Material Analysis with an Elasto-Plastic FEA Model

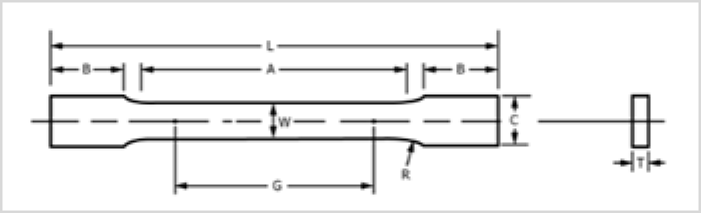
Brett Cavanaugh, Joshua Corbett, Isabel Dunn, and Lucy Zimmerman

Introduction

The objective of our project was to enhance our understanding of material properties by generating and utilizing stress-strain curves. These curves were obtained through testing specimens and imported into SolidWorks for simulation analysis. By evaluating and comparing the performance of different material models—linear and elasto-plastic—we aimed to understand their impact on simulation results.

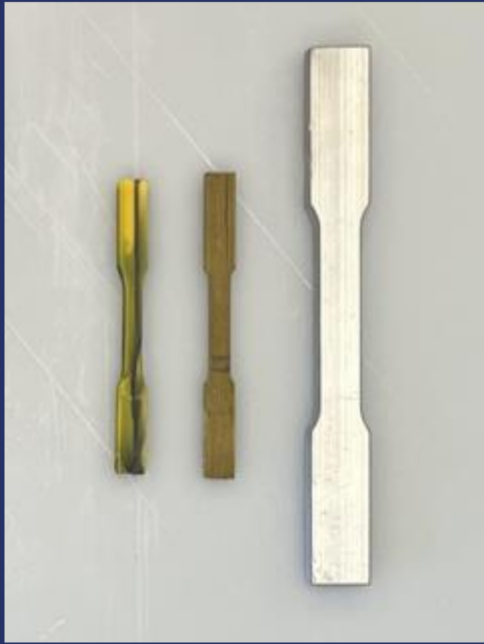
To achieve this, we conducted material tests using the Instron machine with dog bone test specimens, extracting key mechanical properties. Using a cantilever beam as a case study, we analyzed the performance of each material model across simulations, highlighting the differences in behavior and accuracy. This approach allowed us to explore the practical applications and limitations of these models in engineering design.

Dog Bone Tensile Test Specimens

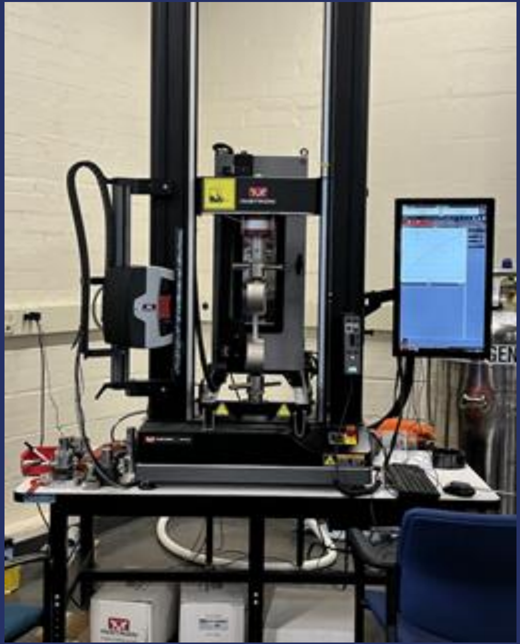


Dimensions (in)	Brass 360	Aluminum 6061	Polypropylene
G - Gauge Length	1.000 +/- 0.003	2.000 +/- 0.005	2.000 +/- 0.010
W - Narrow Width	0.250 +/- 0.005	0.500 +/- 0.010	0.500 +/- 0.020
T - Thickness	0.150	0.250	0.130 +/- 0.020
R - Radius	0.250	0.500	3.000 +/- 0.040
L - Overall Length	4.000	8.000	6.500 +/- 0.020
A - Reduced Parallel Length	1.250	2.250	N/A
B - Grip Section Length	1.250	2.000	N/A
C - Grip Section Width	0.375	0.750	N/A
D - Distance Between Grips	N/A	N/A	4.500 +/- 0.020
WO - Overall Width	N/A	N/A	0.750 +/- 0.250

