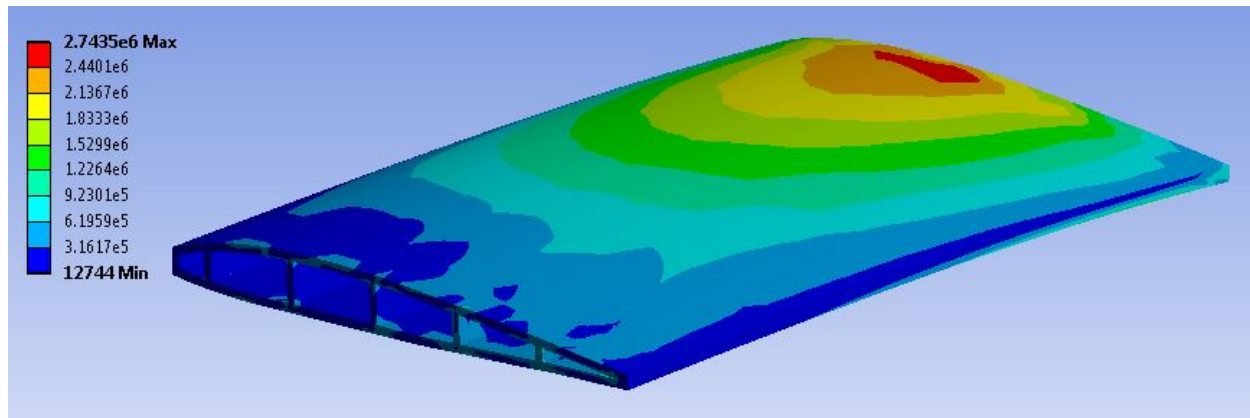


Figure 14: Von Mises stress contour plot for Workbench model

On thing that is noticeable however, is that the contour lines of the shell element model are smoother. The biggest discrepancy between the models occurs near the edges. Since the model in Workbench had to be shortened, as opposed to fixing the entire clamping area, it could lead to slightly different stresses near the edges of the part.

The von Mises stress for the Workbench model is listed in **Table 3** below. Once again, with a max stress of 2.8717 MPa, it is well within the yield condition and is valid for elastic behavior.

Table 3: Maximum von Mises Stress nodal output taken from APDL model

Nod e	X Position (mm)	Y Position (mm)	Z Position (mm)	Max von Mises Stress (MPa)
3300	26.858	97.038	0.13559	2.8717

The vertical displacement contour plot shown in **Figure 15** also accounts for the unsymmetric deflection especially near the forward end of the test specimen, however, it is less pronounced than the shell model produced in APDL.

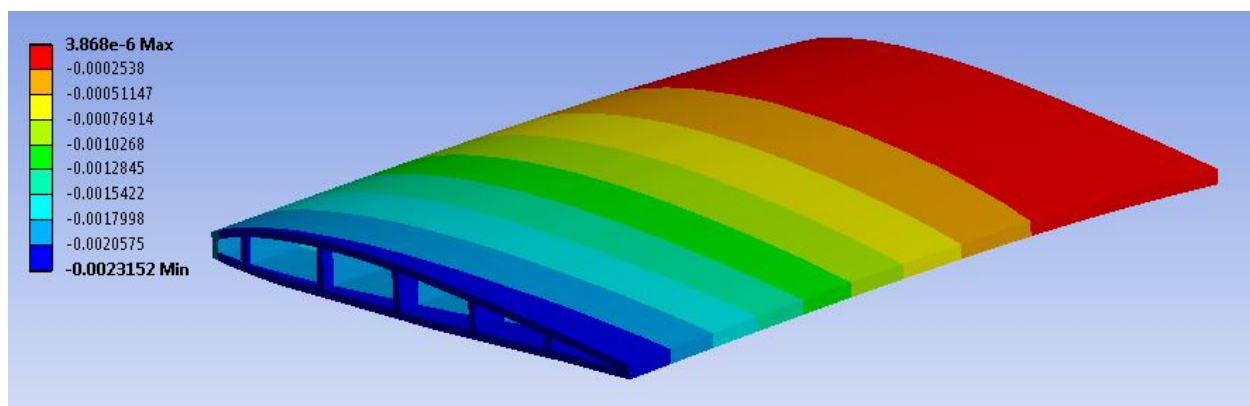


Figure 15: Vertical displacement contour plot for Workbench model

The region of interest for the model's nodal displacement is shown in **Figure 15**. Since Workbench produced a mesh of tetrahedral solid elements, choosing them is not as straight forward as the shells. Once again, to account for the diameter of the probe as well as minor shifts in testing, the region was chosen in the area of where the measurement was taken and an average of the nodal displacements was obtained.

