PROGRESSIVE DAMAGE AND ORTHOTROPIC ANALYSIS OF ADDITIVE MANUFACTURED COMPOSITE PARTS

Farah, K.[1], Hoque, M.[2], Khoshaba, M.[1], Kolodko, J.[2], Yu, X.[1], Barnett, B.[3*], Sinclair, A.[1*]
[1] Mechanical and Industrial Engineering, University of Toronto, Toronto, Canada
[2] Materials Science and Engineering, University of Toronto, Toronto, Canada
[3]Pratt & Whitney Canada Corporation, Mississauga, Canada

*Corresponding Author (barry.barnett@pwc.ca)
*Corresponding Author (sinclai@mie.utoronto.ca)

Keywords: *composite additive manufacturing, progressive damage analysis, orthotropic analysis*

ABSTRACT

Polymer composites produced by additive manufacturing techniques such as fused deposition modeling (FDM™) do not have an established failure criterion due to the complexities of the material deposition during printing. Material anisotropy is one of the key modeling issues. Current research on failure mechanisms focuses on 2-dimensional (2D) dogbone samples, whereas our focus is on functional 3-dimensional (3D) engineering components. This project employs finite element modeling and mechanical testing of an engineering component produced by additive manufacturing using thermoplastic fiber filled materials.

The project investigated the design of an actuation lever for a gas turbine engine. Using topology optimization techniques, several candidate geometries were generated with an optimized stiffness to weight ratio. Progressive damage and finite element analysis were conducted to assess material selection and candidate design geometries. Finite element simulations enabled evaluation of stress levels and overall lever deformation given anisotropic material properties.

A multilinear hardening model was used to capture both elastic and plastic behavior of the lever after large deformation of the orthotropic material. The Hill yield criterion was used to accommodate the orthotropic material strength properties. The tensile, compressive, and shear stresses were simulated and compared with published values for the corresponding axis/plane to determine any likely failure mode. This investigation incorporated progressive damage analysis in a similar manner to accommodate plastic yielding at locations of high stress. The stiffness of each failed element was reduced to zero in the finite element simulations, and its influence of such failures on the surrounding elements was examined through iterative calculations with increasing load.

Quasi static load testing of the lever on a tensile testing machine was compared to the finite element analysis. The failure locations between analysis and test correlated well; however, the predicted load capability was only approximately 70% of the test result.

1 INTRODUCTION

Traditionally many gas turbine actuation components are manufactured from stainless steel. It is desirable to use lighter weight materials for these components such as polymer composites. There are many conventional polymer manufacturing processes that could be used for these components but the intent of this study is to evaluate the feasibility of using additively manufactured polymers. One of the many challenges of polymer composite additive manufacturing is the anisotropy of the material, specifically when using fused deposition modeling (FDM™), the material is highly anisotropic. This makes it challenging to establish a clear failure criterion and predict the failure mode and mechanism of a component. This can be attributed to the complexities of the material deposition during printing, material anisotropy being one of the modeling issues. Current research on failure mechanisms focuses on 2-dimensional (2D) dogbone samples to follow ASTM standards of testing [1], while this project's focus is on functional 3-dimensional (3D) engineering components. This paper focuses on finite element modeling and mechanical testing of an engineering component produced by additive manufacturing using thermoplastic fiber-filled materials.

2 MATERIALS AND METHOD

An actuation lever used in gas turbine engines was redesigned for additive manufacturing by the University of Toronto PWC2 Capstone team, and is the 3D component being tested in this research. The lever translates linear motion from a pin, which is mounted on an actuating ring, to the vane stem in order to rotate a compressor vane. This project aims to find a material and geometry for that lever, and then numerically simulate its performance; this will then be followed by physical testing to verify the performance.

2.1 Materials and Geometry

The materials selection process utilized the CES EduPack database [2] in combination with market availability research and concluded Antero 840CN03 as the final recommended material. The advantages of Antero 840CN03 are low density, high strength, and capability of working in a high temperature environment with exposure to aviation fluids [3][4][5][6][7]. From the manufacturing perspective, Antero has a reasonable raw material cost, printing time, and purchasing price for the printer [8][9]. Fused deposition modeling (FDM™) is the chosen manufacturing technique because its printers/printed parts are widely available and there is significant research on FDM™ material testing [10].

The lever geometry was designed by optimizing material distribution by utilizing topology optimization simulations in SOLIDWORKS 2020 [11], resulting in the following final geometry, depicted in Figure 1. The rectangular vane stem hole on the left and the circular pinhole on the right were designed to mate with existing engine components.