Tensile Testing



Before



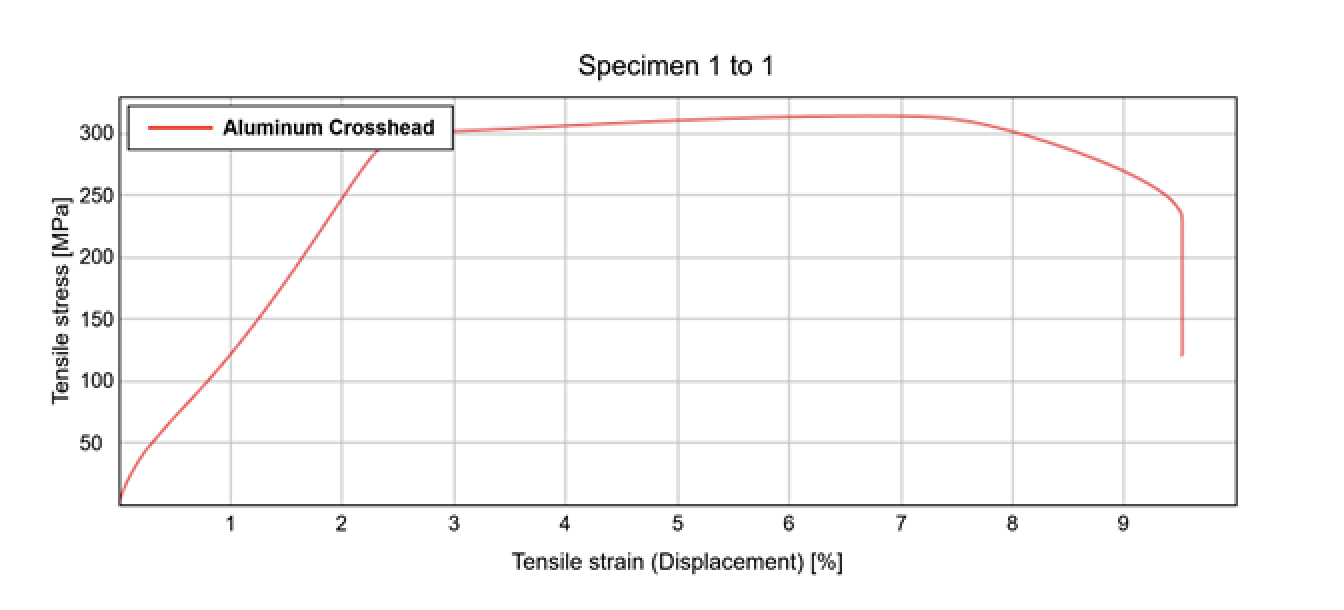
During

After

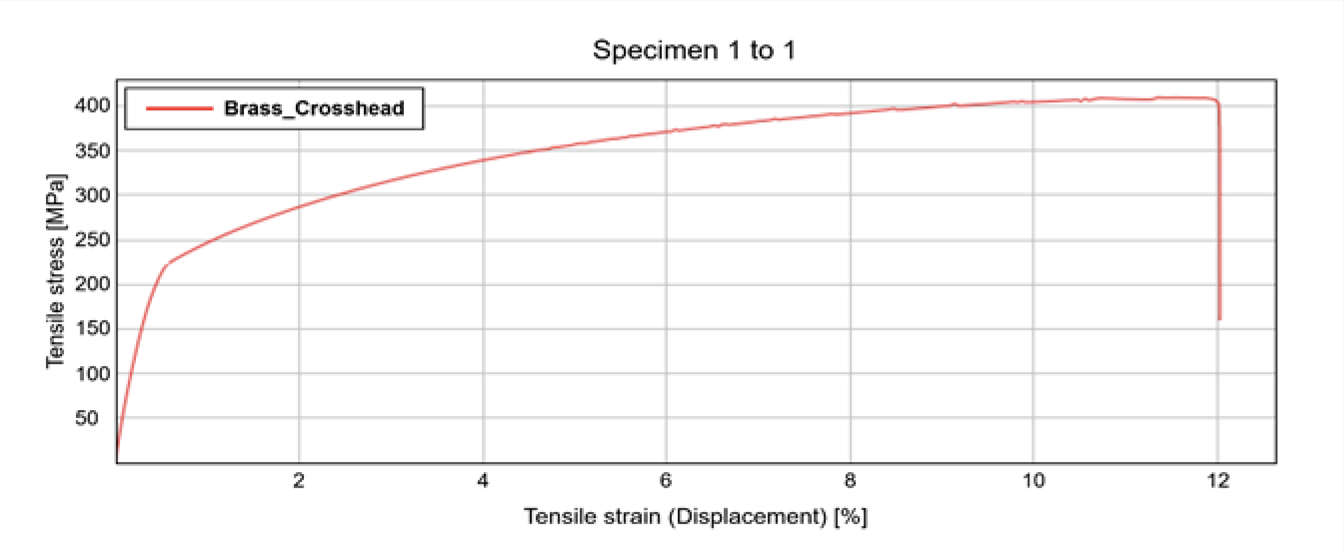
Instron Testing Procedure

1. The Instron was turned on.
    1.1. The test was set up with external testing standards in mind.
    1.2. A static extensometer was used for the Brass 360 specimen to allow for proper separation.
2. The test specimen was loaded into the machine.
