Finite Element Analysis of Additively Manufactured Products

Megan Lobdell, Brian Croop and Hubert Lobo

DatapointLabs, Ithaca, NY, USA

Summary

With the growing interest in 3D printing, there is a desire to accurately simulate the behavior of components made by this process. The layer by layer print process appears to create a morphology that is different from that from conventional manufacturing processes. This can have dramatic impact on the material properties, which in turn, can affect how the material is modeled in simulation. In the first stage of our work, we seek to test an additively manufactured material for mechanical properties and validate its use in ANSYS simulation using the Cornell Bike Crank model.

Keywords

additive manufacturing, mechanical properties, material model, validation, ANSYS, 3D printing

1. Introduction

In additive manufacturing (commonly referred to as 3D printing), direct metal laser sintering (DMLS) is one of the leading manufacturing processes for metal part production. It is considered a cost effective and environmentally friendly alternative to conventional manufacturing processes for smaller production runs because it does not require part-specific tooling or molding. It produces less waste than subtractive manufacturing processes. It can also be used to produce currently unmanufacturable geometries such as internal chambers for weight reduction in the part.

DMLS uses a laser to fuse metal powder layer by layer into a solid part. The geometry for each layer is extracted as a slice from a 3D CAD drawing. Before production, simulation can be used to confirm the validity of these novel geometric designs to see if they meet the performance requirements. Material data for such simulations is often based on properties of an equivalent, but traditionally manufactured, material. It is suspected that the additive manufacturing process may produce product with different mechanical properties; concerns also exist about density variation and void formation, all of which could impact simulation accuracy as well as real world performance. Quantifying these effects would greatly increase the confidence of engineers and designers to use this compelling new technology.

This paper probes the differences of between measured material properties compared to these published values. The data are used to evaluate their effect on an ANSYS simulation at small strains through slightly deforming a modified Cornell bike crank and comparing results from digital image correlation (DIC) software to ANSYS.

The Cornell bike crank experiment [1] is part of a university lab course that teaches the importance of validation in finite element analysis. A bicycle-like crank geometry is subjected to a controlled static load case similar to the kind of force it might experience from a person using a bicycle. One end of the crank is rigidly fixed while the other end is subject to a static rotational force. Four crank geometries exist, each created to highlight stress states of varying complexity. The cranks are particularly designed to enable the calculation of deformation using classical analytical methods and can also be easily reproduced in ANSYS simulations. Experimental strains are measured using strain gauges placed at specific locations of the crank. Measured strains are compared to calculated and simulation values to provide a measure of simulation accuracy. The original classroom experiment was found to contain sources of experimental error which made it difficult to provide a reliable basis for validation. Borschoff et al [2] eliminated strain gauges from the experiment by switching to digital image correlation (DIC) methods, significantly improving the correlation accuracy to ANSYS. In their work, they showed that ANSYS correlated to real life within 10% for linear elastic simulations of metals, which is similar to the accuracy of ANSYS compared to hand calculations for this geometry.

2. Procedure

In the present work, one of the Cornell cranks was modified to enhance stress states in particular regions of the part. ANSYS was used to tune the geometry for this purpose. The fixed end was also modified to make it more rigid (Figure 1).

