Finite Element Analysis of a Cart Frame

By: Ben Wilfong

February 18, 2022

In partial fulfillment of the requirements of the course
ME422-02
Introduction to Finite Element Fundamentals
Professor S. Jones

1 Introduction

This report performs a finite element analysis on the frame of the cart design for my capstone project. The frame is shown below.

I plan to determine the deflection and bending stress for a worst case and realistic loading scenario representative of the items being carried by the cart. The bending stress will be used to determine the likelihood of failure in bending and the deflection will be used to determine whether the middle wheels of the cart will contact the ground and bind its motion. The use of complex, non-linear contact elements will be avoided, so the plywood panel that sits atop this frame as well as the bolted interactions will be neglected. Instead, the assumption is made that all connections between members are rigid and that loads can be applied in a way that distributes them over the members available (the plywood would also require the use of a highly non-linear material model). This assembly is made of 80/20 aluminum extrusions, and given its complex cross section, would require small elements that would easily exceed the abilities of my laptop. To mitigate this, beam elements will be used to reduce computational expense. This report begins by validating the use of beam elements through a 1-D and 2-D cantilever beam validation and then analyzes the full frame.

2 1D Transverse Load

2.1 Model Geometry

The geometry for the model is shown below in Figure 1 (units are in m.). This is an unaltered screenshot of the engineering drawing generated by Solidworks.

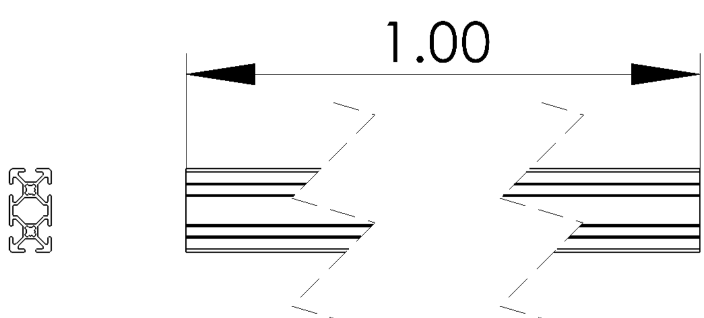

Figure 1: Geometry of the 1D Validation Case

The cross-section is not dimensioned in Figure 1 because the solid model was downloaded directly from the manufacturers website [1]. In the analysis, Ansys Space Claim was used to transform the solid model into a 1-D beam model.

2.2 Material Properties

The material for the model is 6105-T5 aluminum [2], which is assumed be behave as a linear, elastic, isotropic material. Figure 2 shows the definition and assignment of this material to the Ansys Model.

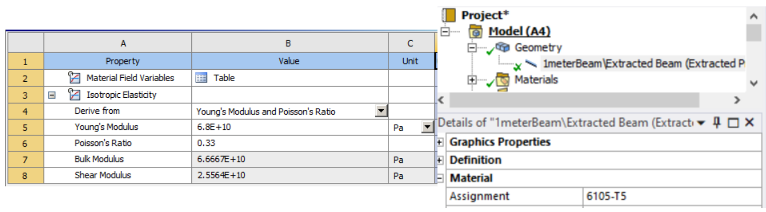

Figure 2: Material Definition and Assignment

2.3 Element Type, Mesh, and Model Size

For this validation, the default mesh provided by Ansys was used. This model used quadratic beam 188 elements. A total of 43 nodes and 21 elements are in the model. No element quality metrics are needed because these are all 1D elements. Figure 3 shows the final mesh used for this model.

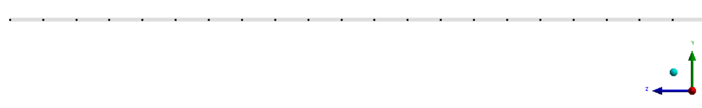

Figure 3: Final Mesh for Simulation

2.4 Load Configuration

A transverse point load of 250 N in the negative y direction is applied to the right end of the beam causing transverse bending. The application of this load is shown in Figure 4. For visualization purposes, the cross section is displayed throughout the rest of this 1D analysis.

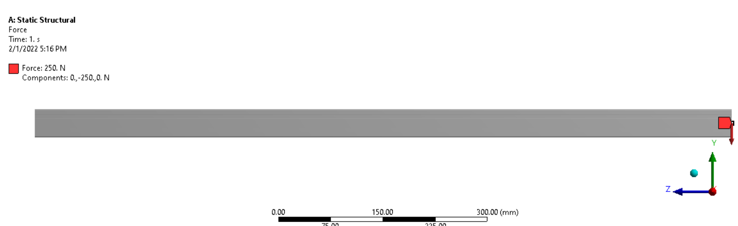

Figure 4: Loading Configuration for the Model

[1] https://8020.net/20-2040.html

[2] Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

2.5 Support Conditions

1 fixed support was applied to the left end of the model. The application of this support is shown in Figure 5.

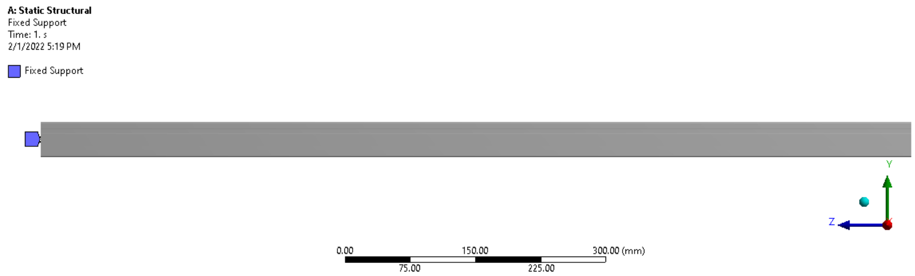

Figure 5: Support Constraints

2.6 Solution Verification

The solution solved was a static structural analysis performed by Ben Wilfong. The screenshots in Figure 6 are taken from Tables 11 and 12 of the ANSYS Report Preview.

Analysis Data Management	
Solver Files Directory	C:\Users\wilfonba\AppData\Local\Temp\WB_WILFONBA-STG4_wilfonba_31732_7\unsaved_project_files\dp0\SYS\MECH\

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Figure 6: Solution Confirmation and Simulation Authorship Confirmation

2.7 Results and Analysis

The results for the model are validated against the deflection as a function of distance from the fixed support and the maximum bending moment of a 1D analytical solution. Hand calculations are attached as Appendix A. Figure 7 shows the deflection along the length of the beam.