3. The crosshead speed was set.
    3.1. Metals: 2.25 mm/min
    3.2. Plastics: 5 mm/min
4. Young's modulus and the stress versus strain curve were designated for data collection.
5. A tensile force was applied to the test specimen until it broke.
6. The data was exported.

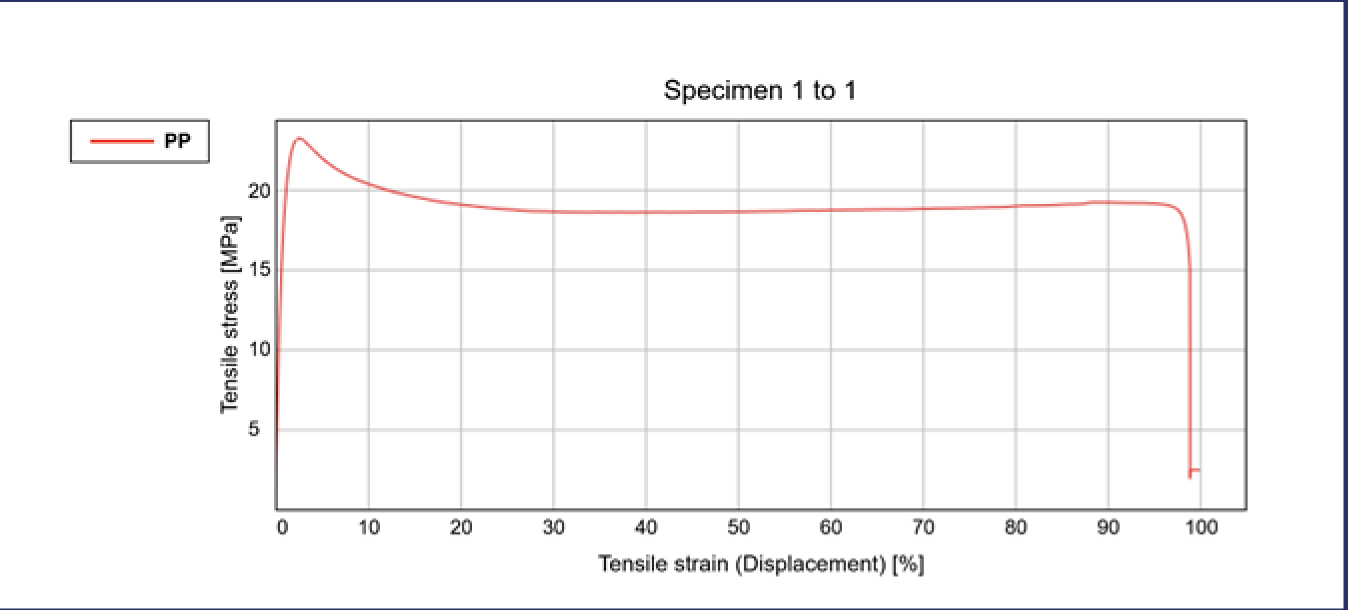
Aluminium 6061 Stress Versus Strain Curve