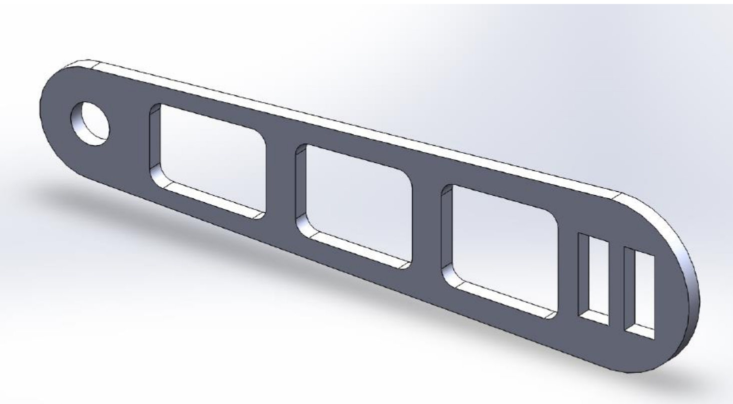

Figure 1 Modified Cornell crank used in present work

Three cranks were printed side-by-side in a single XY plane along with six tensile bars, three in the x direction and three in the y direction. The printing was performed by Incodema 3D, Inc. Ithaca, NY through direct metal laser sintering of EOS Aluminum AlSi10Mg gas atomized powder from EOS GmbH with a chemical powder composition as provided below (Table 1). A 370 W, 100-500 µm variable diameter Yb fiber-laser on a EOS M280 was used to sinter the 30µm layers. The laser traversed the pattern at a 1300mm/s scanning speed with a hatching distance of 0.19 mm. For each layer the scanning path was rotated 66°. After cooling, specimens were cut away from the bed.

Table 1 Chemical Composition of Powder

Element	Si	Mg	Fe	Mn	Cu	Zn	Ti	Al
Measured Concentration (%)	9.7	0.35	0.19	0.07	<0.001	<0.002	0.004	Balance

To generate the material properties needed for the material model, tensile properties and density were measured. Tensile tests were performed on an Instron 8872 servo-hydraulic testing machine. The tests were performed in conformance with ASTM E8 [3] at a crosshead speed of 5mm/min. Data were taken in the 0° and 90° orientations using tensile bars created as explained above. 51.5 mm long dogbone specimens with gauge lengths of 20 mm, widths of 4 mm, and 5 mm thickness were used. Force was measured from a load cell on the Instron while displacements on the test specimens were measured using DIC (Figure 2) which were then converted to strain using ARAMIS software.

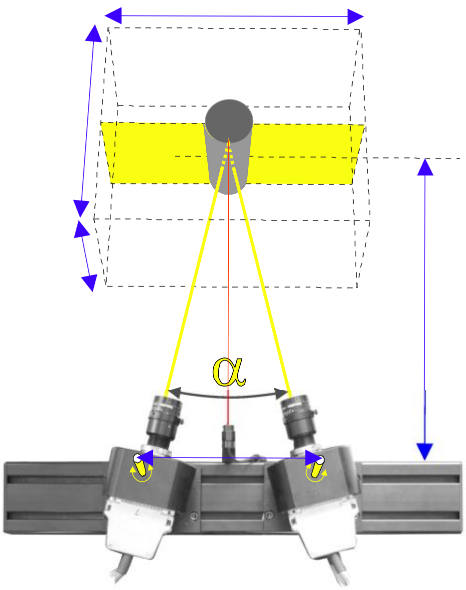

Figure 2 Digital Image Correlation schematic diagram

For the elastic validation experiment, the crank was prepared for DIC by painting it with a spray pattern. It was rigidly held at one end (right) using a fixed key through the rectangular holes while the other end with a circular hole (left) was pulled upward via a cylindrical pin attached to the actuator of the load frame as shown in Figure 3. With the rectangular end fixed, the actuator was moved to apply a force of 650N in the y direction, initially perpendicular to the crank. The pin in the hole was allowed to freely rotate and move in the x direction. The test itself was performed at a speed of 20mm/min up to 650N force.

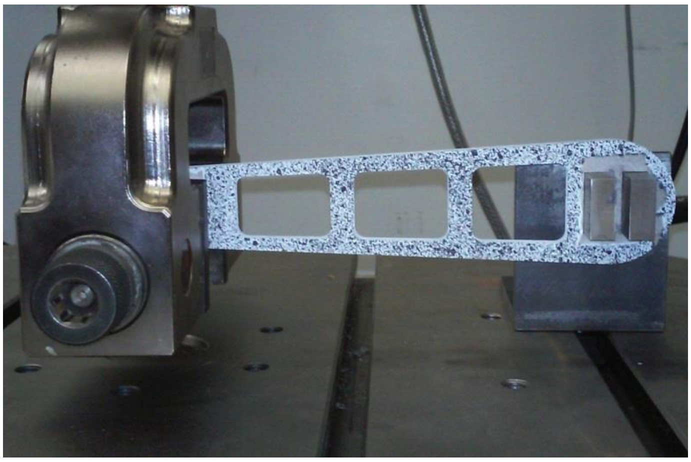

Figure 3 Experimental set-up

Two simulations were used to probe both the elastic and plastic material models. Material properties were based on either published or measured values. An element size of 0.5mm was selected after performing a mesh senativity test. The first simulation a simple elastic material model was used. The simulation was conducted within the expected elastic region at small strains. The elastic model was tested using two sets of material properties, published and measured values.

