# DESIGN OPTIMIZATION

**12/9/2022**

**Humfrey Kimanya** **Linh Vu** **Rafael Luna**

## ABSTRACT

This is a Final Design project Report (FDR) of an optimized balsa structure that efficiently supports a limb in a human vehicle. The structure is designed to occupy the given space envelope, where the theoretical analysis of Strength-to-Weight (S/W) ratio was computed. The structure is designed to occupy the given space envelope, where theoretical analysis of the Strength-to-Weight (S/W) ratio was computed. The structure was designed to support up to a load of 250 $lb_f$. By designing using Siemens NX and analyzing using NASTRAN, the S/W ratio was found to be **4007**. The design was manufactured by using different rectangular cross-sections of wood beams made of balsa. A rig with similar loading and boundary conditions to the bearing surface will be used to test the structure. The actual value of S/W will be compared to the theoretical one. This project provides practice in using FEA software and structural engineering for optimization.

## PROBLEM STATEMENT

A balsa structure that could support a limb in a human vehicle was required. The structure was required to weigh no more than 0.15 $lb_f$, and a load between 10 $lb_f$ and 250 $lb_f$ is applied at the top of the structure. According to Fig. 1, the structure should be within the space envelope and touch the support only where allowed. The material and cross-sectional properties of balsa wood are given in Table 1.

| Material Properties | | Geometry | |
|---|---|---|---|
| | | Cross-section size [in x in] | Length [in] |
| Mass density $\rho$ [$lb_m/in^3$] | 0.0065 | 1/8 x 1/8 | 24 |
| Young's Modulus $E$ [psi] | 0.55E6 | 3/16 x 3/16 | 24 |
| Shear Modulus $G$ [psi] | 0.25E6 | 3/32 x 3/32 | 24 |
| Ultimate Strength $S_U$ [psi] | 1000 | 1/16 x 1/4 | 24 |

Table 1 – Material Properties of Balsa and given cross-sectional geometry.

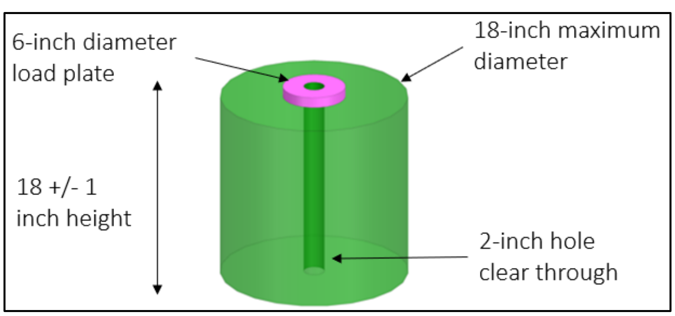


Figure 1 – Space envelope of the structure.

## COLLABORATION STRATEGY

To collaborate efficiently in a team of 3, each of the team members first designed their structure in Siemens NX. The initial designs are shown in Fig. 3 in the Annex. A Preliminary Design Review (PDR) was then created to define the next steps using a Pugh matrix approach as shown in Table 2 in the Annex. Additional optimization was conducted individually up to the point a final design was chosen based on the highest S/W and the assembly difficulty level. From there, all team members contributed to manufacturing the final structure and writing this FDR.

## DESIGN APPROACH/RESULTS

To develop a design in Siemens NX, simple polygon geometries were created, and then geometrical patterns were associated with the polygons. The initial iterations are shown in Fig. 3 (Annex). The material properties were input, and a NASTRAN analysis was executed. After each member of the team developed their initial iteration plans, stress analyses were then conducted to understand the behavior of the structure.

### *Setup*

The main goal of this project was to design a structure that could support up to 250 $lb_f$ applied vertically downward. Our approach was to design for a high S/W and have the failure load as close to 250 $lb_f$ as possible. This is because, even though the true S/W ratios are higher for an overdesigned structure with a critical load higher than 250 $lb_f$, the considered S/W ratio would decrease because only a maximum load of 250 $lb_f$ can be used in this ratio. This is one reason for choosing to optimize Design 3 despite Design 1 having higher scores in the Pugh matrix as shown in Table 2 in the Annex. Moreover, Design 3 had more optimization room for improvement.