Brass 360 Stress Versus Strain Curve



Polypropylene Stress Versus Strain Curve



SolidWorks FEA Analysis - Models

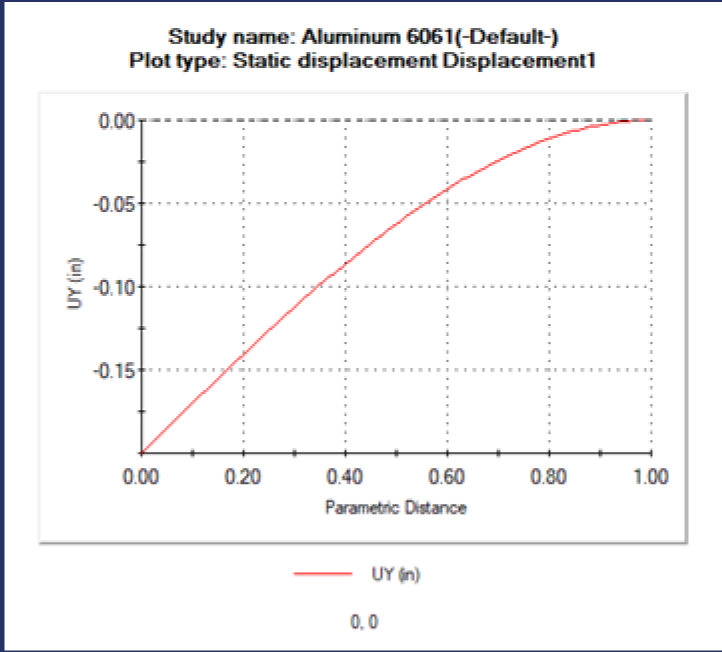
Linear Static

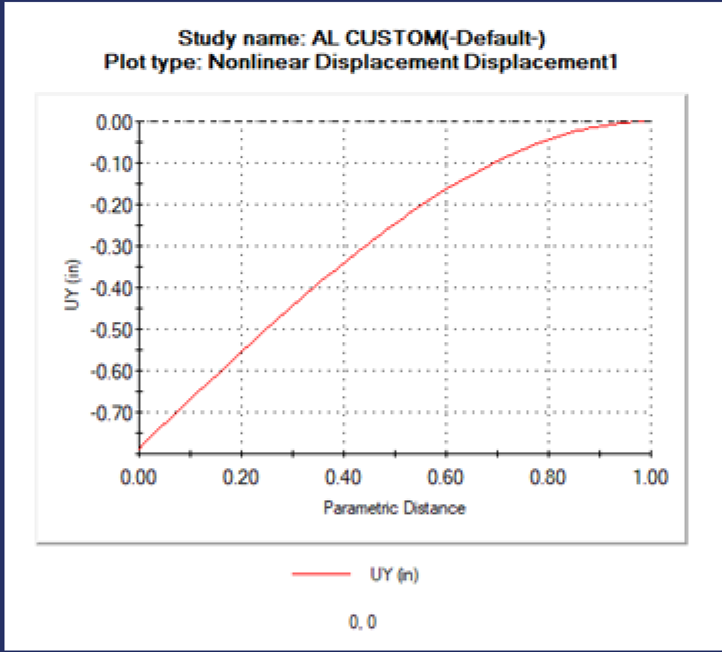
Assumes a proportional relationship between stress and strain that does not account for plastic deformation.