One crank was tested to failure and the load to fail and the failure locations were noted. For simulations to the point of failure, a multilinear isotropic hardening (MISO) model and a bilinear isotropic hardening material model were used. The material properties for these simulations came from measured data.

3. Results

3.a Material model

Material testing between the two directions finds little difference in modulus (65.9 GPa at 0° versus 65.6 GPa at 90°). However, there is signifcant difference of tensile strength and failure strains depending on direction as noted in Table 2. In other papers using AlSiMg materials slight differences in the material properties based on the orientation have been reported, both in-plane with and in the build (z) direction [4].

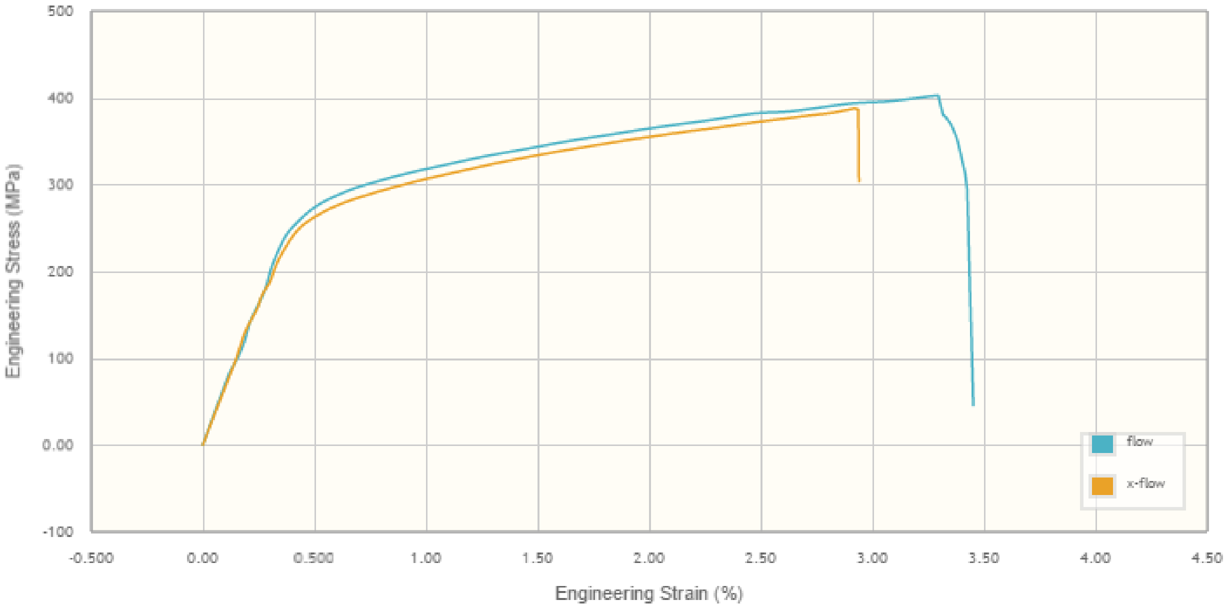

Figure 4 Representative curves for the 0° and 90° directions

For equivalent materials, both A360 [5] and A380 [6] were recommended as having similar chemical compositions; data from the literature [7,8,9] for these materials is presented in Table 2. Comparing these we note that we measured a lower modulus and density than literature. Further, the measured yield stress was significantly higher than literature reported values with the A360-T6 being the closest to our measurements. EOS, the metal powder supplier notes that: "Conventionally cast components in this type of aluminium alloy are often heat treated to improve the mechanical properties, for example using the T6 cycle of solution annealing, quenching and age hardening. The laser-sintering process is characterized by extremely rapid melting and re-solidification. This produces a metallurgy and corresponding mechanical properties in the as-built condition which is similar to T6 heat-treated cast parts."[10]