Design 3 was developed by calculating the largest square that could fit under a 6-inch-diameter ($\boldsymbol{D}$) load plate which is defined by $\boldsymbol{D}\frac{\sqrt{2}}{2}$. Additionally, Design 3 was the simplest structure to fabricate compared to the octagon and pentagon bases of Design 1 and 2, as shown closely in Fig. 4 (Annex). Vertical beams supported the largest loads while the crossed beams held the vertical beams in place. To reduce vulnerability in buckling, the structure was divided into six sections. This reduced the effective length of the beams and raise their $\boldsymbol{P_{critical}}$. According to Eq. 1 below, $\boldsymbol{P_{critical}}$ is inversely proportional to effective length $\boldsymbol{L}$. Additionally, the squares around vertical beams maintained their integrity since they restricted buckling of the columns radially outwards.

$$P_{critical} = \frac{a\pi^2 EI}{L^2} \tag{1}$$

 

In the very first iteration, all members of the structure had the same cross-section of (3/16" x 3/16") to then analyze models of stress and buckling.

***Model Reduction***

During optimization, the vertical beams contained the highest compressive stress values. These column beams were kept at (3/16" x 3/16") cross-section, while smaller cross-sections were chosen for the braces and the squares around vertical beams to reduce the weight of the entire structure. Similarly, the cross sections of the outer vertical beams were reduced since the inner vertical beams supported the most stress. Consequently, the S/W ratio was **initially 2632** and then increased because only the inner beams were receiving most of the loads.

Specifically, S/W ratio is calculated using Excel, based on Eq. 2 below,

$$\frac{S}{W} = \frac{P_{fail}}{W} \tag{2}$$

Where $P_{fail}$ is defined to be either the $P_{critical}$ or $P_{ultimate}$, whichever is smaller. $P_{critical}$ of the structure relates to the applied force, $P_{applied}$, of 250 $lb_f$ through the slenderness ratio of the first buckling mode, $\lambda$, in Eq. (3). While in Eq. (4), $P_{ultimate}$ of the structure relates to the applied force through the ratio between the maximum absolute value of stress, $|S_{max}|$, and the ultimate strength of the material, $S_U$.

$$P_{critical} = P_{applied} \cdot \lambda \tag{3}$$

$$P_{ultimate} = P_{applied} \cdot \frac{S_U}{|S_{max}|} \tag{4}$$

Following these calculations, the structure was optimized, and the **final S/W is 4007**. Fig. 2 shows the final model with geometry and boundary conditions.

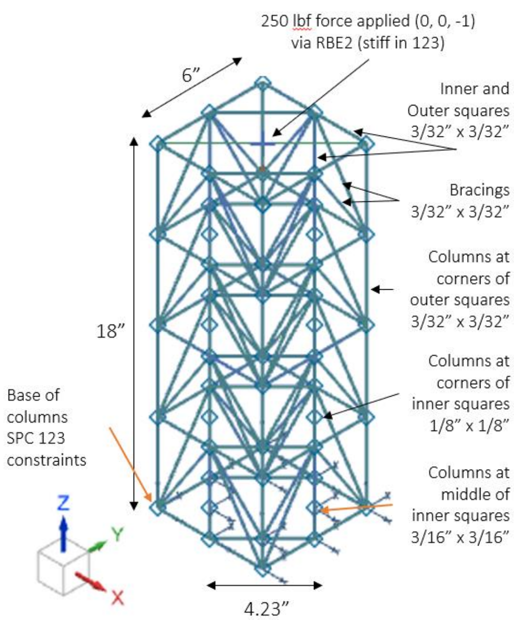


Figure 2 – Geometry and boundary conditions of final structure. The load was applied via RBE2 connector at the top and the base had fixed translation constraint.