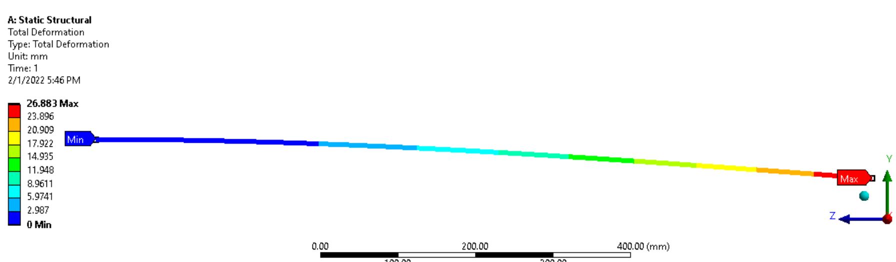

Figure 7: Deflection Results

Plotting the modeled deflections against the analytical solutions yields the results shown in Figure 8. The maximum relative error between the finite element model and the analytic solution is 1.13%. Figure 9 shows the bending moment along the length of the beam. The maximum bending moment of 250 N·m matches the analytic solution exactly.

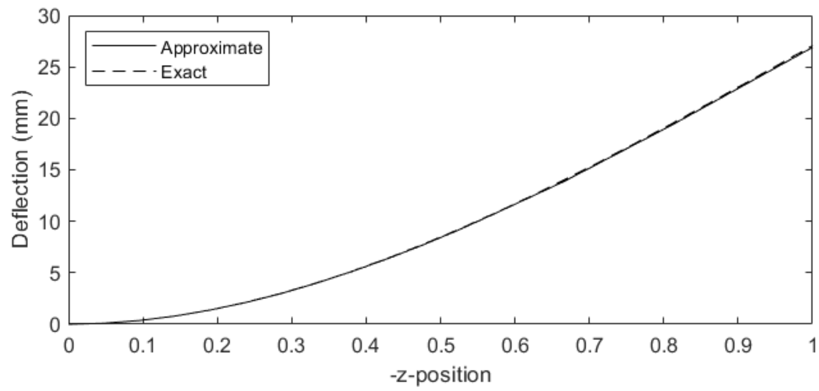

Figure 8: Theoretical vs. Experimental Results

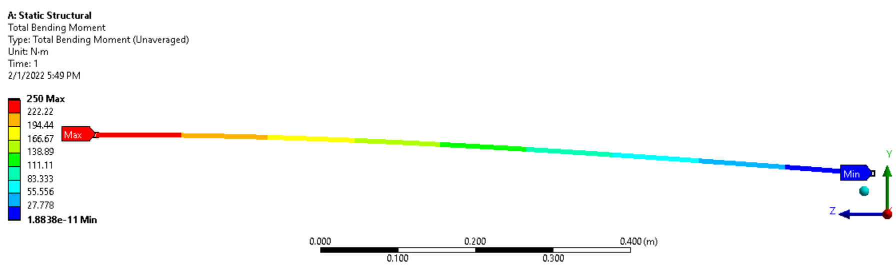

Figure 9: Bending Moment Results

3 2D Transverse Loading

3.1 Model Geometry

An isometric view of the geometry used in this model is shown below in Figure 10 (units in m.). This is an unaltered screenshot of the engineering drawing generated by Solidworks.

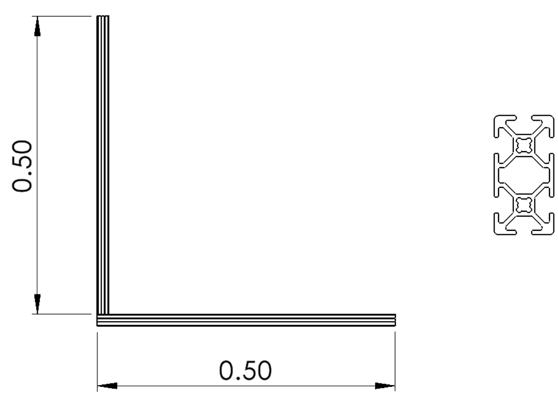

Figure 10: Geometry of the 2D Validation Case

The cross-section is not dimensioned in Figure 10 because the solid model was downloaded directly from the manufacturers website [3]. In the analysis, Ansys Space Claim was used to transform the solid, two-body model into two, 1-D beam models.

[3] https://8020.net/20-2040.html

3.2 Material Properties

The material for the model is 6105-T5 aluminum [4], which is assumed be behave as a linear, elastic, isotropic material. Figure 11 shows the definition and assignment of this material to the Ansys Model.

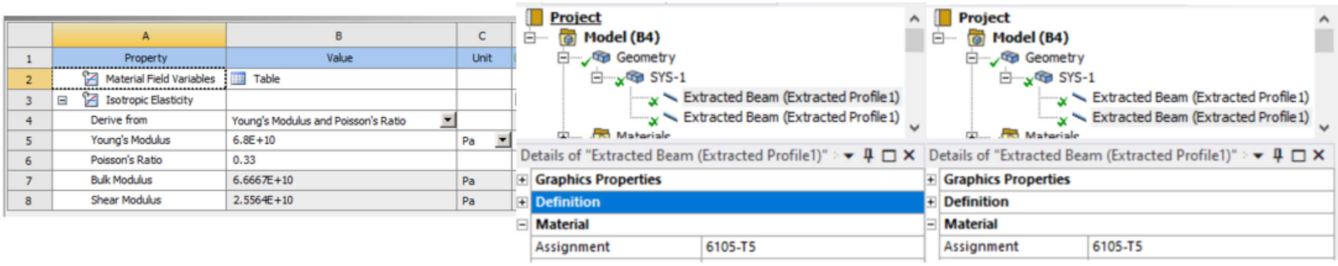

Figure 11: Material Definition and Assignment

3.3 Element Type, Mesh, and Model size

For this validation, the default mesh provided byAnsys was used. This model used quadratic Beam 188 elements. A total of 91 nodes and 30 elements are in the model. No element quality metrics are needed because these are all 1D elements. Figure 12 shows the final mesh used for this model.