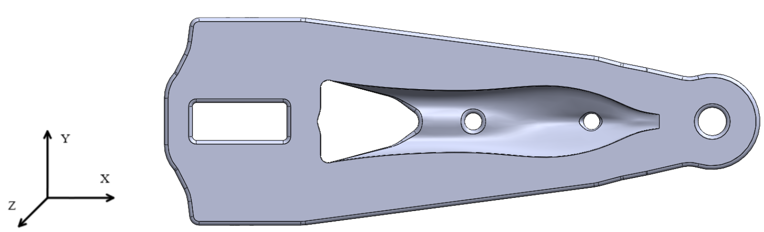

Figure 1. Resulting CAD model of the final lever design.

2.2 Numerical Simulations

Numerical simulations using Antero as the material choice were conducted to predict failure load and locations. The numerical simulations were performed using ANSYS 2021 R2 student version [12]. The material properties were inputted as orthotropic values to simulate the properties of the component when printed on the x-y plane. The inputted values were obtained from published data retrieved from both FDM™ manufacturers and published research papers [4][5][6][7][13]. In addition to the material properties, a multilinear hardening model was used to capture both elastic and plastic behavior of the lever after large deformation of the orthotropic material. The Hill yield criterion was used to accommodate the orthotropic material strength properties.

ANSYS finite element modeling was set up to simulate the scenario that the vane is jammed and refuses to rotate. Therefore, fixed supports were applied at the four faces of the vane stem, implying that the vane stem is rigid and not allowed to translate or rotate. A bearing load was applied at the inner faces of the pin hole in the positive y-direction (Figure 2). The bearing load was chosen since it could distribute the force to the cylindrical surface in the radial direction with a magnitude proportional to the cosine of the angle of the face normal to the load direction [14]. The magnitude of the applied bearing load is 10 times the normal service load.

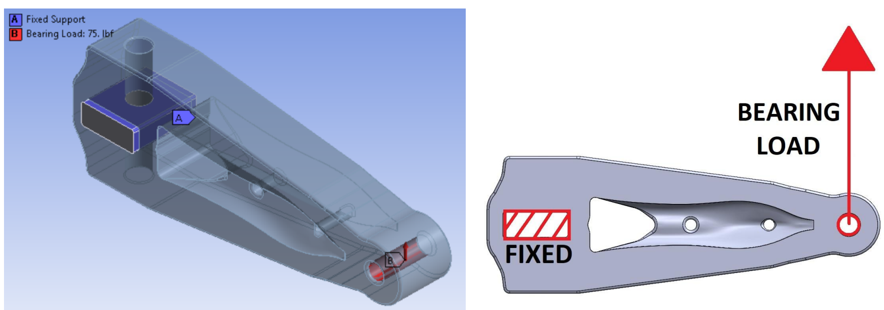

Figure 2. Fixed support and bearing load.

Mesh conditions were set to obtain accurate results while minimizing computational efforts. A global mesh size of 0.02’’ was used. A technique of contact sizing to 0.01" was applied to the contact region between the vane stem hole and the vane stem to simulate their interference more accurately. The contact type between the vane stem hole and the vane stem was set to be frictional, which allows the faces of the vane stem and the vane stem hole to separate and slide past each other with a friction coefficient of 0.25. Figure 3 shows the final mesh result with 48168 mesh elements in total.

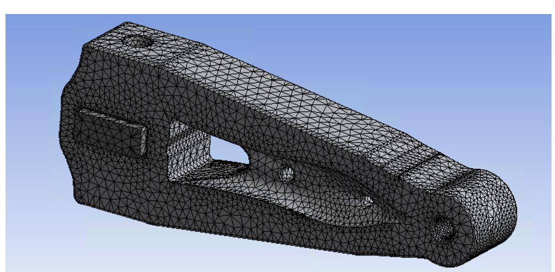

Figure 3. Selected mesh.

