

FEA Analysis of Composite Compliant Beams in CATIA and Abaqus



Jacob Middleton
& Tony Kitchen

12/13/2019

Contents

Introduction	1
Analysis	1
Materials and Dimensions	3
CAD Results	4
CATIA.....	4
Abaqus	7
Conclusion and Observations	9
Lessons Learned.....	10

Introduction

The main goal of this project was to develop an FEA model in both CATIA and Abaqus that would evaluate a beam made from a composite material. Analytical background for the models was developed using small deflection equations, as well as compliant mechanisms principles, to compare those different methods as they pertained to this problem

In pursuit of the analytical foundation, we first needed to establish a set of material properties to use for the composite in the equations, primarily a unified elastic modulus that could be used in the equations. To find this value, we first generated an axial load model in CATIA using the selected materials, with a core to shell ratio of 0.33. Loading this model revealed a combined elastic modulus for the composite, which was then used in later analysis to produce estimates. Following the written analysis, models were developed to produce simple cantilever beams under bending due to point loads. We used different beam lengths, to see how that impacted the accuracy of the CAD results relative to the written analysis.

Going into this project, we anticipated that Abaqus would be better suited to FEA applications, as that is what it is designed for, whereas on CATIA, FEA is just one of many features.

Analysis

The following mathematical models were used for the analytical foundation. It was determined that a vertical deflection of 2% of the length of the beam would be considered “small” for this analysis, and we focused on the end of the beam. The objective of the analysis was to identify a force for each length that would result in that vertical deflection “ y ”

$$y = \frac{Fx^2}{6EI} * (3L - x)$$

-This equation is for small deflections, $y=0.02L$; $x=L$ for beam end

Figure 1: Model 1

$$b = y, \gamma = 0.8517, \frac{b}{L} = \gamma \sin \theta, K_{\theta} = \gamma \Pi, \frac{FL^2}{EI} = K_{\theta} * \theta$$

-These are the compliant mechanisms approximations.

Figure 2: Model 2

Models 1 and 2 produced very similar force values at each length tested. The results from this analysis for a beam with a 1 mm out of plane depth are below.

A: $l=5$ inches, $F=2165N$, $y_{\text{expected}}=2.54mm$

B: $l=8$ inches, $F=845.87N$, $y_{\text{expected}}=4.064mm$

C: $l=12$ inches, $F=375.94N$, $y_{\text{expected}}=6.096mm$

Figure 3: $b=1mm$; Force calculated, and expected displacement from that force

With Model 1, increasing the depth of the model, or “b,” results in a force that is multiplied by the same value the “b” value was. So to increase from 1 mm to 3 inch depth, we simply multiplied the F value for a given length by 76.2.

The following image shows a cross sectional view of the model that was axially loaded in CATIA to determine an E value for this analysis.