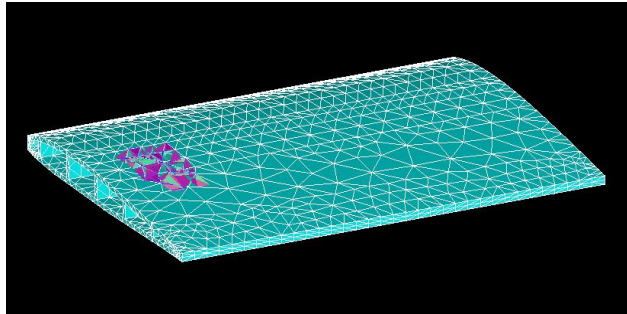


Figure 16-a

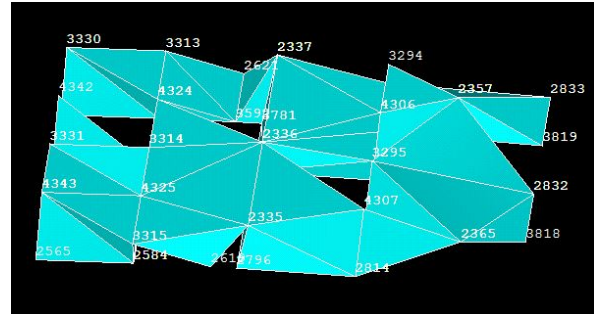


Figure 16-b

Figure 15: (a) Model showing region of deflection reading and selected elements of interest. (b) Selected elements and associated nodes for Workbench model

The nodal displacement data is listed in **Table C-2** in **Appendix C**. The model predicts the deflection/stiffness to within 0.42% error. The average displacement is -1.7171mm compared to the actual measured displacement of -1.71mm.

4.4 Comparing Models

The models have two completely different loading and boundary conditions and use different elements but both are accurately able to predict the part deflection to within the target of 5%. **Table 4** shows some of the important metrics for the two models. The % Error Energy Norm listed is not for any particular region, rather, for the entire model, therefore, it is expected that it would be reduced to be within the 10% rule-of-thumb.

Table 4: APDL and Workbench model comparison data

	APDL Model	Workbench Model
# Elements	2592	16298
# Nodes	2401	28729
% Error Energy Norm	22.5	17.8
Max von Mises Stress (MPa)	2.6015	2.8717
Average Deflection (mm)	1.765	1.717
% Error (actual = 1.71mm)	3.2	0.42

Despite the evenly spaced quad mapped mesh in the shell elements model created in APDL, not only is the accuracy reduced, but it shows a higher associated % Error Energy Norm. Even though the Workbench model seems to have a less ordered mesh, it is still more accurate, likely because it has six times more elements than the APDL model and almost 12 times more nodes. For a simple model such as the one outlined in this report, it may not be prohibitory to run such complex meshes, but if the complexity is increased, one would want to avoid it.

5. Conclusions and Recommendations

Although only one data point was compared for the prototype, with so many possible sources of error ranging from material properties, to idealized loading conditions and fixturing, it is somewhat surprising that the models give results so close to the actual test data. Though future models may vary due to inconsistencies in material properties from different vendors, manufacturing defects, or loading artifacts, both results give confidence that FEA produces trustworthy results for evaluating future concepts when careful consideration is given to the model assumptions, elements, mesh, loads, and boundary conditions. When the model is not properly meshed for example, as shown with the automatic mesh only in Workbench, the errors in the model were an order of magnitude greater, or up to about 20% discrepancy from the test results.