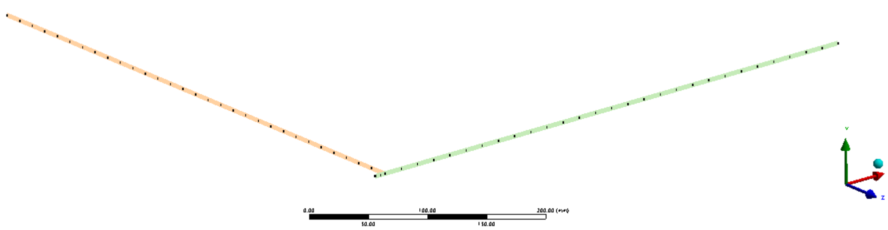

Figure 12: Final Mesh for Simulation

3.4 Load Configuration

A transverse point load of 100 N in the negative y direction is applied to the structure as shown in Figure 4. For visualization purposes, the cross section is displayed throughout the rest of this 1D analysis.

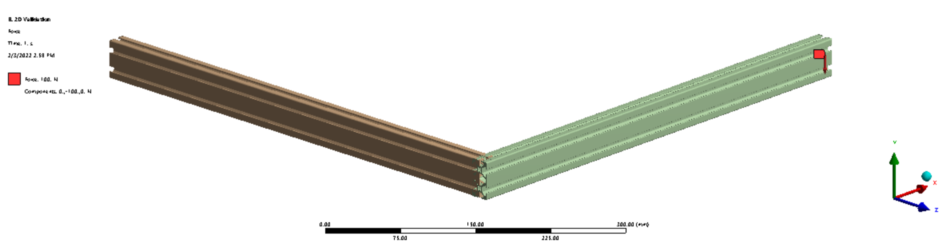

Figure 13: Loading Configuration for the Model

[4] Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

3.5 Support Conditions

1 fixed support was applied as shown in Figure 14.

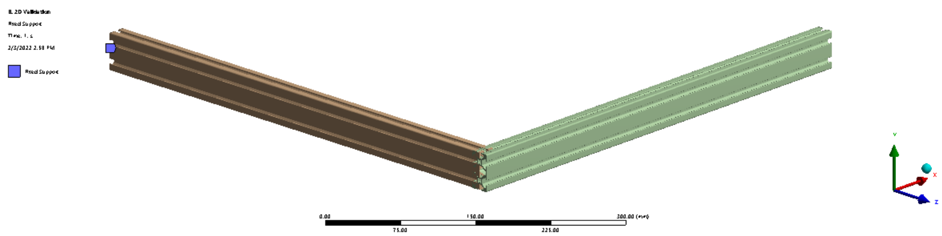

Figure 14: Support Constraints

3.6 Solution Verification

The solution solved was a static structural analysis performed by Ben Wilfong. The screenshots in Figure 15 are taken from Tables 11 and 12 of the ANSYS Report Preview.

Analysis Data Management	
Solver Files Directory	C:\Users\wilfonba\Documents\ME422\Project\Real Validation\projectValidation_files\dp0\SYS-1\MECH\

Object Name	*Static Structural (B5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Figure 15: Solution Confirmation and Simulation Authorship Confirmation

3.7 Results and Analysis

The results for the model are validated against a finely meshed solid body solution detailed in Appendix B. Figure 16 shows the total deflection throughout the structure. Qualitatively, the trend in deflection looks very similar to

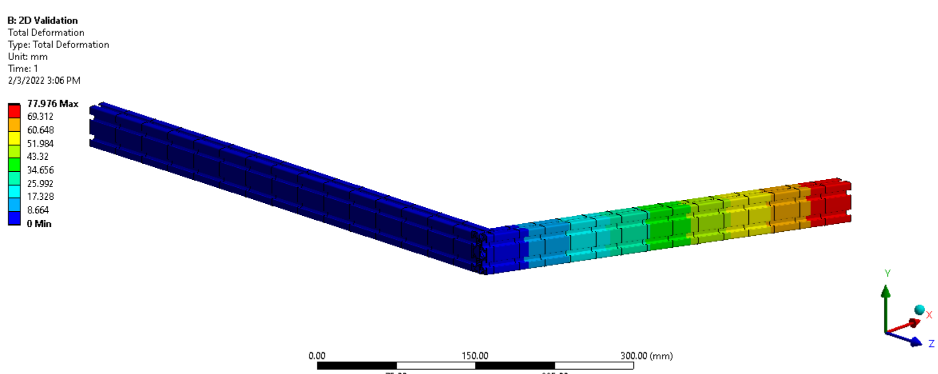

Figure 16: Deflection Results

that of the solid body model. The maximum deflection of the beam model is 77.8 mm while the maximum deflection of the solid body model is 75.4 mm. This corresponds to a relative error of only 3.13%, validating the use of beam elements for modeling. The bending moment throughout the structure is given below in Figure 17

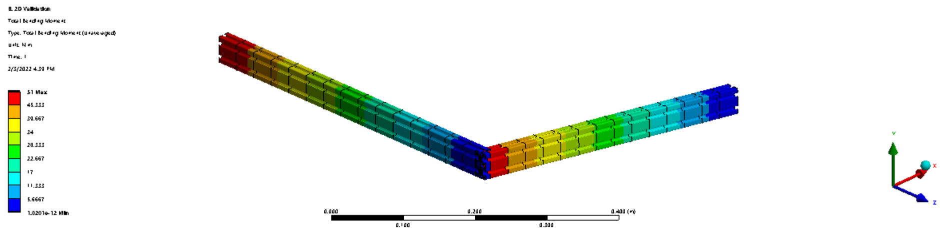

Figure 17: Bending Moment

The analytical solution is given by

$$M = Fl = (100 \text{ N})(0.51 \text{ m}) = 51\text{N} \cdot \text{m}.$$

The beam model matches the analytical solution exactly. The beam model results were calculated in just 1.4 seconds of CPU time, while the solid body model took 940 seconds to solve. By using beam elements, the computational cost is decreased by a factor of 671 while still yielding meaningful results.