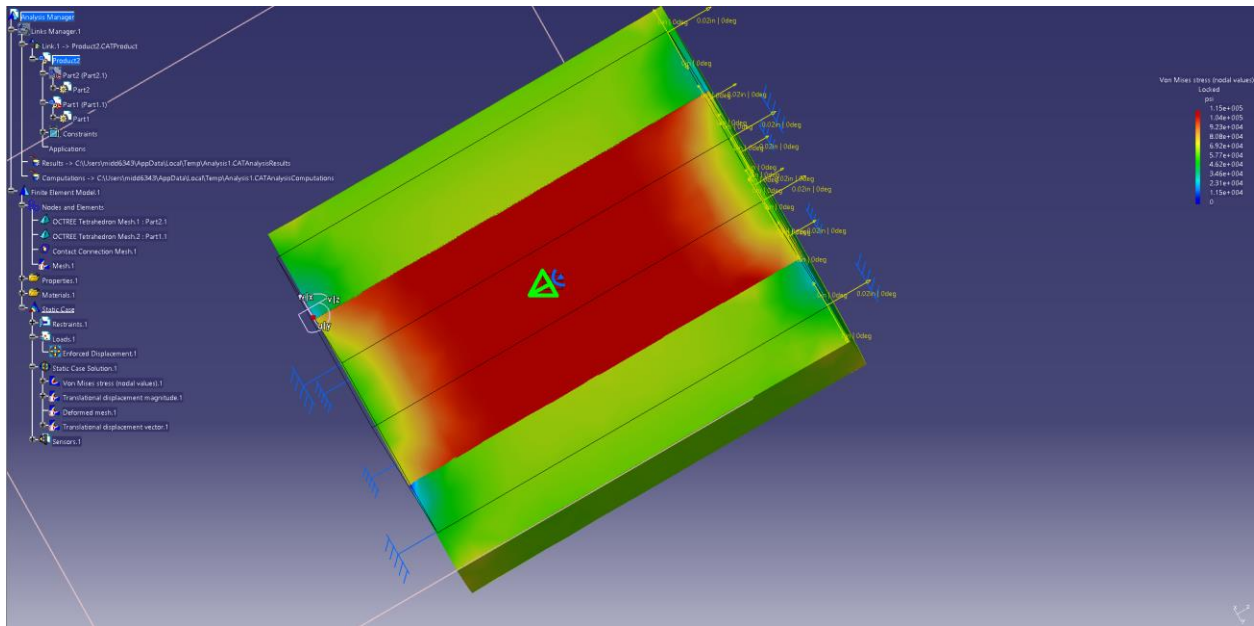


Figure 4: Cut plane analysis of axially loaded specimen in CATIA. The load is a traction, rather than a point load.

A similar analysis was performed in Abaqus, and produced a surprisingly similar E value. The results of that model are shown below.

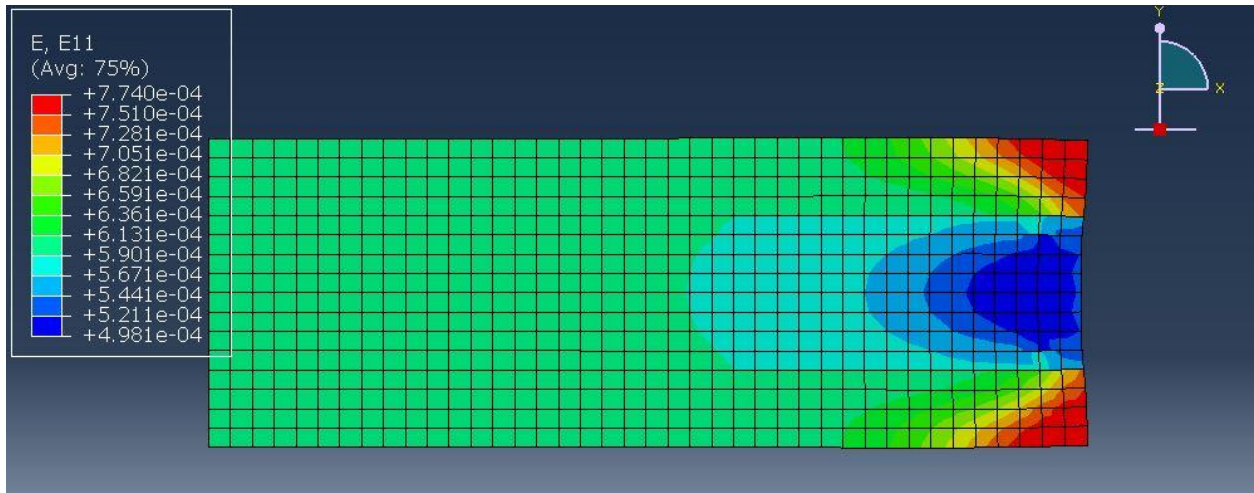


Figure 5: Axially loaded Abaqus model; E11=strain in the x direction; The load was a 34.47 MPa traction in the x direction

The value we determined from our analysis in both programs is shown below.

$$E=53281.89 \text{ MPa}$$

Materials and Dimensions

The materials used in the composite were as follows:

Material	E (psi)	Sy (psi)	Poisson's Ratio	Density (lb/in ³)
Magnesium Alloy (shell)	6.5×10^6	19000	0.33	0.065
Aluminum Alloy 1100 O (core)	1.04×10^7	5000	0.34	0.1

The constant dimensions of the beam were as follows:

Total h (in)	Core h (in)	Core/shell area ratio
2	1	1

CAD Results

CATIA

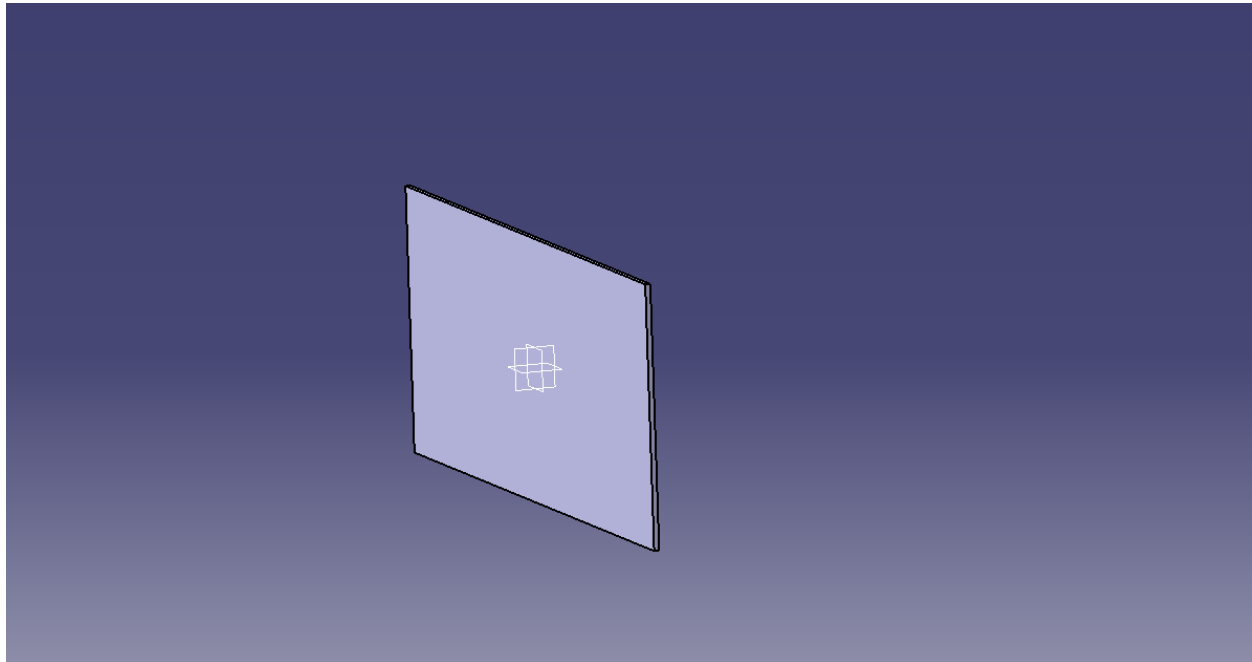


Figure 6: Isometric view of the beam, showing its 1 mm thickness

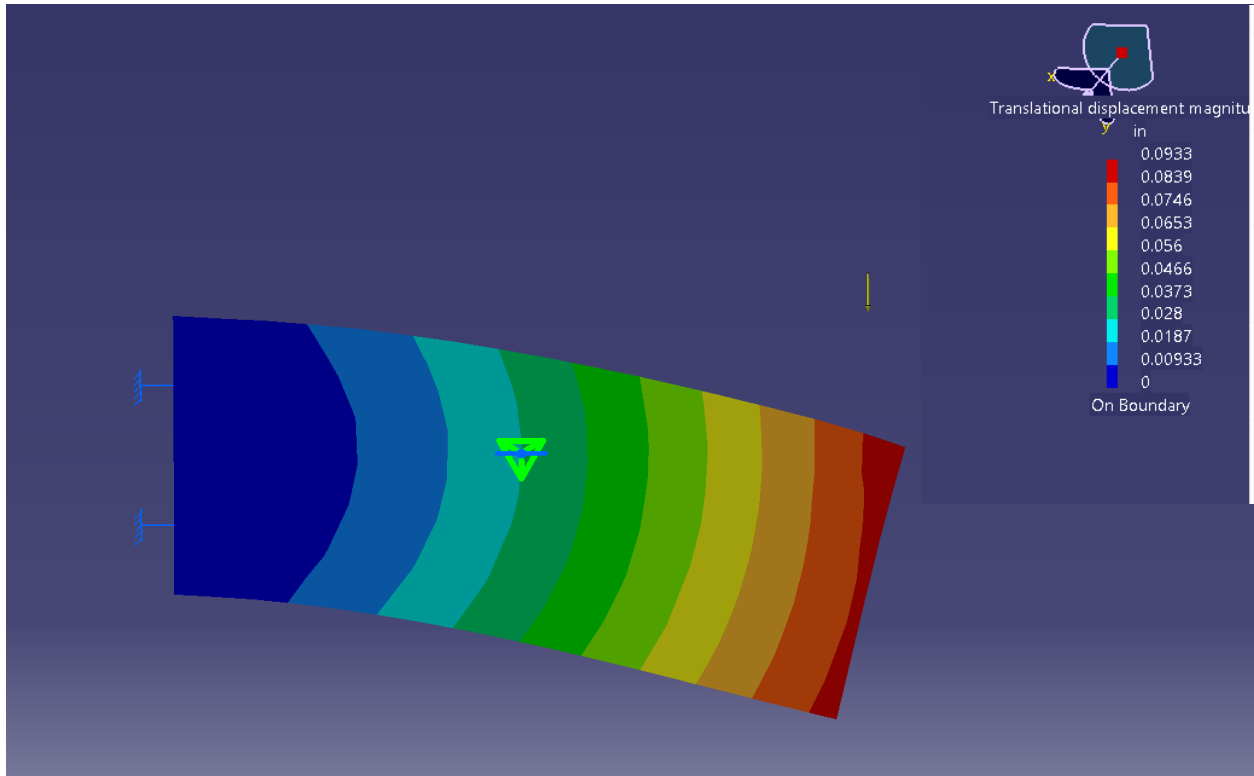