## DISCUSSION

After performing theoretical analyses using Siemens NX, NASTRAN, and Excel, the following discussions can be made on our final design.

1. **At what load, and how do you anticipate your structure will fail?**
   The final design structure is anticipated to fail by buckling when a load of at least **239 $lb_f$** is applied. This was analyzed using NASTRAN by taking into consideration the design specifications (1) the maximum stresses (356 psi) were less than the material's ultimate strength of 1000 psi, (2) the weight of the structure (0.06 $lb_f$) was under 0.15 $lb_f$, and (3) the load at failure was within 10 - 250 $lb_f$ range.

2. **Design philosophy and optimization approach – what were the primary design handles you manipulated to try and reach optimal strength to weight?**
   The following techniques were adapted to design an optimized structure. Firstly, reducing the cross-section of less critical members was done so that the weight of the entire structure is lowered. Next, to mitigate buckling, adding braces and cross members that can reduce the effective length of the vulnerable members was also used.

3. **Model checkout/validation checks that you performed.**
   To confirm that the analysis in NASTRAN was set up correctly, the displacement, stress values, and reaction force results were checked. The structure was vertically displaced by less than 10% of its length; stress values did not exceed the given ultimate strength of 1000 psi; and the sum of the reaction forces was confirmed to match the applied load and the weight of the structure. This approach was done to confirm the validity of each iteration during the optimization process by using Excel. An example is shown in Table 3 in the Annex.

4. **What did you do to ensure that the NASTRAN model you built is accurate?**
   To ensure that the built NASTRAN model was accurate, boundary constraints were set correctly (fixed translation constraint) at the bottom of the structure. Additionally, making sure that the RBE2 connection is applied correctly was important. Here, each mesh point where the load is applied was verified to ensure that it is being distributed uniformly and realistically displaced. To model joint behavior between structure members, nodes were merged for beams that were used to simulate intersections without previous connections.

5. **What did you do to ensure that the Balsa model you built is accurate relative to the NASTRAN model?**
   While the modeled structure in Siemens NX produced ideal geometric configurations, it is worth noting, however, that the manufactured structure was not. Nonetheless, various techniques were employed to make the real structure relatively accurate to the NASTRAN model. To build joints at intersection points, notches of 45° were created on one end for each member. See Fig. 5 in the Annex. Furthermore, to accommodate the space required for gluing, each member

 

was slightly shortened in length by 1mm. Another approach employed was to minimize the use of glue since it added more weight. This was done at intersection points at cross beams by installing them on opposite sides of the structure. An illustration is shown in Fig. 6 in the Annex.

6. **How simulated model differs from reality?**
   It is known that there is a wide range of material properties of balsa wood because of manufacturing procedures. This causes uneven distribution of material properties along all the members which differ from the constant material properties used in the simulated analysis. In another manner, the nodes for the joints in the structure are modeled perfectly in simulation but as explained in the question before, there is mismatching when notches are glued together. The process of making the notches adds to the difference in the model because of the difficulty of making them. As an example, see Figure 7 in the Annex. Similarly, the geometry of the structure can be seen to vary from plane to plane, causing extra deformities not accounted for in the simulation (Fig.8, Annex). Since the material was given in 24-inch-long members, it was necessary to cut those into the desired shorter lengths using paper knives. This method causes the cut members to be fractured easily following the wood fiber directions. Lastly, something that was not taken into consideration is the weight of the glue used in the structure which reduces the S/W ratio.

7. **Do you expect the experimental strength to weight to be the same as simulated (why and how would it be different)?**
   The experimental S/W ratio is expected to be different because of (1) the amount of glue used, (2) notch misalignment at joints, (3) the uneven material properties of balsa wood, and (4) the added parchments at joints that were not manufactured correctly, as explained in the previous question.

8. **What would happen if the actual material failure stress was higher/lower, Modulus was higher/lower?**
   A different material failure stress dictates whether the structure can support more axial stresses or not. With higher failure stress, the structure would inherently be under-designed since it can support more load than theoretically computed. The vice versa is true. Modulus, on the other hand, affects the mode of failure of the structure. With larger elastic modulus, the structure becomes less vulnerable to buckling, and vice versa. This can be explained by Euler's buckling formula shown in Eq. 1 above.