4 Validation Conclusion

In conclusion, the use of 1-D beam elements have been shown to provide quantitatively meaningful results both in simple, single beam systems as well as in jointed systems. Additionally, the use of beams decreases the computational cost by a factor of 671. Without the use of beam elements, it would not be possible to model the cart frame of interest with the computational resources available.

5 Model Preparation for Complete Frame

5.1 Model Geometry

The model geometry for the full analysis is shown below in Figure 18.

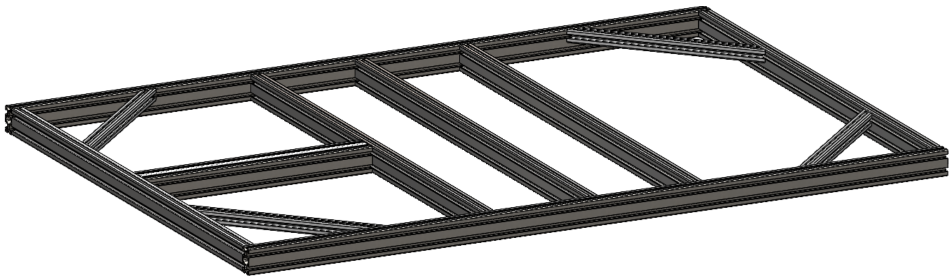

Figure 18: Complete Frame Geometry

This Solidworks assembly of this frame was imported directly into Ansys and all solid bodies were converted to 1-D beams using Space Claim. The result of this conversion is shown below in Figure 19

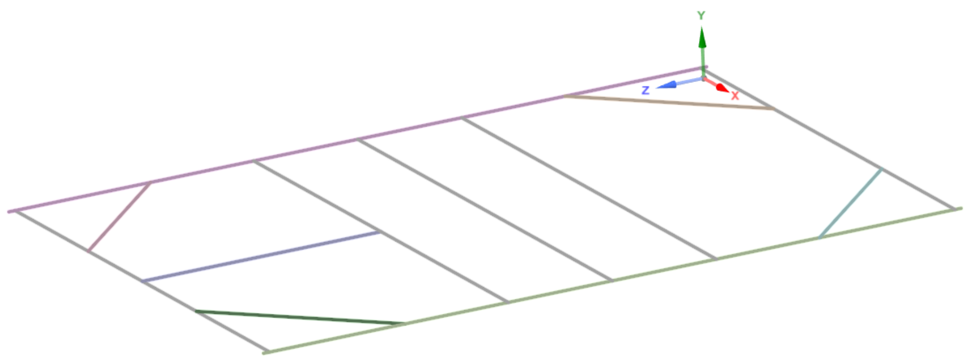

Figure 19: Complete Converted Frame Geometry

Symmetry was not used in this analysis in order to simplify application of boundary conditions.

5.2 Material Properties

The material for the model is 6105-T5 aluminum [5], which is assumed be behave as a linear, elastic, isotropic material. Figure 20 shows the definition of this material. Verification of material assignment is available in Figure C.1 in Appendix C.

Property	Value	Unit
Material Field Variables	Table	
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	6.8E+10	Pa
Poisson's Ratio	0.33	
Bulk Modulus	6.6667E+10	Pa
Shear Modulus	2.5564E+10	Pa

Figure 20: Material Definition

[5]Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

5.3 Element Type, Mesh, and Model Size

For this validation, a body sizing of 10 mm was applied to all bodies of the model. This model uses quadratic Beam 188 elements. A total of 1536 nodes and 772 elements are in the model. No element quality metrics are needed because these are all 1D elements. Figure 21 shows the final mesh used for this model.

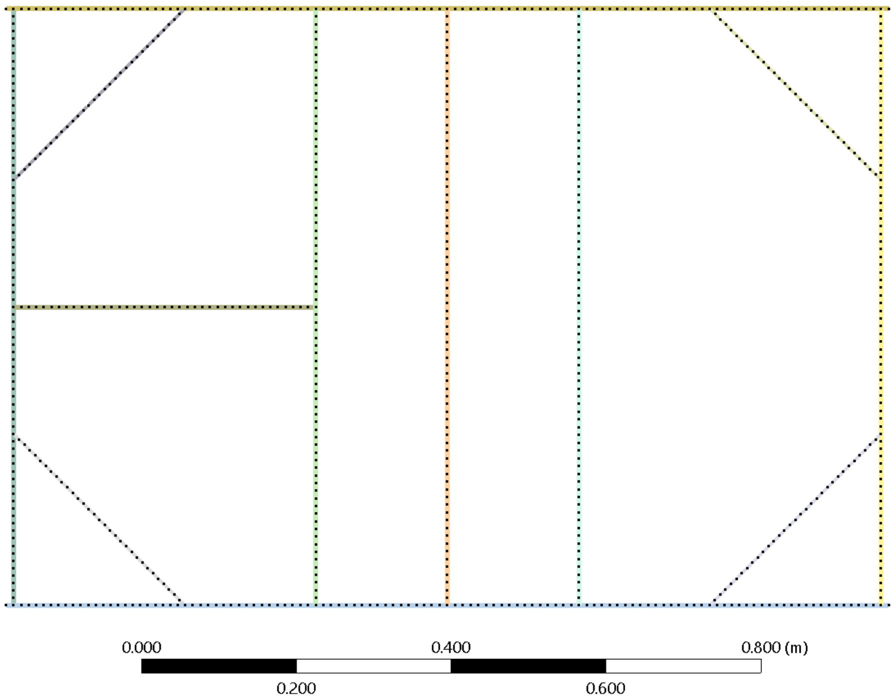

Figure 21: Final Mesh for Simulation

5.4 Support Conditions

A displacement constraint with free motion in the x and z directions and 0 displacement in the y direction was applied to the three vertexes shown in Figure 22

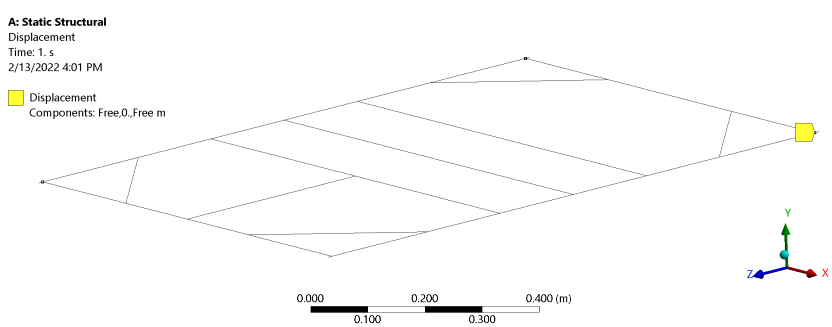

Figure 22: Displacement Condition 1

A displacement constraint with restricted motion in all three axes was applied to one vertex shown in Figure 23.