Progressive damage analysis was performed to show material performance. Progressive damage analysis is capable of evaluating how damage will progress through the lever as the load is increased [15]. Following the maximum stress damage initiation criteria, an element is assumed to fail once any one of the stress components exceeds the material strength limits [16]. These limits were defined as the orthotropic strength of the material, that is, tensile, compressive, and shear strength in three orthogonal directions or planes. The “instant stiffness reduction” model is chosen as the damage evolution law. Specifically, once the stress reaches the damage limit, the material stiffness will be reduced to a user-specified value to simulate failure [17], [18]. In this case, it is assumed that for both fiber and matrix, the stiffness will instantly be reduced to zero in both tensile and compressive loading modes when the element fails. Through the progressive damage analysis, one could evaluate the damage status of each mesh element, examine the damaged areas and the progression pattern, and determine regions with a high likelihood of failure with an increasing load. The FEA was also used to determine the lever’s deflection behavior at different loads by a Load/Deflection analysis. This was done to predict the maximum amount of deformation the part can endure before failure.

2.3 Physical Testing

The levers were manufactured by Fused Deposition Modeling (FDM™) using Antero 840CN03. The manufacturer of both the Antero material and the levers tested is Stratasys using their Fortus 450 machine [19]. The levers were printed on the X-Y plane with 100% in-fill, 0.01 in (0.25 mm) layer height, and approximately 0.02 in (0.5 mm) layer thickness. The levers were then tested on a 30 kN tensile testing machine with custom-designed fixtures (Figure 4). The test was designed to resemble the in-service loading condition of the lever. The lever was fixed by a rectangular metallic bar that is inserted in a rectangular slot on the lever and held in position by a fixed fixture. The rectangular bar suppresses the movement of the lever along the Y and Z axes and rotations about all three axes, allowing translation only in the x-direction.

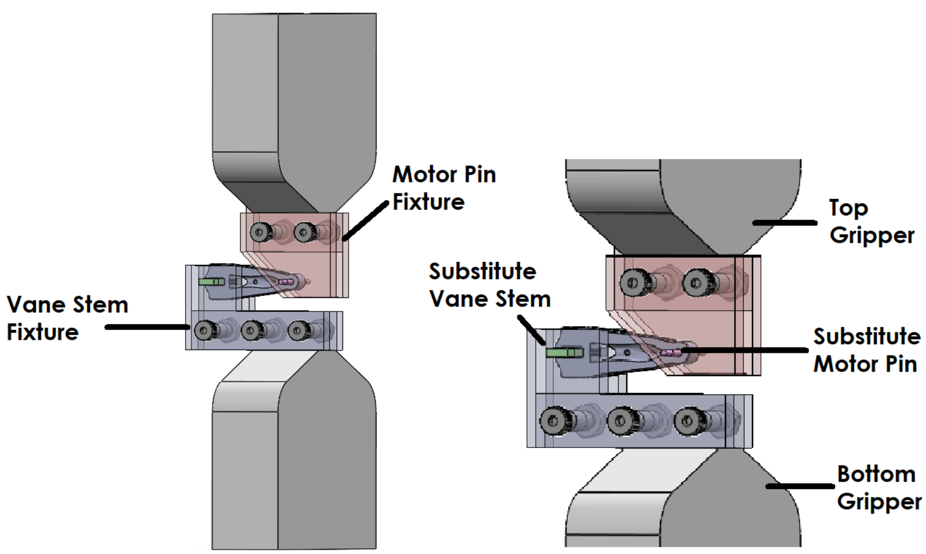

Figure 4. Testing rig assembly.

On the opposite end of the rectangular slot, a circular pin is inserted in a hole on the lever and is pulled upwards simulating a cantilever bending. All tests consisted of the lever being pulled until failure, as shown in Figure 5. The force and displacement results were recorded for each test in order to compare the failure loads and deformations of the physical results to the ANSYS results.

Figure 5. Diagram of testing scenario.

3 RESULTS AND DISCUSSION

3.1 Numerical Simulations

Based on the progressive damage analysis result shown in Figures 6 - 11, it can be seen that significant damage progression occurs between 90lbf and 120lbf. Since numerical simulations are ideal, it is very probable that the parts will fail at lower loads due to imperfections in the testing process or manufacturing defects. Damage is first seen to progress past vane stem corners at 75 lbf where it is increasingly likely that cracks will propagate and cause damage at an earlier load in physical testing. Therefore, it is assumed that parts may fail at loads between 75lbf and 90lbf.