Figure 7: FEA at L=5 inches, $y_{max}=0.0933in$

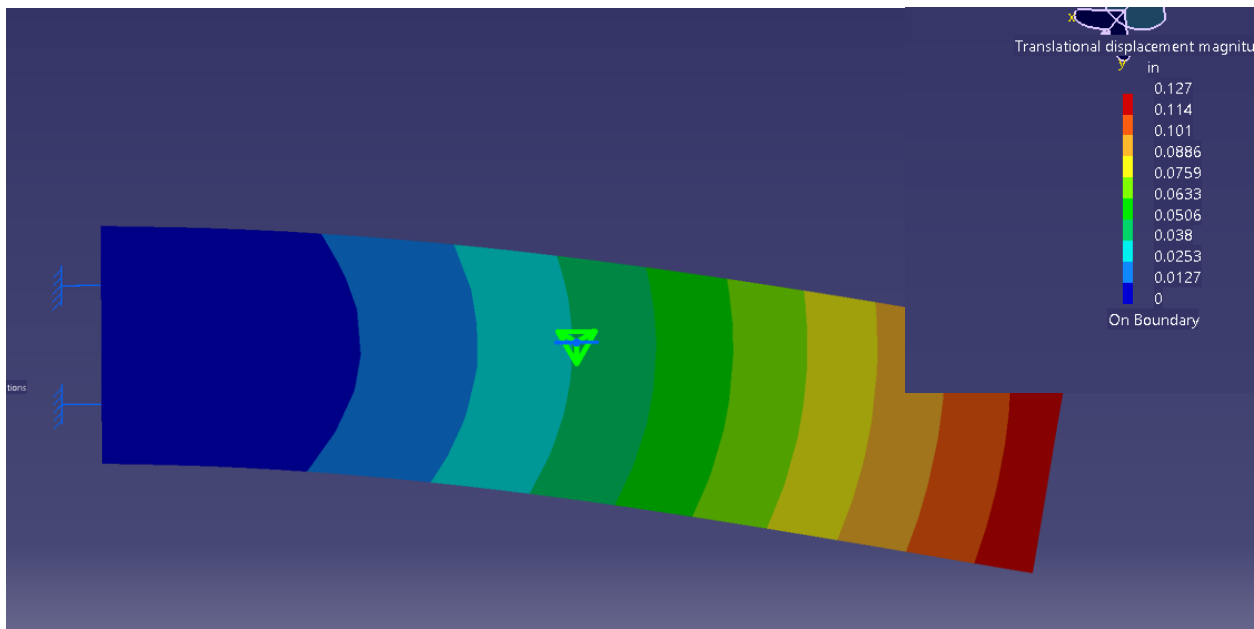


Figure 8: FEA at L=8 inches, $y_{max}=0.127in$

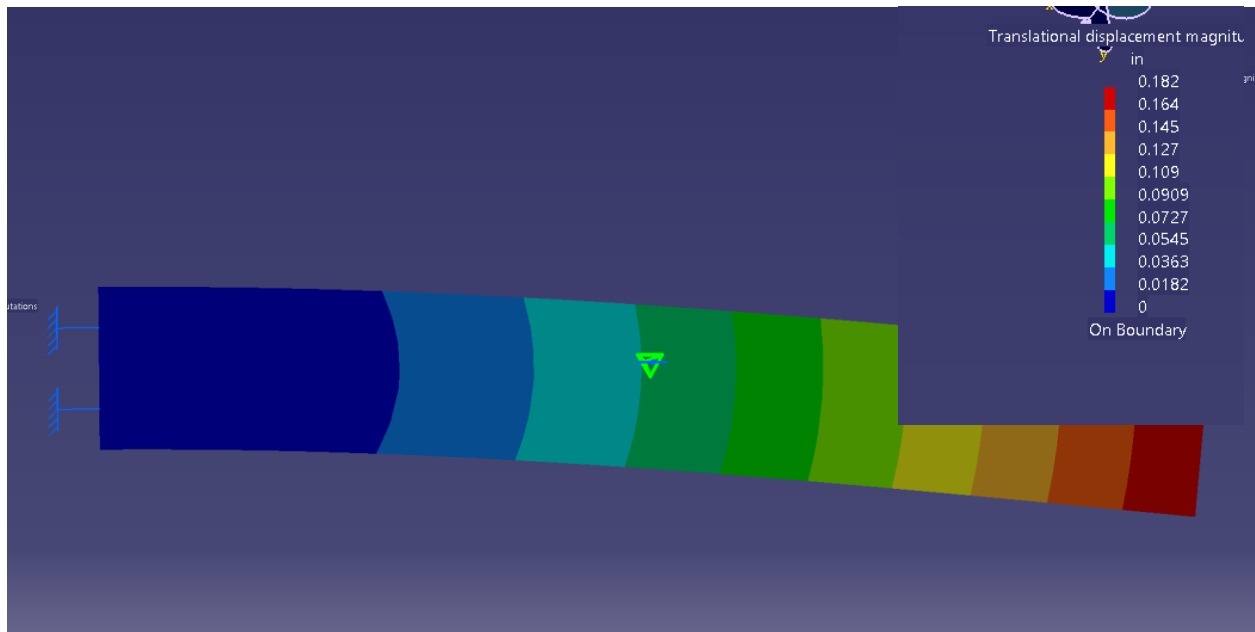


Figure 9: FEA at L=12 inches, $y_{max}=0.182in$

To simulate more realistic beam geometry, we also tested the 12 inch length with a thickness, or “b,” of 3 inches. The model and results of that test are shown below.