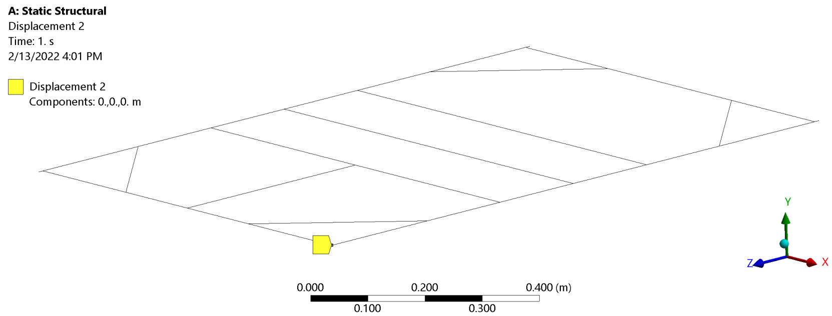

Figure 23: Displacement Condition 2

A simple support constraint was applied to the edge shown in Figure 24

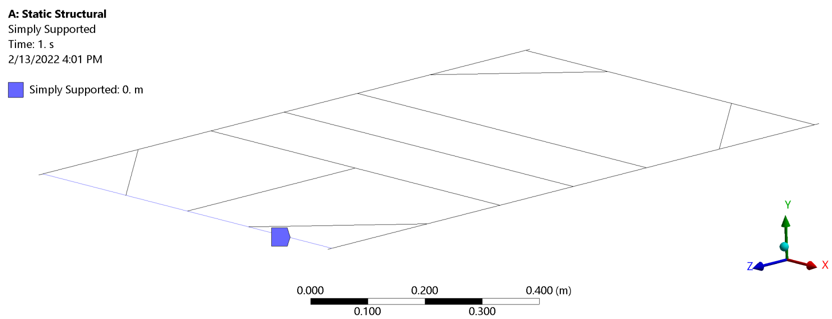

Figure 24: Simple Support Condition

6 Worst Case Loading

6.1 Load Configuration

The worst case loading scenario is when the entire designed for load is applied to the center of the cart as a point load. This worst case load of 2500N is applied as shown in Figure 25

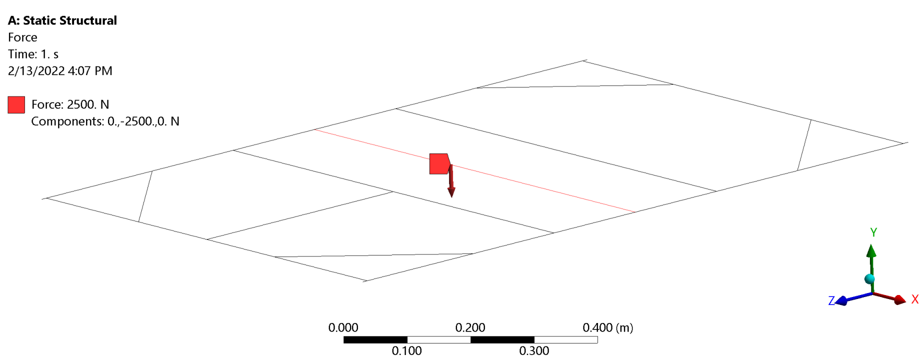

Figure 25: Loading Configuration for Worst Case Analysis

6.2 Solution Verification

The solution solved was a static structural analysis performed by Ben Wilfong. The screenshots in Figure 26 are taken from Tables 14 and 15 of the ANSYS Report Preview.

Analysis Data Management	
Solver Files Directory	C:\Users\wilfonba\Documents\ME422\Project\Final Analysis\frameAnalysis_files\dp0\SYS\MECH\

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Figure 26: Solution Confirmation and Authorship Confirmation

6.3 Results and Analysis

The worst case deflection and bending moment are shown in Figures 27 and 28 respectively.

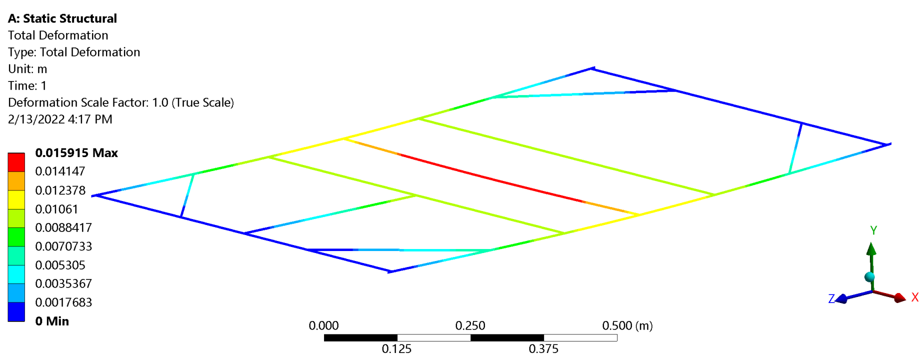

Figure 27: Worst Case Deflection

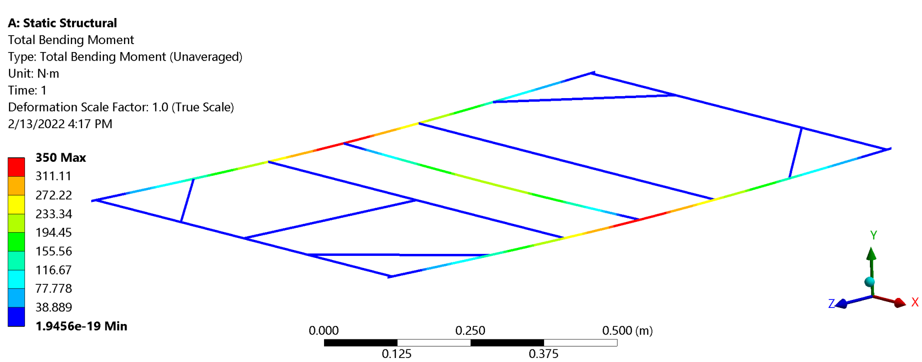

Figure 28: Worst Case Bending Moment

Since the largest deflection and bending moment both occur at locations without numerical singularities, the maximum values of 15.9 mm and 350 N·m can be read directly from the scale on the left. This maximum bending moment yields a maximum bending stress of

$$\sigma_{\text{bend,max}} = \frac{My}{I} = \frac{(350\ \text{N}\cdot\text{m})(0.020\ \text{m})}{4.5257\times10^{-8}\ \text{m}^4} = 154.7\ \text{MPa}.$$