Plasticity von Mises

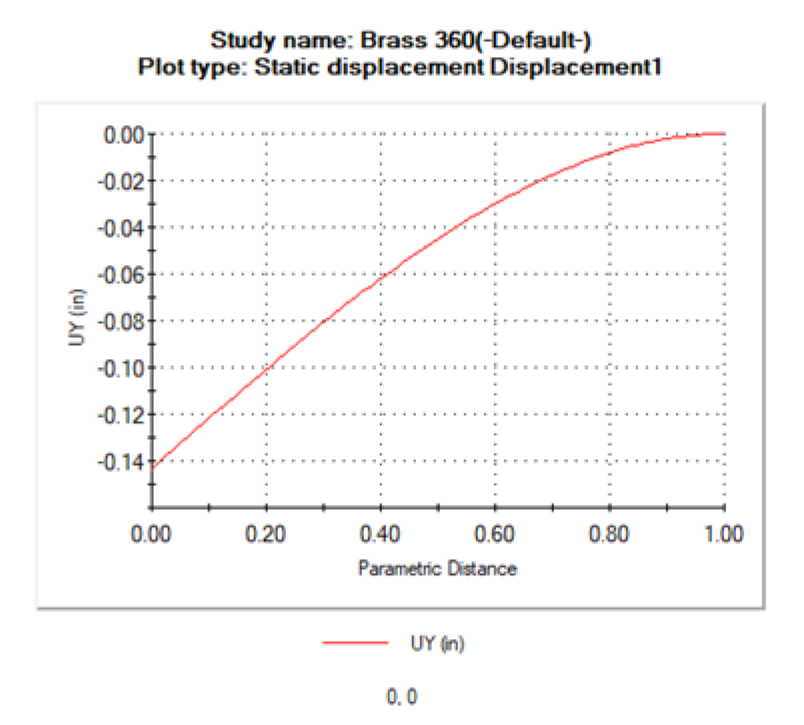
Simulates plastic deformation by determining when a point in the material reaches its yield strength based on imported data.