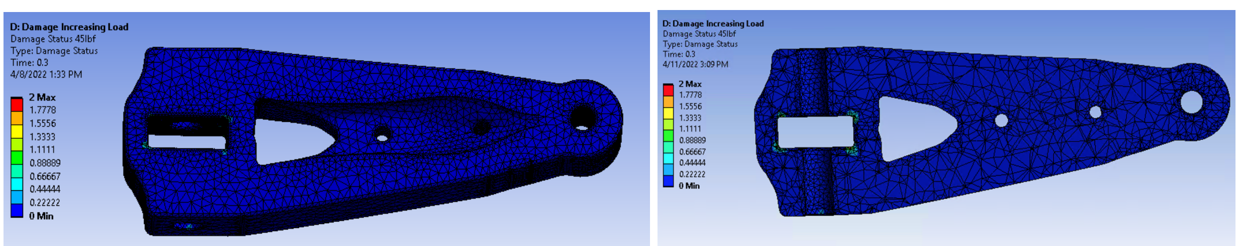

Figure 6. Damage status at 45lbf: full part (left), cross-section (right).

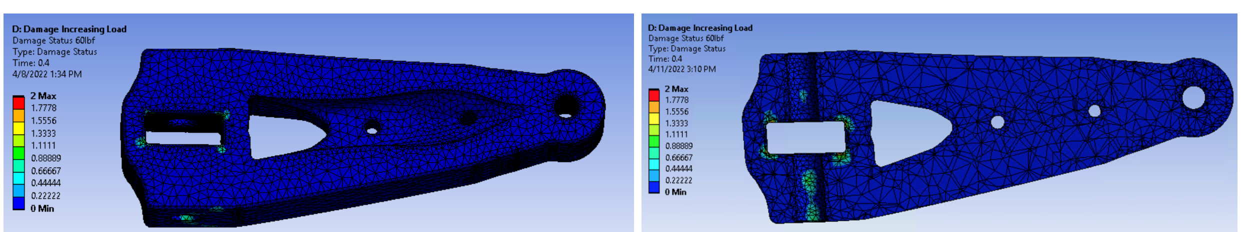

Figure 7. Damage status at 60lbf: full part (left), cross-section (right).

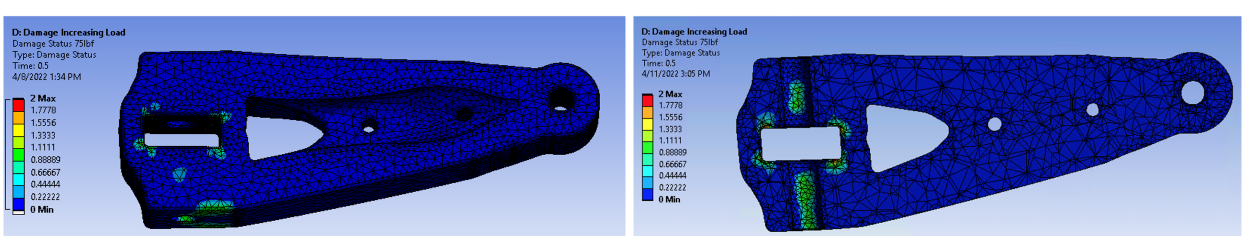

Figure 8. Damage status at 75lbf: full part (left), cross-section (right).

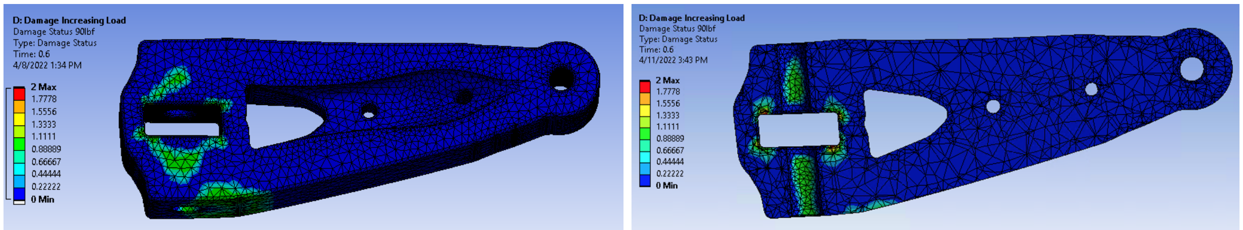

Figure 9. Damage status at 90lbf: full part (left), cross-section (right).