Table 2 Material properties from tensile experiments and the literature

Direction	Modulus (E) (GPa)	Offset Yield Stress (MPa)	Tensile Strength (MPa)	Strain at Fail (%)	Density (g/cm^3)
EOS AlSi10Mg 0°	65.9±3.7	292±6.3	405±13.2	3.37±0.06	2.63
EOS AlSi10Mg 90°	65.6±4.0	268±11	371±15	2.77±0.15	2.63
EOS AlSi10Mg Published[10]	70±5	275±10	445±20	3.37±0.06	2.67
A360-O [7]	71	165	317	3.5	2.68
A380-O [8]	71	159	324	3.5	2.76
A360-T6 [9]	71	250	310	4	2.63

3.b Crank Simulation in the Elastic Region

Simulations were run with two material models, one with the modulus of 71GPa from published values of equivalent A360-O and A380-O conventionally made cast aluminum and the other with the in-house generated modulus data of 65.6 GPa. The crank simulation demonstrated good correlation between normal x-strains (along the length of the crank) as demonstrated in the images of the x-strain contours for elastic normal strains. All images are scaled for the same strain range so that it is possible to quantitatively compare images based on color.

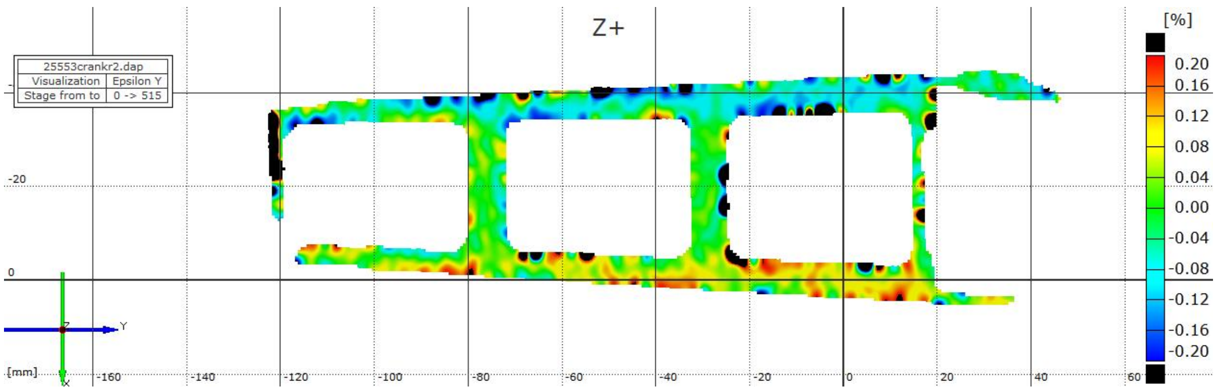

Figure 5 Strain contour along the length of the crank

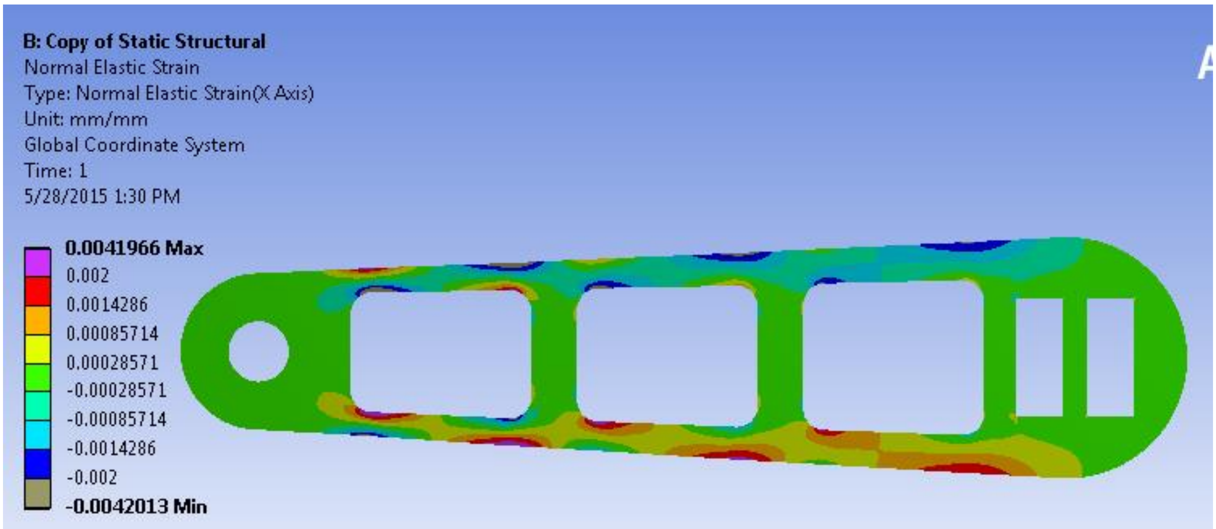

Figure 6 ANSYS x-strain contour with measured modulus

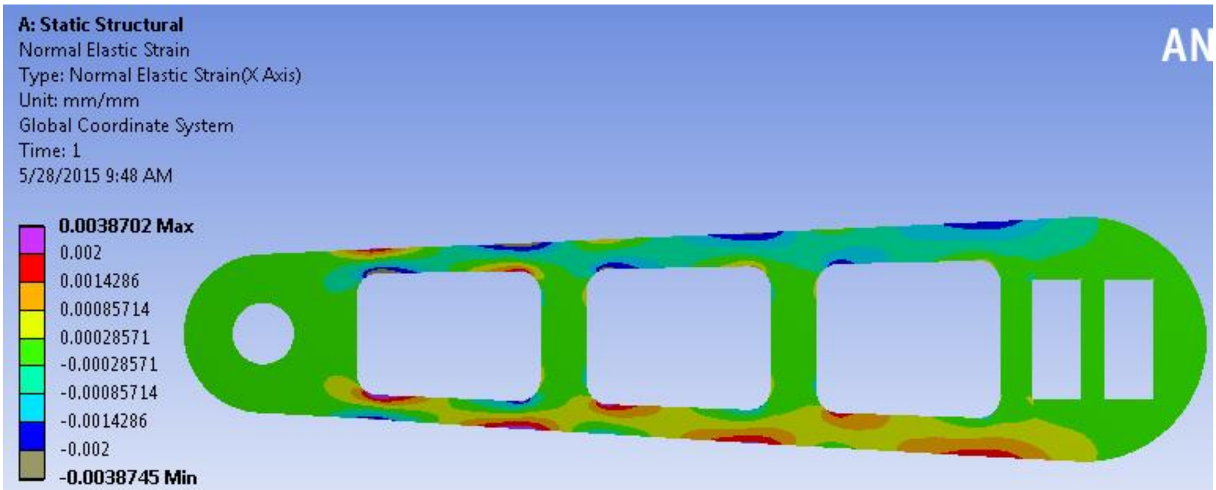

Figure 7 ANSYS x-strain contour with published modulus

3.c Crank Failure Simulation

A large deformation simulation was carried out by applying a Multilinear Isotropic Hardening (MISO) material model. Matereality's CAE Modeler for ANSYS was used to build the model from the worse case 90° orientation stress-strain data (Figure 8).