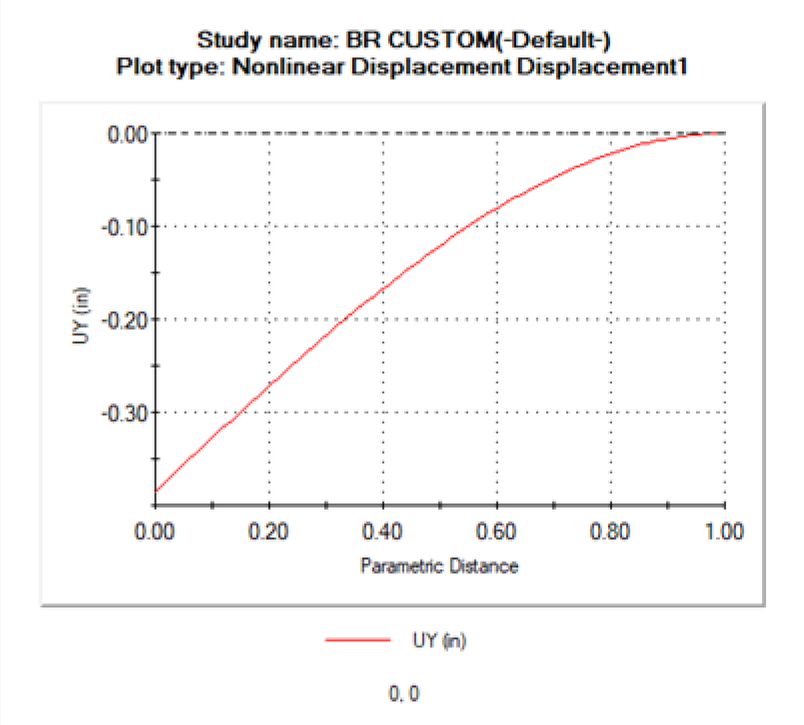
Aluminum Displacement Comparison



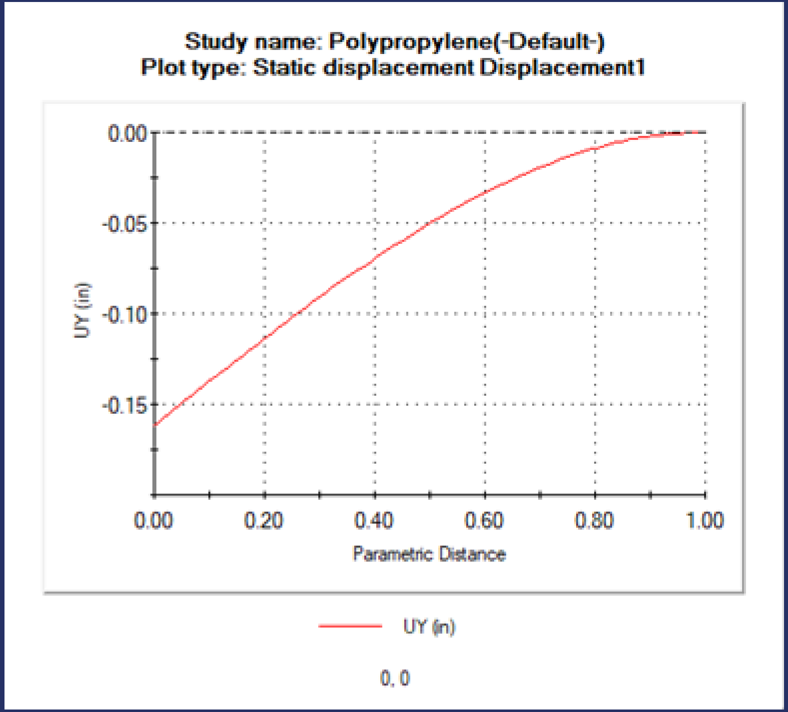


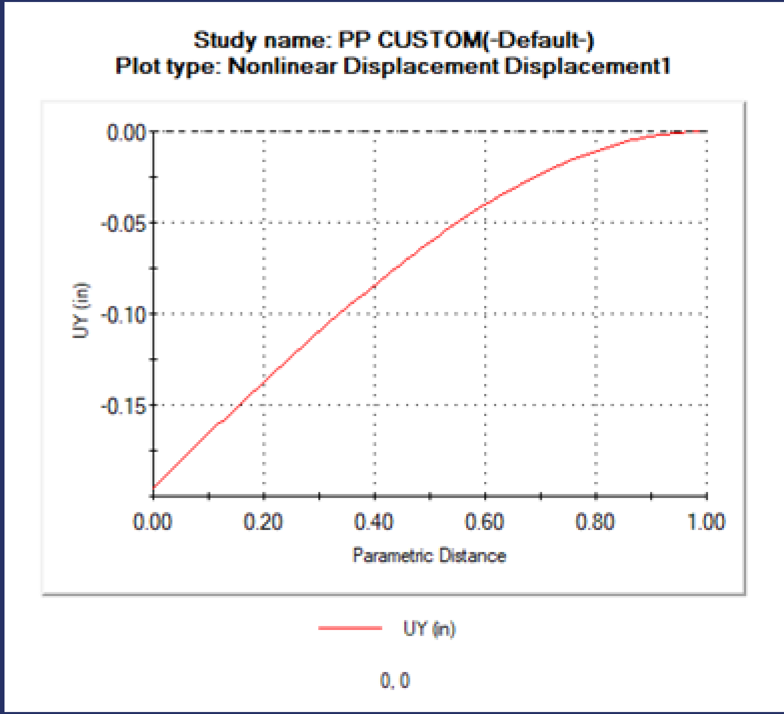
Brass Displacement Comparison





Polypropylene Displacement Comparison





Maximum Displacement Comparisons

Material	Linear Model	Nonlinear Model
Aluminum 6061	0.200	0.788
Brass 360	0.143	0.368
Polypropylene	0.162	0.195

Conclusion

Throughout the project, we overcame challenges, from issues with the waterjet in Bray Laboratory to the implementation of a custom elastoplastic model in SolidWorks. Vin Montuori and Professor Gary Leisk were kind enough to help us through our challenges when it came to fabrication and analysis, respectively.

Overall, the analysis was a success and we are able to conclude that all three materials benefit from using an elastoplastic model rather than the standard linear model used in SolidWorks. Aluminium 6061 showed a difference in maximum deflection of 0.6 inches from linear to plasticity von Mises, Brass 360 saw approximately 0.25 in, and polypropylene was 0.04 in. The higher values seen in the metals could be due to the fact that they are better suited for a plasticity von Mises study, but further analysis would need to be conducted to confirm this assumption.