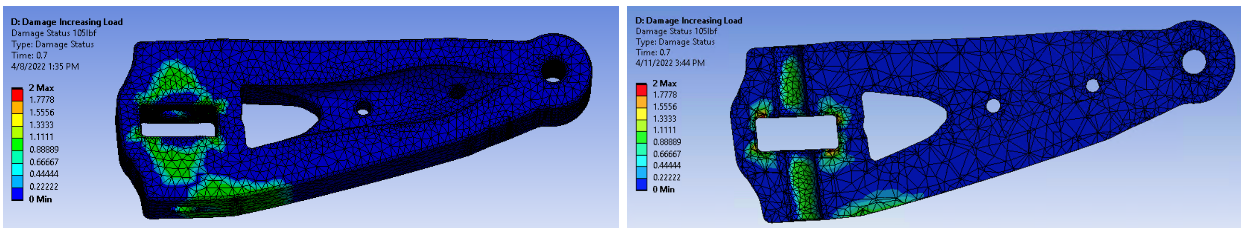

Figure 10. Damage status at 105lbf: full part (left), cross-section (right).

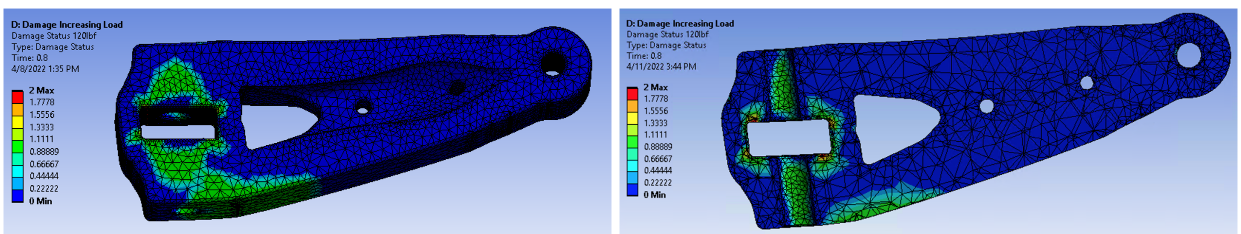

Figure 11. Damage status at 120lbf: full part (left), cross-section (right).

The Load/Deflection analysis of the lever was numerically simulated with loads up to the predicted failure load established by the Progressive Damage analysis. The results of the analysis are represented in the graph below (Figure 12). Note that in the graph, the force has been represented in Newtons (N) as opposed to pound force (lbf). Due to limitations of the version of the simulation software that was used to conduct this analysis, the simulation was completed with loads only up to 70lbf (~300N). Despite this limitation, it can be observed that the lever is capable of significant deformation in relation to its size before failure. Based on the results of the Progressive Damage analysis, it can be predicted that the part can deform up to 8mm before failure.

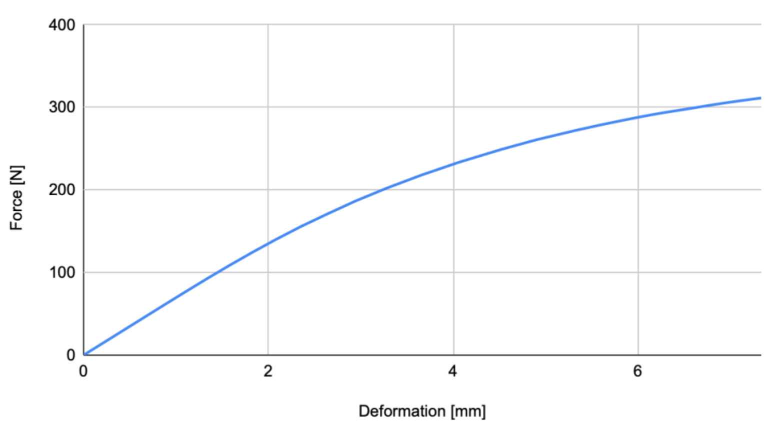

Figure 12. Load/Deflection Curve

3.2 Physical Testing

The physical tests consisted of testing 9 levers made of Antero 840CN03, the proposed material, and printed in the proposed geometry. Figure 13 contains the load-deflection curves for the Antero 840CN03 levers. The results for this test show great variation: four levers failed prematurely before the expected peak, four levers reached a peak and then a relatively constant load for 5-12 mm of additional deformation, and 1 lever failed between those two extremes. The variation in the test results is believed to be due to printing defects and variation in the printing parameters of the levers causing some premature failures. Despite this variation, all levers followed a somewhat similar curve pattern. The levers that were able to reach a peak and maintain a constant load suggest that Antero may have high toughness. This could be confirmed by further physical testing.