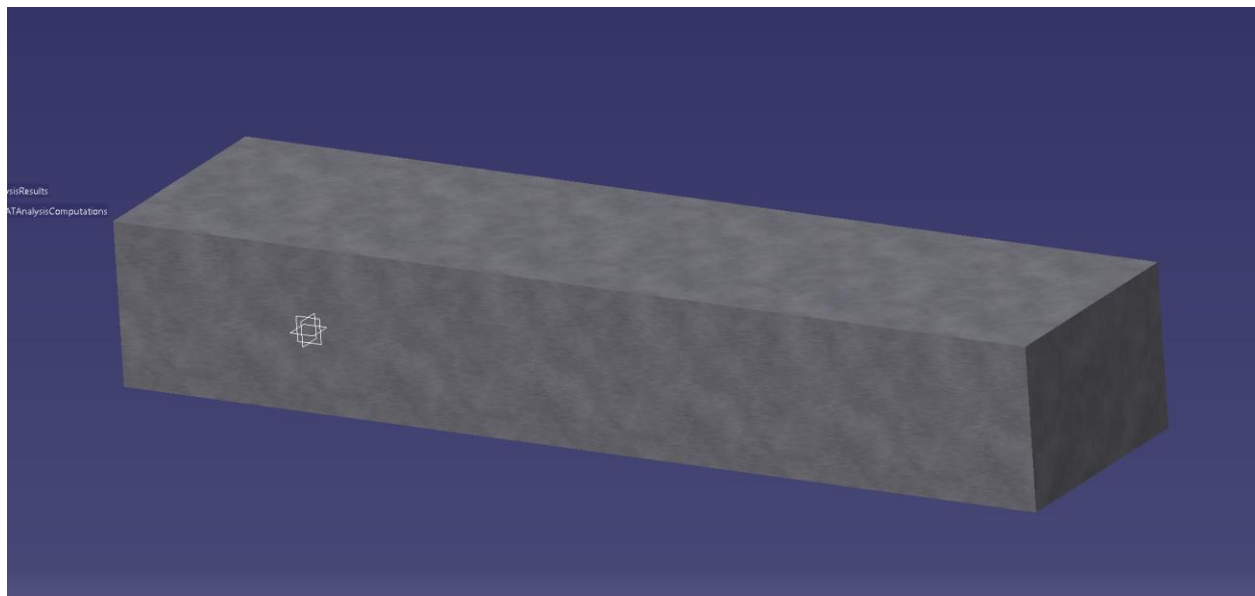


Figure 10: CATIA b=3in, L=12 in model

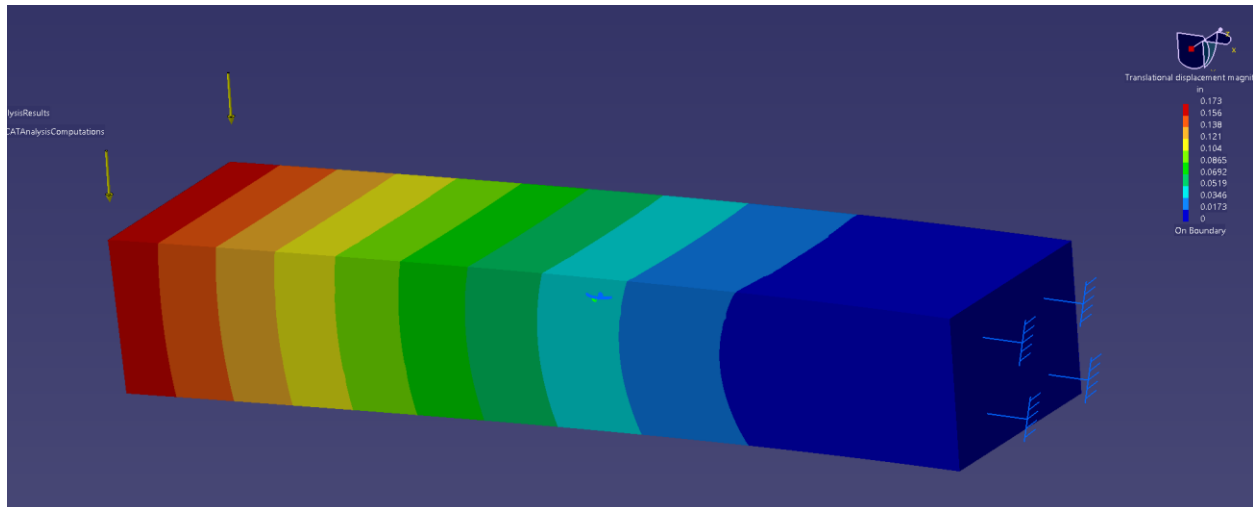


Figure 11: FEA at $b=3\text{in}$, $L=12\text{in}$ results; note the clamp and load are on opposite sides of this capture than on previous captures

While working in CATIA, we determined that it struggles with FEA for this application in a number of areas. First of all, it can be seen from these results that the accuracy of the program in hitting our expected deflection values decreased with increasing length, though this was helped by increasing the density of the mesh. That being said, increasing the density of the mesh hits on another problem in the program. CATIA is not as powerful in FEA, and the processing time required to even generate the mesh, let alone analyze it, is relatively large. Finally, we noticed that CATIA struggles to respect mates that are put into place to simulate composite material by mating multiple parts of different materials in the analysis. We attempted to solve this last issue by implementing a new method, whereby a single part is assigned material properties determined roughly from analysis performed in Abaqus. This dramatically improved results.

Abaqus



Figure 12: Abaqus model used ($L=12\text{in}$ in this image)