## ACKNOWLEDGMENTS

Our team acknowledges the guidance and support from Professor Christopher Muir, Mr. Jim Alkins, and the Teaching Assistant team in completing this project.

## REFERENCES

ME 204 Fall 2022 – Mechanical Design, Project 4 Description, University of Rochester.

# ANNEX

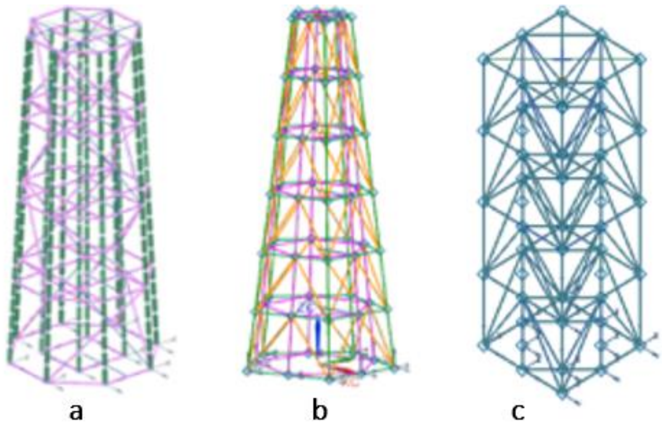


Figure 3 – Preliminary designs: (a) – Design 1 by Humfrey Kimanya, (b) – Design 2 by Linh Vu, (c) – Design 3 by Rafael Luna

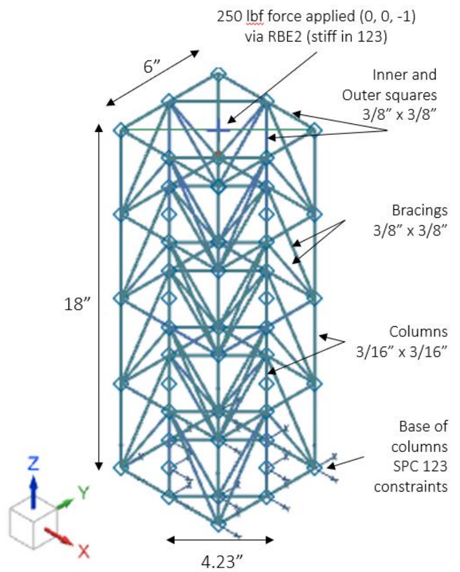


Figure 4 – Initial design of chosen structure (Design 3 in Fig. 3c). The load was applied using RBE2 connector at the top and the base had fixed translation constraint at the bottom.

| Criteria | Design 1 | Design 2 | Design 3 |
|---|---|---|---|
| Ease of assembly | + | Baseline | + |
| Robustness | | Baseline | |
| Optimization | + | Baseline | - |
| PDR S/W value | + | Baseline | - |
| Total | +3 | Baseline | -1 |

Table 2 – Pugh matrix for initial designs in PDR (shown in Fig. 3). Design 3 was later chosen for FDR because it became the best design after optimization.

| | | |
|---|---|---|
| $P_{APP}$ | 250 | lbf |
| $S_{ULT}$ | 1000 | psi |
| G | 386.6 | $in/s^2$ |

*Table 3(a)*

| | Smax | Smin | Lambda | Mass | \|Smax\| | $P_{ULT}$ | $P_{CR}$ | P* | safety | W | S/W |
|---|---|---|---|---|---|---|---|---|---|---|---|
| | psi | psi | | $lbf\text{-}s^2/in$ | psi | lbf | lbf | lbf | safe/fails | lbf | |
| Design 1 (hkimanya) | 2.37E+02 | -6.75E+02 | 1.53E+00 | 1.84E-04 | 674.68 | 370.55 | 382.00 | 370.55 | safe | 7.13E-02 | 3505.89 |
| Design 2 (lvu4) | 4.22E+02 | -8.33E+02 | 2.71E+00 | 1.89E-04 | 832.68 | 300.24 | 677.50 | 300.24 | safe | 7.29E-02 | 3428.40 |
| Design 3 (rluna3) | 4.32E+02 | -4.82E+02 | 6.97E+00 | 5.10E-04 | 481.89 | 518.79 | 1743.44 | 518.79 | safe | 1.97E-01 | 1268.41 |