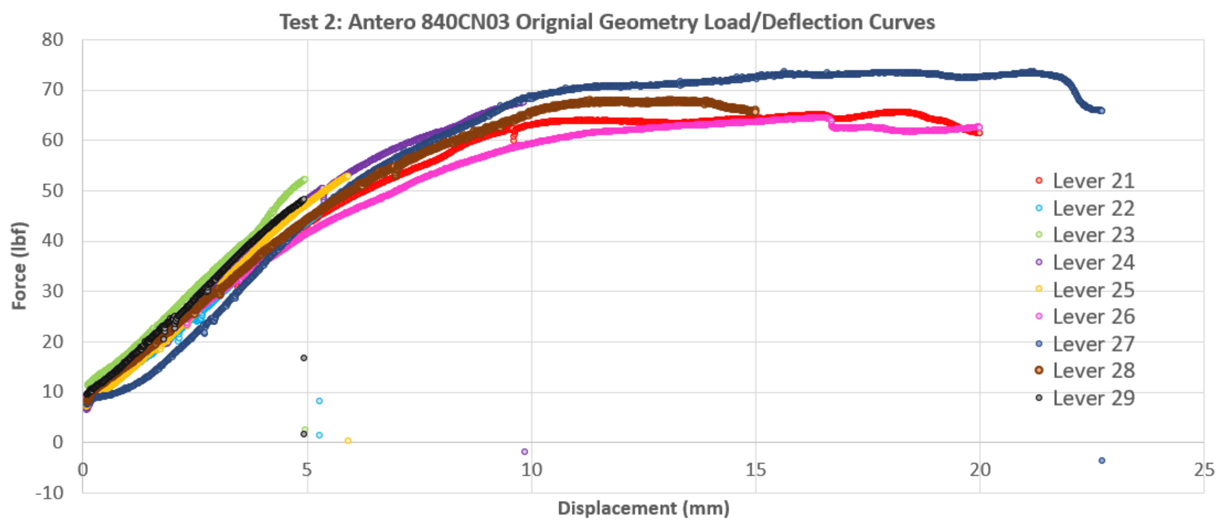

Figure 13. Antero 840CN03 load/deflection curves.

Table 1 contains a summary of the average physical test failure loads compared to the ANSYS predicted failure loads. It is evident that the physical results are about 25 lbf lower than those predicted by ANSYS. This was not unexpected, as ANSYS simulations are idealized solutions that do not take into account the variation and defects found in FDM™ printed parts. It is important to note that the Antero average is lower than expected due to the four levers that failed prematurely; most likely due to printing defects or FDM™ related factors. When looking only at the 5 levers that did not prematurely fail, the average failure load is about 60 lbf. The large variations and inconsistency found in FDM™ printed parts imply that further testing is required to make any solid conclusions.

Table 1. Failure load results summary.

Material	Levers Tested	Physical Test Average Failure Load (lbf)	ANSYS Predicted Failure Load (lbf)
ANTERO 840CN03	9	51.4	75

Table 2 contains a summary of the average deformations in the y-direction at failure compared to the ANSYS predicted deformations at the same failure load. It is evident that the failure load deformations for the physical results featured deformations approximately 1.6 times higher than what was predicted by ANSYS. This observation is not completely unexpected knowing that ANSYS cannot take into account external factors that would increase the deformation of the levers, such as printing defects due to FDM™ 3D printing technology and the play introduced due to the inconsistency in machining the vane stem hole. ANSYS does not take into account the fact that Antero is reinforced with carbon nanotubes, which contribute to ductility [20], and instead looks at the homogeneous material properties. As a result, the deformation predicted by ANSYS is not as high as one would expect, where the reinforcements in Antero would suggest that it could withstand greater deformation. As a result, the Antero specimens were able to maintain a high load for several mm of deflection (Figure 13).

Table 2. Failure deformation results summary.

Material	Levers Tested	Physical Test Average Deformation in Y (mm)	ANSYS Predicted Deformation in Y (mm)
ANTERO 840CN03	9	12.0	7.3

ANSYS simulations predicted the fracture locations on the levers to be the bottom right corner and top left corner of the rectangular slot due to high compressive stresses introduced. These fracture locations were seen in the Antero levers that managed to reach and then maintain a peak load (Figure 14a); however, the levers that failed prematurely featured unexpected fracture locations (Figures 14b-14c). These results further support the conclusion that defects, layer separation, and layer lamination issues may have been present in the manufactured test specimens. It was found that the edge next to the rectangular slot on the levers would fracture prematurely due to the thin dimensioning, causing the 1-2 layers to snap easily. As a result, it is suggested that the edge should be thickened to increase the number of printed layers and avoid this issue.