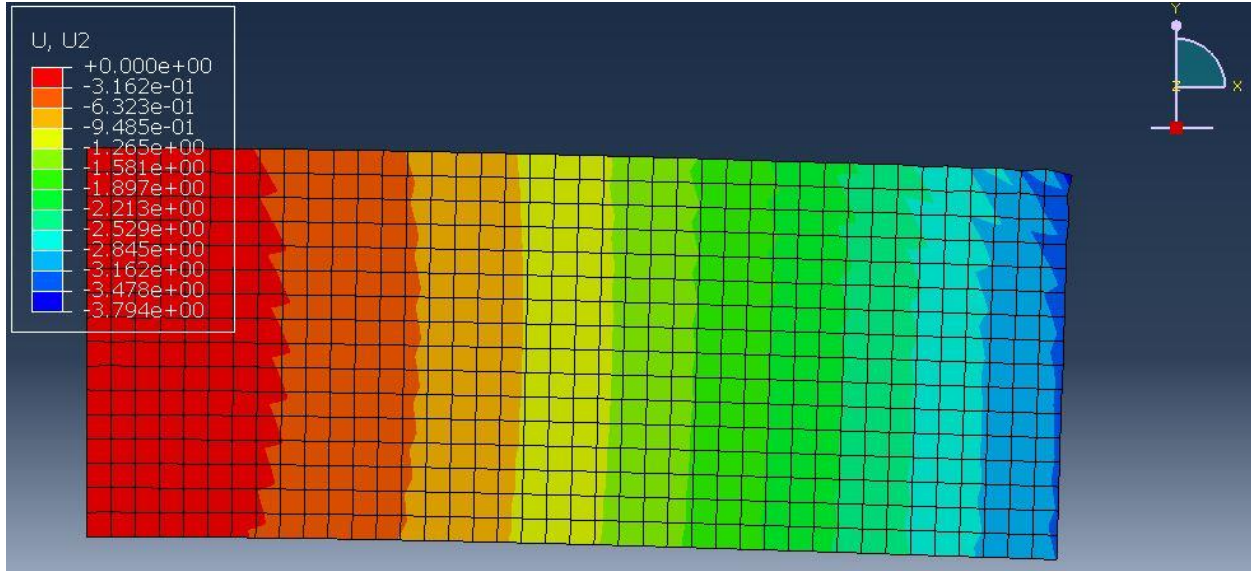


Figure 13: FEA at L=5in results, $y_{max}=3.8\text{mm}$

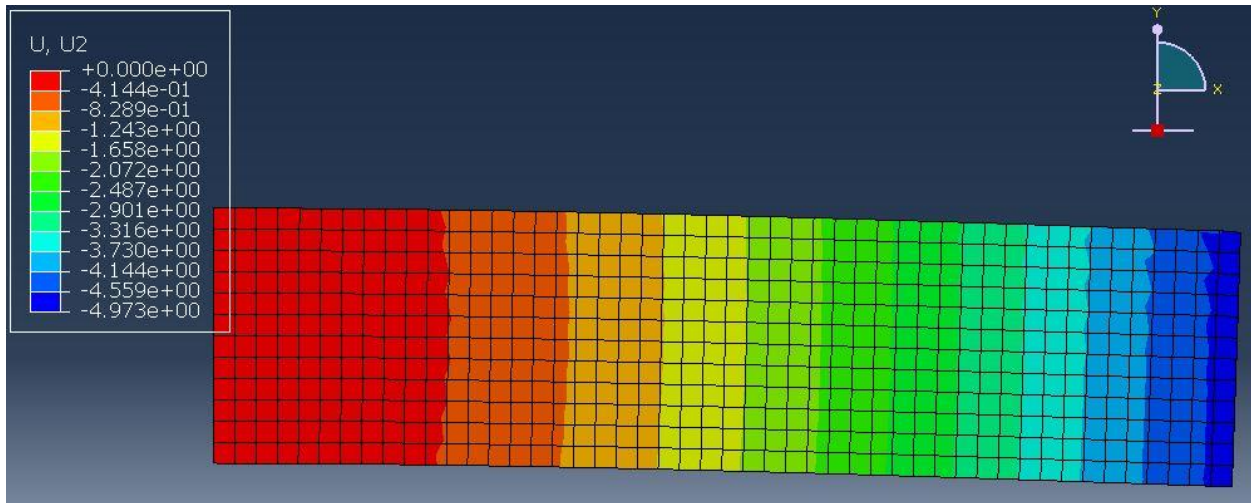


Figure 14: FEA at L=8in results, $y_{max}=4.973\text{mm}$

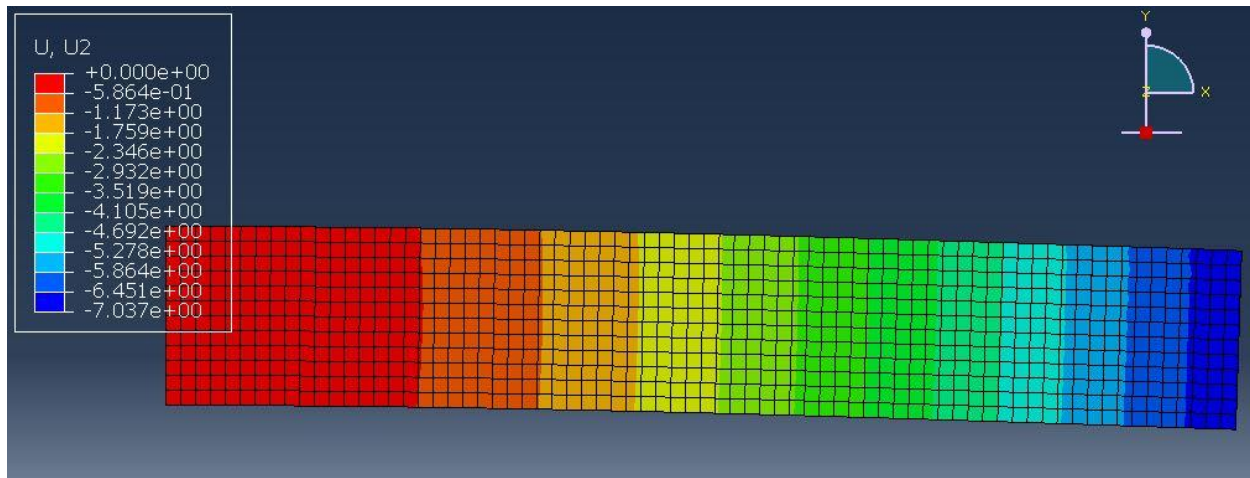


Figure 15: FEA at L=12in results, $y_{\max}=7.037\text{mm}$

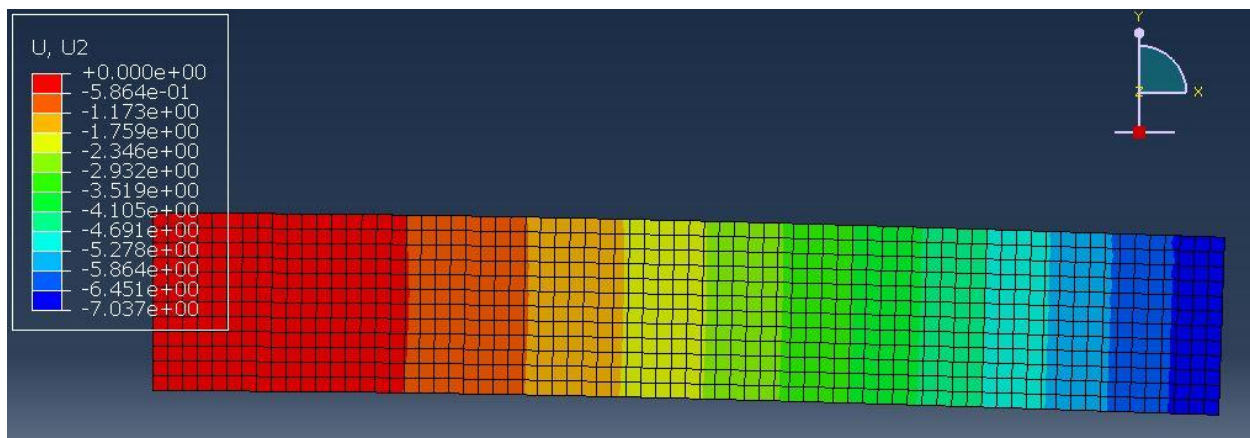


Figure 16: FEA at b=3in, L=12in results, $y_{\max}=7.037\text{mm}$

The U values in the above figures are vertical deflection, measured in mm. It should be noted that the depth for all models except for the final image above were 1 mm in Abaqus as well.

Abaqus produced values for vertical deflection much more consistently close to those we expected, though still not perfect. The primary challenge face in Abaqus is that it is limited to 1000 nodes in the student edition, hampering the capability of the analysis. Additionally, this node limit makes it wiser to work in 2D images when possible, which can impact visualization ability, though it is not a major issue in this case.

Conclusion and Observations

In conclusion, we determined that Abaqus is ultimately the better FEA tool, as it does not face the same issues CATIA does when working with more complex cases such as composite materials. Again, this is likely due to the fact that Abaqus is a dedicated FEA tool, and CATIA is not. Even with the node limitation in Abaqus, it still produced more accurate values to what was expected than CATIA did, and it would be even better without the node limit. While CATIA works a little bit better with a single part using a single

set of material properties, the fact that in this application, finding those material properties was dependent on another FEA program cements our position that Abaqus is the better choice for composite analysis.

Lessons Learned

- Abaqus is better for FEA
- Materials such as plastic should be implemented with great caution in CATIA, as apparently it struggles with such low E values (see the “sausage incident”)
- It is important to look closely at what is happening in mates when running a multi-part analysis in CATIA, to root out unusual and undesired behavior
- Compliant mechanism methods produce similar results to more traditional approaches, and hold up in FEA
- Composite materials, as expected, have E values that land in the middle between those of their constituent materials