*Table 3(b)*

Table 3 – Team Data Information from the PDR. The third column from the right in 3(b) illustrates the crushing safety of the designs by comparing the maximum stresses with the ultimate strength in 3(a).

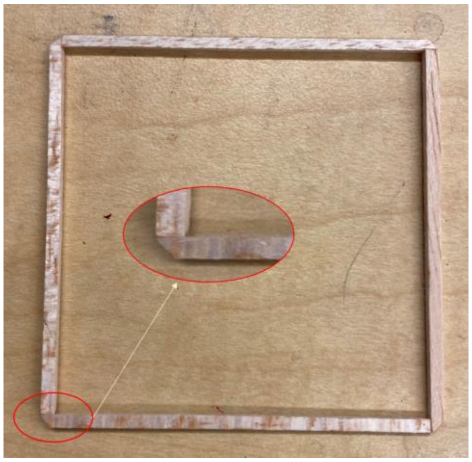

Figure 5 – 45° notch at one end of each member

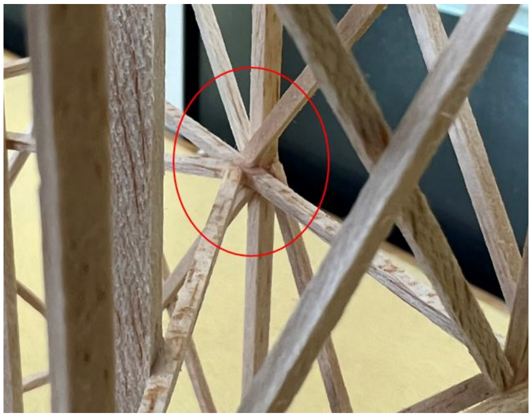

Figure 6 – Merging nodes in real-life structure to minimize the amount of glue used.

Figure 7 – Difficulty of merging nodes in real-life structure.

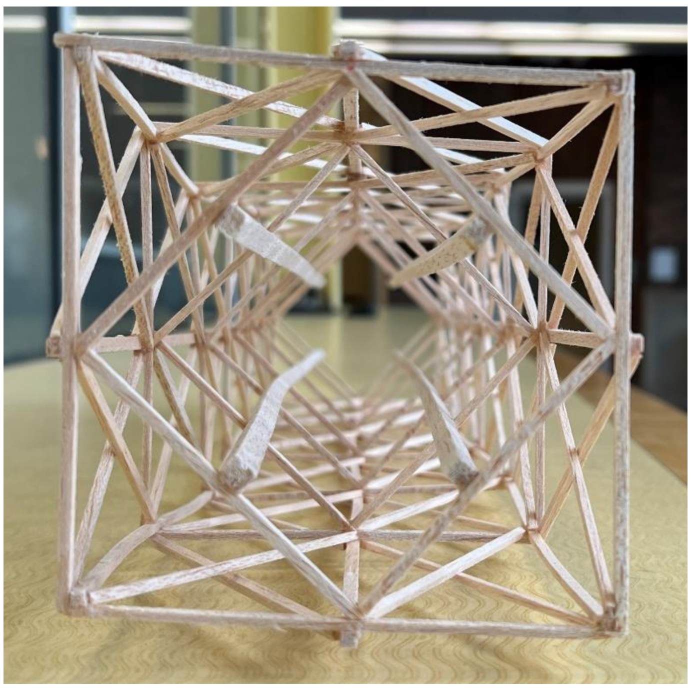

Figure 8 – Variation from plane to plane in real-life structure.