Part of the reason the Workbench model turned out to have more accurate results is that the interface makes it easier to iterate on the loading and meshing conditions than APDL. So even though the model may not start out very accurate, a better one can be found through experimentation. One example of why Workbench is well-suited for experimentation is the ability to toggle boundary conditions or mesh properties on and off using the suppress feature. That was particularly useful since it allows the user to compare options very quickly. An added bonus with Workbench is the ease of interfacing the analysis with CAD packages.

The downside of Workbench, however is that some of what is happening seems to be out of the user's control. This part about it invited skepticism towards the results, whereas APDL seems more deterministic since it feels like more decisions are made by the user. The APDL model is expected to improve and if more elements were used it might approach or better the accuracy of the Workbench model, and perhaps more efficiently.

Another surprising thing was that the 3D printed materials were well-modeled by linear elastic materials models using experimental data for the material properties. As academia and industry start to use 3D printing more and more for production settings, it will become increasingly important to characterize important material properties of the materials used by consumers every day.

Moving forward with the project to leverage 3D printing to improve performance and production of UAVs, FEA will play an integral role not only in the detailed design phase, but in the conceptual stages as well.

Appendix

Appendix A

Table A-1: Ranked stiffness/weight testing data for all test samples

Test Specimen	Material	Post Process	Stiffness(N/m)	Stiffness/Weight (N/kg-m)	Weight (g)
Aluminum FDM Inserts	ABS (A)	N/A	19297.97	303.19	63.65
CF FDM Inserts	ABS (A)	N/A	16456.54	273.23	60.23
Plated FDM Benchmark (0.007")	ABS (A)	Plated	24805.08	254.12	97.61
Plated FDM Benchmark (0.007")	ABS (A)	Plated	23688.09	249.35	95
Corrugated CF Double	Carbon Fiber	N/A	18869.90	161.01	117.2
Corrugated CF Double	Carbon Fiber	N/A	11704.87	99.87	117.2
FDM Benchmark (Horizontal)	PLA (P)	N/A	5648.92	97.68	57.83
FDM Inserts, No Inserts	ABS (A)	N/A	4765.35	93.95	50.72
Wireprint, Orbit plated Copper (0.1mm)	Fiber Filled Nylon	N/A	5903.44	89.91	65.66
Plated FDM Benchmark (0.11mm)	ABS (A)	Plated	8320.45	85.24	97.61
Lattice with CF Inserts	Acrylic (SLA)	Inserts	4264.86	79.58	53.59
Wireprint, SLS, FFN, Hybrid (top skin)	Fiber Filled Nylon	N/A	3694.16	79.51	46.46
Double Layer CF	Carbon Fiber	N/A	3072.61	76.66	40.08
FDM Benchmark (bad quality)	PLA (P)	N/A	2622.47	72.30	36.27
Foam Filled Aluminum Inserts	ABS (A)	Inserts	4828.97	66.85	72.24
Within Enhance Lattice (unoptimized)	Acrylic (SLA)	N/A	2583.86	65.07	39.71
Single Layer CF (no foam)	Carbon Fiber	N/A	1308.15	58.11	22.51
Wireprint, SLS, Hybrid (top skin)	Fiber Filled Nylon	N/A	2448.66	54.78	44.7
Wireprint SLS	Fiber Filled Nylon	N/A	1687.28	52.81	31.95
Lattice with Wood Inserts	Acrylic (SLA)	Inserts	2517.93	45.58	55.24
Single Layer CF	Carbon Fiber	N/A	1182.76	43.94	26.92
Double Layer CF	Carbon Fiber	N/A	1356.18	42.59	31.84
Corrugated CF Single (double skin)	Carbon Fiber	N/A	3682.32	42.30	87.06
Wireprint Angled	Acrylic (SLA)	N/A	1893.66	41.87	45.23
Within Enhance Lattice Plated	Fiber Filled Nylon	Plated	3041.07	38.67	78.65
SLA Lattice Inserts (no inserts)	Acrylic (SLA)	N/A	1488.71	35.00	42.53
Within Enhance Lattice	Fiber Filled Nylon	N/A	1277.36	30.77	41.51
Shell Skin FDM Chicken Wire	ABS (A)	N/A	1487.69	26.15	56.88
Double Layer FDM	PLA (P)	N/A	823.19	25.34	32.49
Within Enhance FDM	PLA (P)	N/A	1072.78	24.79	43.28
Shell Skin FDM (Vertical)	ABS (A)	N/A	1274.43	24.10	52.88
Within Enhance Lattice	Fiber Filled Nylon	N/A	854.85	19.72	43.36
Shell Skin FDM Chicken Wire	ABS (A)	N/A	1073.95	18.11	59.31
Shell Skin FDM (Horizontal)	ABS (A)	N/A	797.11	15.00	53.14
Single Layer FDM	PLA (P)	N/A	BREAK	-	19.95
Within Enhance Lattice Only (no skin)	Acrylic (SLA)	N/A	BREAK	-	19.45