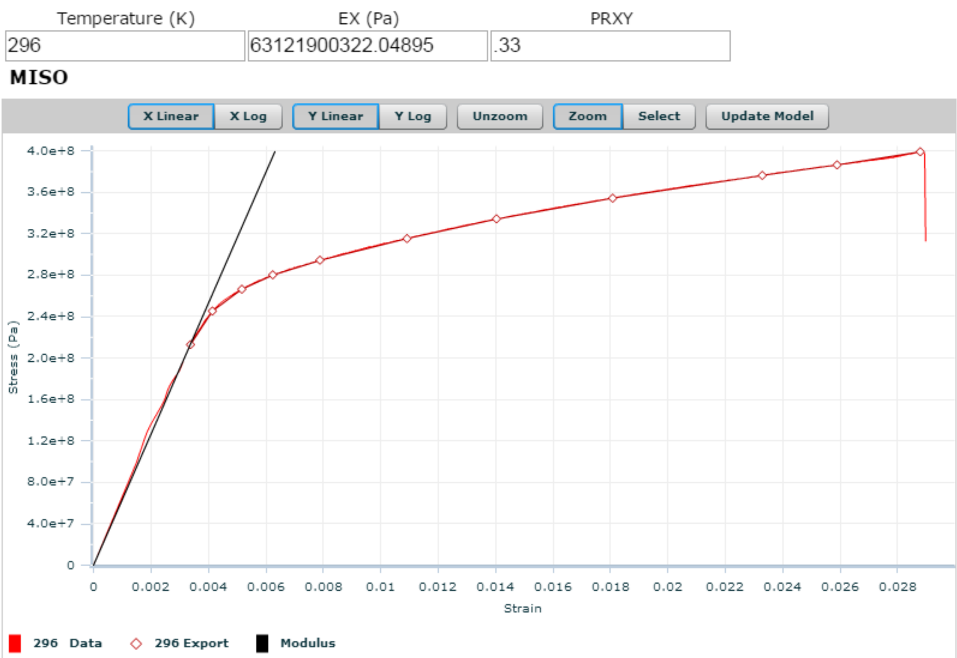

Figure 8 90° orientation MISO Model created using Matereality CAE Modeler for ANSYS

During the experiment, the crank sustained a load of 1800 N before failing (Figure 9).

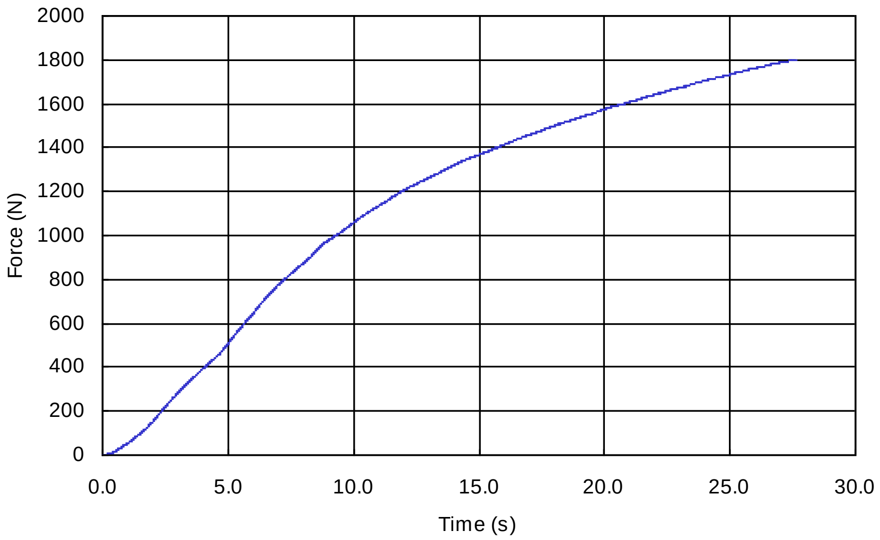

Figure 9 Experimental force v time data for the crank until failure

Brittle failure occurred at the bottom left corner (A) of the crank (Figure 10) following which a subsequent failure occurred in the top right corner (B) as the moving part of the crank continued to break away from the fixed end.