This maximum bending stress is well below the tensile strength of 270 MPa[6] implying that the cart should not statically fail even when loaded in a worst case scenario. This maximum bending stress is however above the fatigue strength of 130 MPa[7], implying that fatigue failure would happen if the cart was loaded and unloaded in this way repeatedly. The maximum deflection along the path shown in Figure 29 is 11.8 mm. This is well below the 1" clearence between our breakover wheels and the ground which means these fixed wheels will not contact the ground and prevent normal operation of the cart.

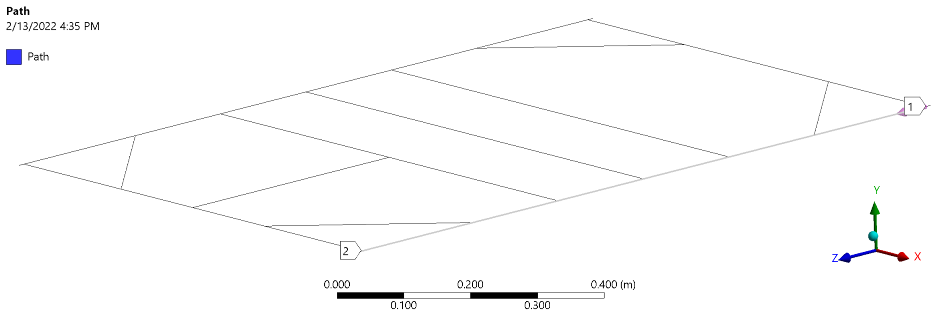

Figure 29: Path Location

7 Realistic Loading

7.1 Load Configuration

For the realistic loading case the total load is split into five pieces and applied to five different members of the frame. The locations of the 500 N loads are shown in Figure 30

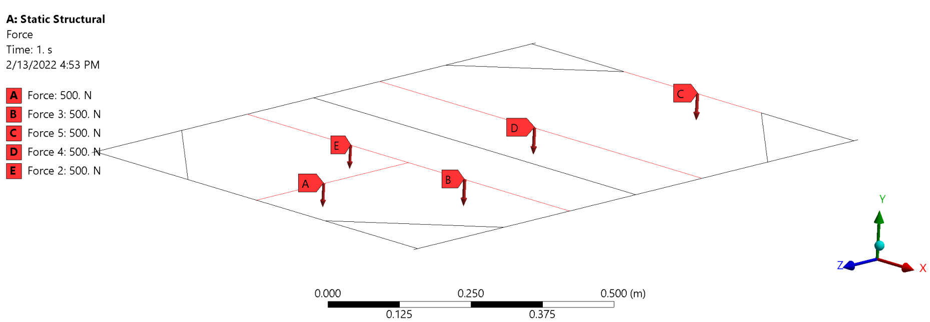

Figure 30: Loading Configuration for Worst Case Analysis

7.2 Solution Verification

The solution solved was a static structural analysis performed by Ben Wilfong. The screenshots in Figure 31 are taken from Tables 16 and 17 of the ANSYS Report Preview.

[6]Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy
[7]Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

Analysis Data Management	
Solver Files Directory	C:\Users\wilfonba\Documents\ME422\Project\Final Analysis\frameAnalysis_files\dp0\SYS\MECH\

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Figure 31: Solution Confirmation and Authorship Confirmation

7.3 Results and Analysis

The realistic deflection and bending moment are shown in Figures 32 and 33 (next page) respectively.

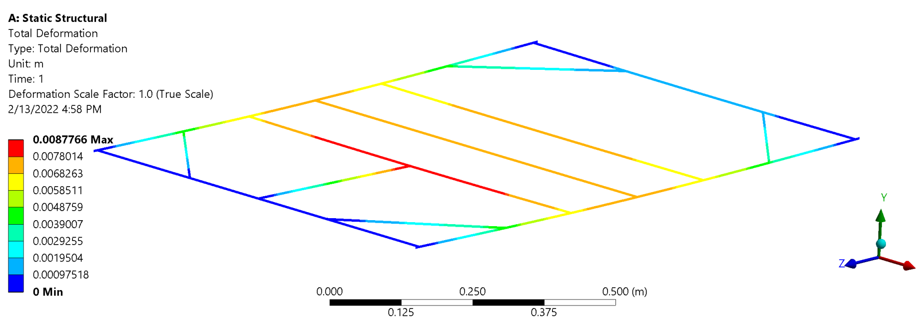

Figure 32: Worst Case Deflection

Since the largest deflection and bending moment both occur at locations without numerical singularities, the maximum values of 8.78 mm and 193 N·m can be read directly from the scale on the left. This maximum bending moment yields a maximum bending stress of

$$\sigma_{\text{bend,max}} = \frac{My}{I} = \frac{(193\text{ N}\cdot\text{m})(0.020\text{ m})}{4.5257\times10^{-8}\text{ m}^4} = 85.3\text{ MPa}.$$

This maximum bending stress is well below the tensile strength of 270 MPa [8] implying that the cart will not statically fail. This maximum bending stress is also below the fatigue strength of 130 MPa [9], implying that the cart will not prematurely fail due to fatigue either. The maximum deflection along the path shown in Figure 29 is 7.20 mm. This is well below the 1" clearence between our break-over wheels and the ground which means these fixed wheels will not contact the ground and prevent normal operation of the cart.