Figure 14. Antero 840CN03 failure locations.

Figure 15 contains images of the tool path the FDM™ machines would print the levers with. ANSYS is incapable of considering this anisotropic 3D-printed tool path or potential manufacturing defects, and rather models a part as a solid object with orthotropic properties. As a result, the intricacies of the tool path are not factored into the simulation and are simplified.

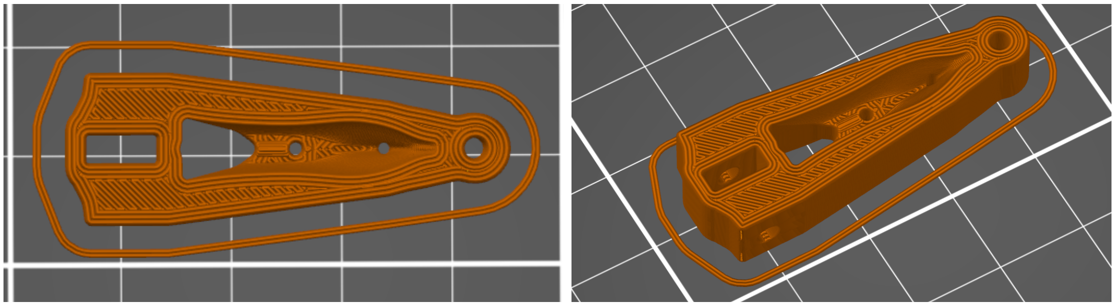

Figure 15: FDM™ tool path for Proposed Lever

4 CONCLUSION

Several conclusions, possible sources of error, and suggestions for next steps were determined when the physical test results and ANSYS results were compared. It was found that the physical testing results vary from the ANSYS simulated results. It is evident that ANSYS predicted better results than what was seen during physical testing, where deformations were predicted to be lower, failure loads to be higher, and stiffness to be higher. It was expected that ANSYS would simulate the levers to have better properties than what was seen in physical testing. In reality, there are additional factors that impact the results that are not taken into account by ANSYS. Furthermore, the simplifications and assumptions used in ANSYS engineering data input could also impact the accuracy of the results.

ANSYS is not capable of capturing the variations of FDM™ printing, such as printing defects or printing parameters. Some or all of the levers could have had printing defects (e.g. voids) that caused the levers to fail at different loads or behave differently when deforming [21]. Because the materials are printed in layers, it is easier for layers to separate or fail if any defects or voids are present [21]. Another important consideration that causes variation in the results is ANSYS's inability to correctly model the FDM™ tool path of the lever, where in ANSYS the lever is modeled as a solid part divided into mesh elements. Finally, variations in the testing results due to the test setup are possible, including variations due to the fixture placement in the tensile machine as well as changes in the machine setup between tests.

To further enhance the validity of the results, it is recommended to gather more information from experimentation, expand the research beyond FDM™, and modify the testing equipment and procedure. For finite element analysis, there was limited information on the material properties. One would suggest printing samples in the specific printing orientation required and then test the mechanical properties to obtain orthotropic results. For example, ASTM D638-14 outlines a standard test method for tensile properties of plastics [22]. Another suggestion is to explore less anisotropic modes of 3D printing such as SLS to lower the variability in physical results. For the physical testing, clear panes of the fixture are recommended to visually see how the lever fails in real time. Also, a variation of force rates and force intervals would show how the damage progresses over time in physical testing compared to FEA. Overall, more specimens and more tests are required to reach definite conclusions.

5 REFERENCES

[1] "Standard test method for tensile properties of plastics". [Online]. Available: https://www.astm.org/d0638-14.html. [Accessed: Apr. 12, 2022]

[2] Granta Design Limited. *CES EduPack software*. Cambridge, UK, 2018.