Appendix B



Figure B-1: Lines created from airfoil geometry and dimensioned in SolidWorks

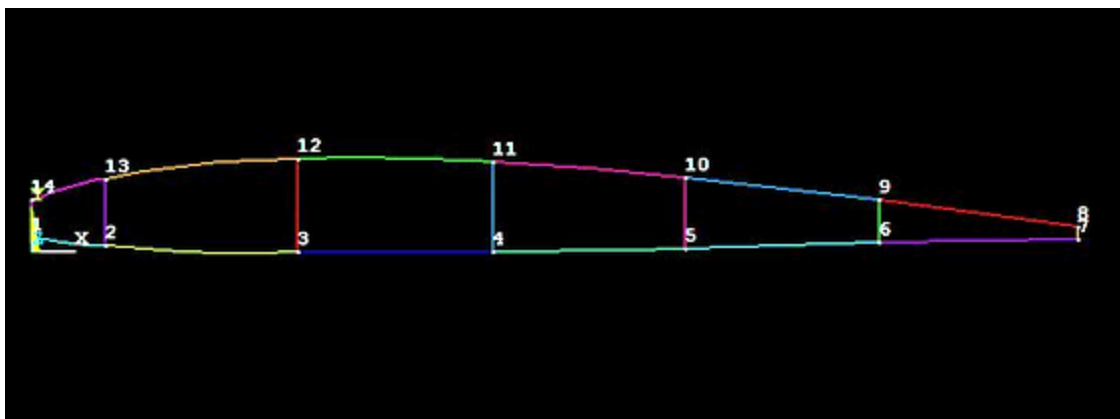


Figure B-2: Line elements created from keypoints in APDL

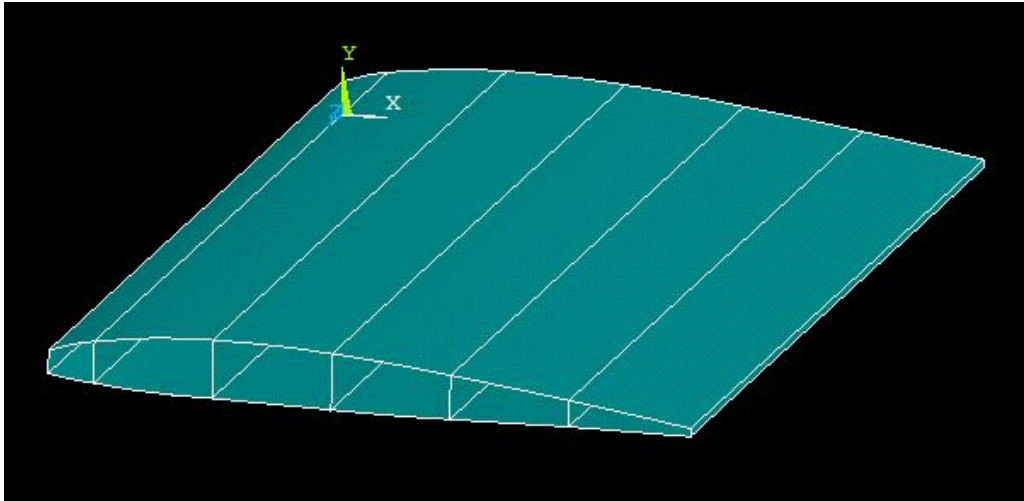


Figure B-3: Plane geometry in APDL created from extruded lines

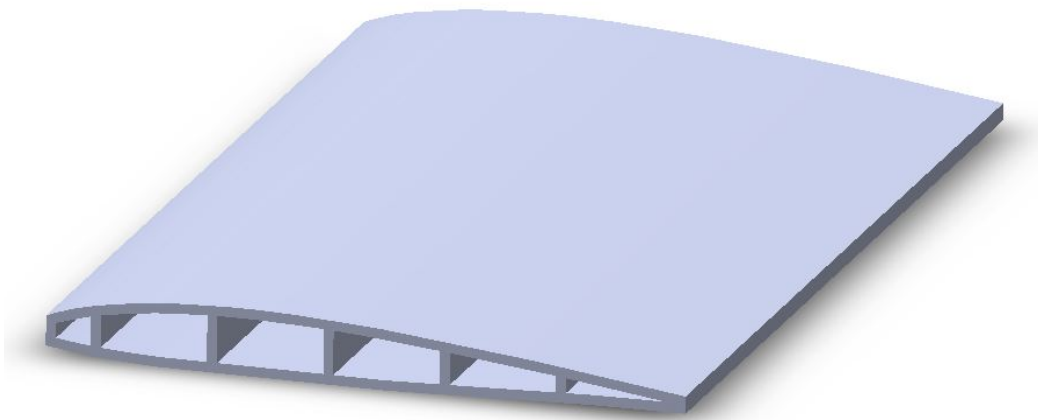


Figure B-4: CAD geometry created in SolidWorks

Appendix C

Table C-1: Nodal position and deflections for region of interest from APDL model

Node	X Position (mm)	Y Position (mm)	Z Position (mm)	Deflection (mm)
1320	42.384	8.29	133.35	-1.6863
1321	42.384	8.29	136.53	-1.7545
1322	42.384	8.29	139.7	-1.823
1369	54.249	7.4	133.35	-1.7234
1370	54.249	7.4	136.53	-1.7941
1371	54.249	7.4	139.7	-1.8652
1416	48.321	7.9091	133.35	-1.704
1417	48.321	7.9091	136.53	-1.7733
1418	48.321	7.9091	139.7	-1.843
1516	36.448	8.587	133.35	-1.6701
1517	36.448	8.587	136.53	-1.7379
1518	36.448	8.587	139.7	-1.8061
			Average Deflection (mm)	-1.765
			Actual Deflection (mm)	-1.71
			% Error	3.2

Table C-2: Nodal position and deflections for region of interest from Workbench model (note: coordinate systems origins do not match between models)