Figure 10 Image of Broken Crank

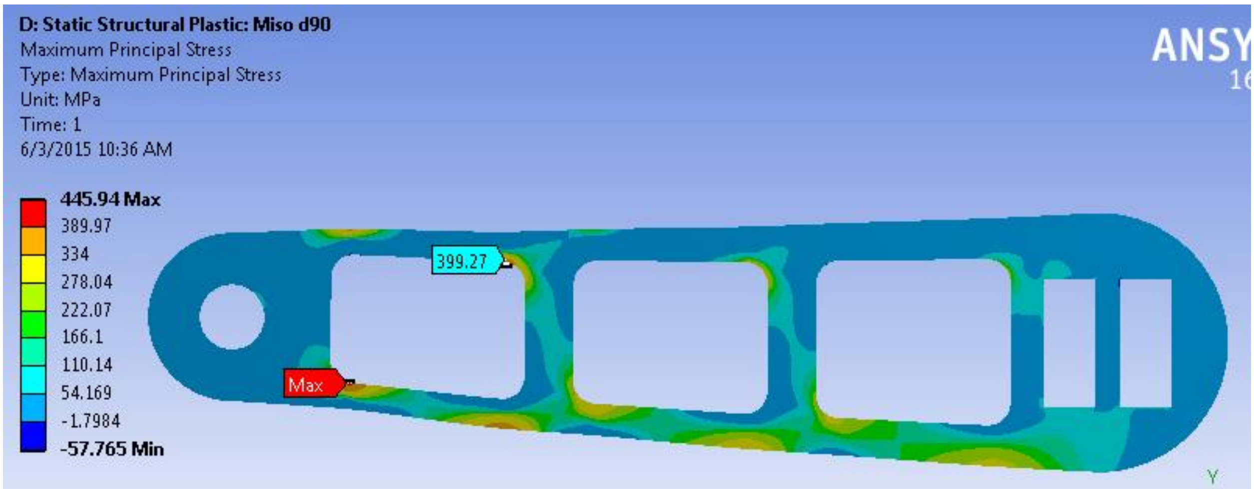

Figure 11 ANSYS showing the maximum principal stress

For the non-linear simulation, the crank was loaded to 1800N which was the measured failure load of the physical experiment. The simulation reported two high-stress locations which coincided with those observed in the experiment: 446 MPa for location A and 399 MPa for location B.

For the prediction of normal failure in brittle materials, we used the Mohr-Coulomb Theory, which anticipates failure at the location where the maximum principal stress exceeds ultimate tensile strength [11]. Following this theory, we note that the ANSYS correctly predicts the failure stresses and locations for this 3D printed material. The representative curve had an ultimate stress at 400 MPa (Figure 8). The difference between predicted maximum principal stress and measured ultimate tensile strength is of the order of 12%. Considering the anisotropy of the crank in the XY plane coupled with the variability of the experimental stress-strain data itself, the error is well within the acceptable bounds. Further the progression of the failure is also correct in that location A being in a higher stress state, fails before location B in the real life experiment.

4. Conclusions and Future Work

ANSYS accurately predicts strain contours of DMLS parts for small strains in the elastic region. These findings may be considered relevant for printed parts that are primarily flat in the x-y plane. Although there is evidence of anisotropy of yield and failure between the 0° and 90° directions, these phenomena are not relevant for small strain modeling.

When simulation progresses to failure ANSYS is able to predict the failure locations, failure stress as well as the progression of failure with good correlation to experiment for testing in the x-y plane. It is believed that greater anisotropy exists in the z plane, perpendicular to the the printing plane. In future work, we will probe the effect of anisotropy on the modeling of 3D printed parts.

5. Acknowledgements

We are grateful to Mr. Kevin Engel and others at Incodema3D for advice on 3D printing and for preparing the test samples, to Mr. Brian Hampton and Mr. Philip Seaton at Fiat Chrysler Automobiles for comments on equivalent and advice on equivalent, conventionally manufactured materials.

6. References

[1] SimCafe - https://confluence.cornell.edu/pages/viewpage.action?pageId=248382509

[2] Borshoff.J, Roy D., Croop B., "The Use of Digital Image Correlation (DIC) and Strain Gauges to Validate Simulation.". NAFEMS USA Regional Conference. (2014).

[3] ASTM E8 „Standard Test Methods for Tension Testing of Metallic Materials", (2015)

[4] Manfredi, Diego, et al. "From powders to dense metal parts: Characterization of a commercial AlSiMg alloy processed through direct metal laser sintering."*Materials* 6.3 856-869 (2013)

[5] Engel K., Incodema3D, Inc, private communication (2015)

[6] Hampton B., Fiat Chrysler Automobiles, private communication (2015)

[7] "Aluminum A360.0-F," http://www.matweb.com/search/datasheet.aspx?matguid=87a0d0817ebd44008a967cbf3e9cd378 (2015)

[8] "Aluminum A380.0-F," http://www.matweb.com/search/datasheet.aspx?matguid=5f92a8f7d6ad416c8ce9398cae14a363 (2015)

[9] "Aluminum A360.0-T6," http://www.makeitfrom.com/material-properties/A360.0-T6-Cast-Aluminum/ (2015)

[10] "Material Data Sheet: EOS Aluminum AlSi for EOSINT M 270," https://www.anubis3d.com/documents/dmls/datasheets/Aluminum-AlSi10Mg.pdf (2015)

[11] Chen, Xiaolin, and Yijun Liu. "Finite Element Modeling and Simulation with ANSYS Workbench." CRC Press, 362-63, (2014)