[3] "Antero 840CN03: A high-performance PEKK-based ESD thermoplastic". *Stratasys*. [Online]. Available: https://www.stratasys.com/materials/search/antero-840cn03. [Accessed: Jan. 05, 2022]

[4] D. Garcia-Gonzalez, M. Rodriguez-Millán, A. Rusinek, and A. Arias. "Investigation of mechanical impact behavior of short carbon-fiber-reinforced PEEK composites". *Compos. Struct.*, Aug. 2015, doi: 10.1016/j.compstruct.2015.08.028. [Online]. Available: https://www.researchgate.net/publication/281068712_Investigation_of_mechanical_impact_behavior_of_short_carbon-fiber-reinforced_PEEK_composites. [Accessed: Jan. 04, 2022]

[5] Stratasys. "Material datasheet for Antero 840CN03". [Online]. Available: https://drive.google.com/file/d/1XwR-3MUKG9cY4QSFDftdKJ9QYzJ1ueyq/view?usp=embed_facebook. [Accessed: Jan. 05, 2022]

[6] B. W. Kaplun, R. Zhou, K. W. Jones, M. L. Dunn, and C. M. Yakacki. "Influence of orientation on mechanical properties for high-performance fused filament fabricated ultem 9085 and electro-statically dissipative polyetherketoneketone". *Additive Manufacturing*, vol. 36, p. 101527, Dec. 2020.

[7] "MatWeb - the online materials information resource". [Online]. Available: https://www.matweb.com/search/DataSheet.aspx?MatGUID=b094b5b798f64ffa85bcad898bfbf57b&ckck=1. [Accessed: Mar. 20, 2022]

[8] Conner Janeteas. Professional Services Manager, Javelin, Jan. 13, 2022.

[9] "Quotation Q128241". Javelin, Jan. 2022.

[10] V. Shanmugam *et al.* "The mechanical testing and performance analysis of polymer-fibre composites prepared through the additive manufacturing". *Polym. Test.*, vol. 93, p. 106925, Jan. 2021.

[11] O. Sigmund and K. Maute. "Topology optimization approaches". *Struct. Multidiscip. Optim.*, 2013 [Online]. Available: https://idp.springer.com/authorize/casa?redirect_uri=https://link.springer.com/article/10.1007/s00158-013-0978-6&casa_token=2CzjsBZ6XiYAAAAA:26JOQdXrXiCaxemEz9MYLyLVYjkhfLIr6Wou2uxv49Ut6pFSwOciLUZpaXa0Er69jWnhILqG_-9EZZQ

[12] "Ansys Student". *Ansys*. [Online]. Available: https://www.ansys.com/academic/students/ansys-student

[13] "Role of infill parameters on the mechanical performance and weight reduction of PEI Ultem processed by FFF". *Mater. Des.*, vol. 193, p. 108810, Aug. 2020.

[14] "Bearing stress simulation". [Online]. Available: https://forum.ansys.com/discussion/616/bearing-stress-simulation. [Accessed: Jan. 05, 2022]

[15] *Damage of Fiber Reinforced Composites | ANSYS e-Learning | CAE Associates*. (Jul. 28, 2014) [Online]. Available: https://www.youtube.com/watch?v=gifMpy3Llso. [Accessed: Jan. 05, 2022]

[16] "Bolted joint behavior of hybrid composites". *Failure Analysis in Biocomposites, Fibre-Reinforced Composites and Hybrid Composites*, Woodhead Publishing, 2019, pp. 79–95.

[17] "MatWeb - The online materials information resource". [Online]. Available: https://www.matweb.com/search/DataSheet.aspx?MatGUID=b094b5b798f64ffa85bcad898bfbf57b&ckck=1. [Accessed: Mar. 20, 2022]

[18] "38.3.2 stiffness and model parameters". [Online]. Available: https://dianafea.com/manuals/d101/Analys/node429.html. [Accessed: Mar. 20, 2022]

[19] "Stratasys - Antero - 2862032-2-quote.pdf". *Google Docs*. [Online]. Available: https://drive.google.com/file/d/1X-bb9QAOkSQXSwifoXk5ffxon1SWtaEq/view?usp=sharing. [Accessed: Jan. 06, 2022]

[20] J. N. Wang, X. G. Luo, T. Wu, and Y. Chen. "High-strength carbon nanotube fibre-like ribbon with high ductility and high electrical conductivity". *Nat. Commun.*, vol. 5, p. 3848, Jun. 2014.

[21] H. Zhao, X. Liu, W. Zhao, G. Wang, and B. Liu. "An Overview of Research on FDM 3D Printing Process of Continuous Fiber Reinforced Composites". *J. Phys. Conf. Ser.*, vol. 1213, no. 5, p. 052037, Jun. 2019.

[22] "Standard Test Method for Tensile Properties of Plastics". [Online]. Available: http://www.dept.aoe.vt.edu/~aborgolt/aoe3054/manual/expt5/D638.38935.pdf. [Accessed: Mar. 26, 2022]