[8] Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy
[9] Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

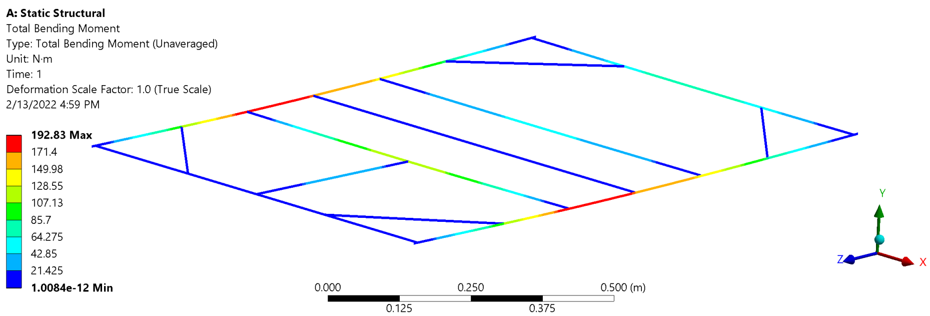

Figure 33: Worst Case Bending Moment

8 Conclusion

The maximum bending stress and middle wheel deflection of the cart in a worst case loading scenario are 154.7 MPa and 11.8 mm respectively. While the motion of the cart will not bind in this scenario, it is expected that the cart would eventually fail in fatigue. The maximum bending stress and middle wheel deflection of the cart in a realistic loading scenario are 85.3 MPa and 7.20 mm respectively. In this scenario, the motion of the cart will not bind and the cart should not fail in fatigue.. In conclusion, this Finite Element model has demonstrated that under realistic loading, the cart frame should not experience fatigue failure in bending or deflect enough to bind motion. The use of beam elements allowed for this conclusion to be drawn in a fraction of the time a 3-Dimensional solid body solution would have taken.

A 1D Hand Calculations

The deflection of a cantilever beam as a function of length from fixed support, x, is

$$\delta = \frac{Px^2}{6EI}(3l - x).$$

The moment of inertia, I, as given by the manufacturer[10] is 4.5357 cm^4 or 4.5257×10^{-8} m^4. Substituting this along with $l = 1\text{m}$ and $E = 68 \times 10^9$ Pa yields

$$\delta(x) = \frac{(250)x^2}{6(68 \times 10^9)(4.5257 \times 10^{-8})}(3 - x).$$

Expanding yields

$$\delta(x) = 0.013539x^2(3 - x).$$

The maximum bending moment is simply calculated via

$$M = Fl = (250)(1) = 250\text{N} \cdot \text{m}.$$

[10] https://8020.net/20-2040.html

B Solid Body Model for 2D Validation

Since an analytic solution to the 2D problem is not readily available, the validation model used was a finely meshed solid body model with the same geometry and loading configurations. The material for the model is 6105-T5 Aluminum [11], which is assumed to behave as a linear, isotropic material. Figure B.1 shows the definition and assignment of this material to the Ansys model.

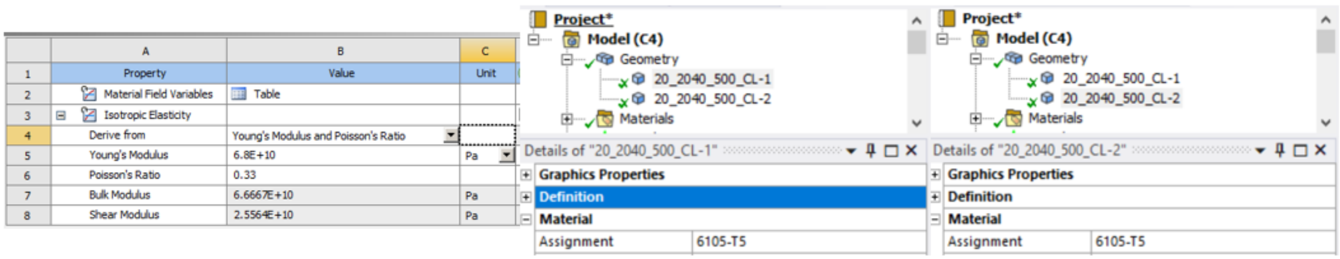

Figure B.1: Material Definition and Assignment

For this validation, a mesh sizing of 1mm was applied to both solid bodies. The model used 2,673,000 Quadratic Tet 10 Elements and 4,415,204 nodes. The average element quality is 0.79. Figure B.2 shows a small section of mesh that is indicative of the entire model.

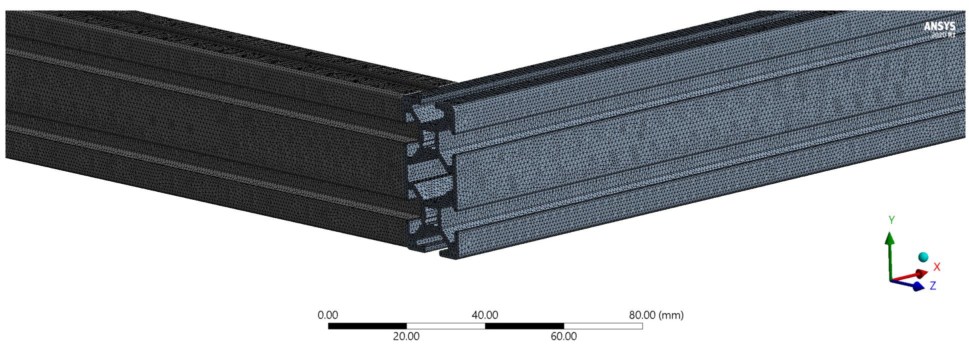

Figure B.2: Final Mesh for Simulation

A transverse point load of 100 N in the negative y-direction was applied at face A and a fixed support was applied to the face at B as shown in Figure B.3.