Node	X Position (mm)	Y Position (mm)	Z Position (mm)	Deflection (mm)
2337	33.341	97.326	28.894	-1.5145
2357	44.127	98.215	26.459	-1.5888
2621	31.262	98.738	28.762	-1.5135
2365	45.675	98.363	14.381	-1.8666
2814	39.674	99.332	12.326	-1.8968
2833	49.042	100.23	28.762	-1.5454
2336	33.341	97.326	21.467	-1.6772
2619	31.262	98.738	12.326	-1.8755
2832	49.042	100.23	28.762	-1.732
2565	21.077	98.414	12.326	-1.8509
2584	26.78	98.561	12.326	-1.8651
2796	32.786	98.82	12.326	-1.8787
3818	49.042	100.23	16.435	-1.8267
3819	49.042	100.23	24.653	-1.6382
3294	39.811	97.814	28.762	-1.5295
4306	39.811	97.814	24.653	-1.6204
3295	39.811	97.814	20.544	-1.7121
4307	39.811	97.814	16.435	-1.8044
3592	31.262	98.738	24.653	-1.603
4324	26.858	97.038	24.653	-1.5943
3314	26.858	97.038	20.544	-1.6841
4325	26.858	97.038	16.435	-1.7745
3315	26.858	97.038	12.326	-1.8653
4325	26.858	97.038	16.435	-1.7745
4343	21.082	96.889	16.435	-1.7608

3390	19.722	97.881	4.1088	-1.9919
4342	21.082	96.889	24.653	-1.5819
2621	31.262	98.738	28.762	-1.5135
			Average Deflection (mm)	-1.7171
			Actual Deflection (mm)	-1.71
			% Error	0.42