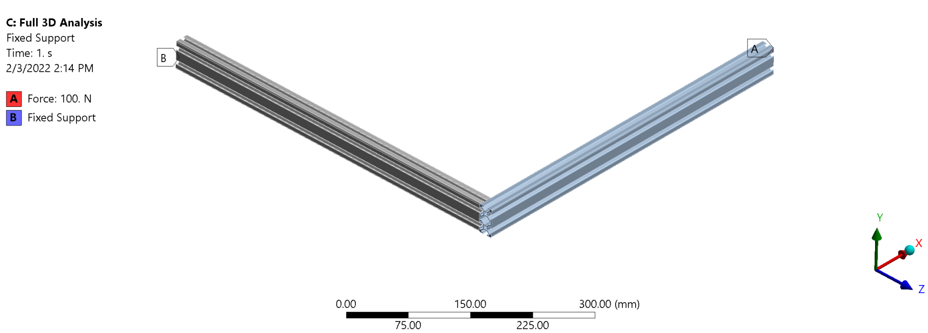

Figure B.3: Support and Loading Configuration of the Model

[11] Material properties obtained makeitfrom.com for 6105-T5 aluminum alloy

The solution solved was a static structural analysis performed by Ben Wilfong. The screenshots in Figure B.4 are taken from Tables 11 and 12 of the ANSYS Report Preview.

Analysis Data Management	
Solver Files Directory	C:\Users\wilfonba\Documents\ME422\Project\Real Validation\projectValidation_files\dp0\SYS-2\MECH\

Object Name	*Static Structural (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Figure B.4: Solution Confirmation and Simulation Authorship Confirmation

Figure B.5 shows the total deflection of the structure.

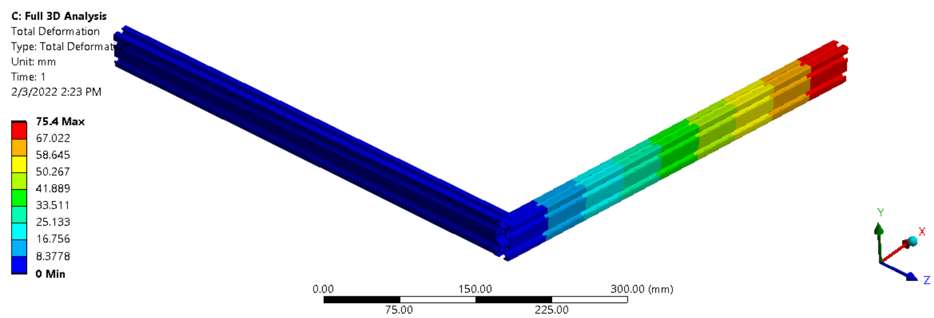

Figure B.5: Total Deflection

Due to the combined torsion and deflection, the maximum deflection of 75.4 mm will be used for validation.

C Material Assignment in Final Model

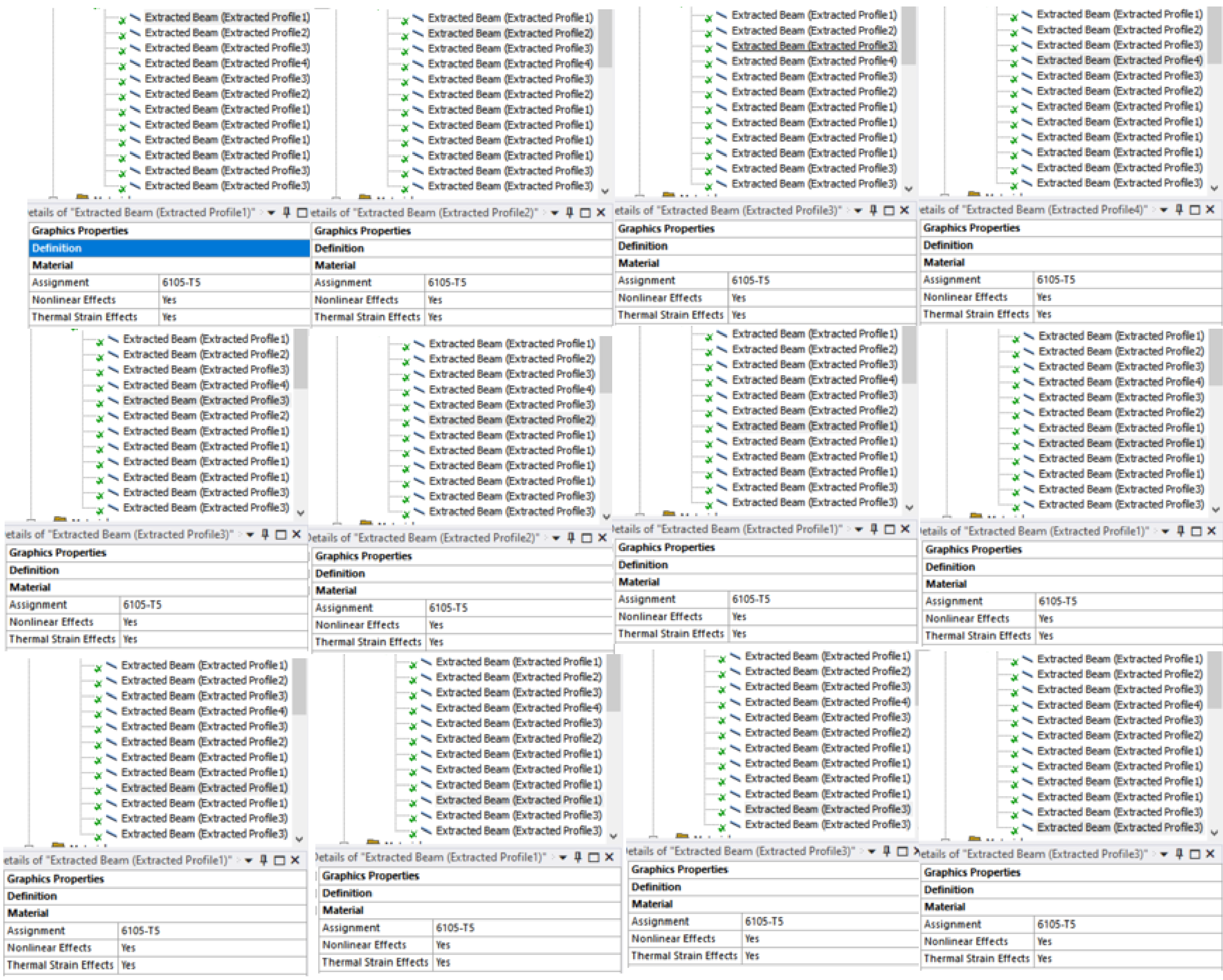

Figure C.1: Verification of Material Assignment