

OPTIMIZATION OF BEAM STRUCTURE

Paul Calderon

Abdulwahab Sayes

Jason Cao

ABSTRACT

Beam finite element models (FEMs) provide an efficient means of predicting deflection, stress, and buckling behavior in structures. The objective of this project was to design a balsa wood structure within a given space envelope and maximize its strength-to-weight ratio while meeting specific load and mass requirements. A top-loaded beam structure was modeled in Siemens NX/NASTRAN. Linear buckling analyses were performed repeatedly, with the goal being to optimize the structure iteratively with provided MATLAB scripts. Member sizes, location and present of cross bracing, and load path analysis were repeatedly adjusted using this method. The final design was fabricated using the provided balsa stock, and in a fashion which yielded a physical model matching closely with the FEM geometry. The analysis predicts a buckling failure mode at 258.3 pounds-force. Using a maximum load of 250 pounds-force, the resulting strength-to-weight ratio is computed as 3168.9.

PROBLEM STATEMENT

The primary objective of the project was to design a structure to be made out of balsa wood within the defined space envelope and to predict its ultimate load and failure mode using NASTRAN as the FEM tool.

The structure is loaded from the top by a 6-inch diameter circular plate (Figure 1). The space envelope is defined as such: a truncated cone is oriented so that its smaller end (4-inch diameter) is in-plane with the load plate, and the structure must exist between this cone and a larger cuboid. On all four sides of the cuboid are L-brackets mounted on the ground which the structure may touch. From the center of each of these brackets emanate half-round bars, running to the center until they intersect the bottom of the cone (Figure 1). The structure may not touch these half-round bars.

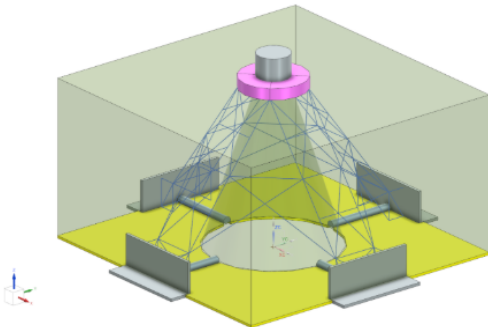


Figure 1 shows the CAD design of the testing apparatus envelope that the structure will go inside. Note the 6-inch pink plate where the load will be applied, and the half round elements that the structure is not allowed to touch.

or the FEM, all analysis must be done using the material properties provided and using beam cross-sections which are

able to be constructed using the balsa stock available. The maximum weight according to the computer analysis must not exceed 0.15 pounds-mass. It must have an ultimate load exceeding 10 pounds-force.

When building the real-world model, only the provided balsa wood and cyanoacrylate may be used. The size of individual stock members cannot be changed; they cannot be split, shaved, or sanded to change their cross-sectional area. Curved sections are forbidden.

COLLABORATION STRATEGY

Team communication was conducted primarily through text messaging, with MS Teams and Gmail used for exchanging CAD files, MATLAB scripts, and NASTRAN output data. At the outset of the project, each team member developed an independent structural design and performed the corresponding finite element analysis to evaluate strength to weight ratio, predicted failure mode, and overall feasibility. These results were summarized on individual comparison slides and evaluated collectively.

A Pugh matrix was used to compare the three concepts using four criteria: ease of assembly, sensitivity to variation in material properties, potential for further optimization, and initial strength to weight ratio. This structured comparison showed that the strongest aspects of the designs created by Jason and Sayes were complementary, leading to the decision to pursue a hybrid configuration that combined features from both.

During fabrication, the team adopted a flexible workflow in which members contributed to cutting, sanding, bracing, and assembling components as needed. No fixed/individual roles were assigned; rather, each team member fulfilled duties as they saw fit. This collaborative approach allowed efficient assembly of the physical structure while maintaining consistency with the modeled geometry.

DESIGN APPROACH/RESULTS

Setup

The initial design created by Sayes had a strength-to-weight ratio of 1260 (Shown in Annex A), which was the highest out of the group. It was designed such that it would contact each of the brackets so as to enforce boundary conditions which would maximize axial loading. The structure's weight was computed as 0.04983 pounds-force, which was the lightest out of the initial designs. Despite this, its failure mode was buckling, occurring at only 62.8 pounds-force. It was agreed upon that this was unsatisfactory and discussion ensued on how to change the structure so as to both increase the load at which it is predicted to fail while simultaneously increasing its strength-to-weight ratio.

Model Reduction

To optimize the structure, three metrics were constantly monitored over repeated design iterations: failure load due to stress, failure load due to buckling, and the strength-to-weight ratio. These metrics (along with others) were computed for each iteration using provided MATLAB scripts which processed the .DAT files generated by NX/NASTRAN.

For a given iteration, if the failure load due to stress was lower than that due to buckling, it was understood that those beams under highest stress needed to have their cross-sectional area increased, and those cross-braces responsible for preventing buckling might be worth reducing in size or eliminating altogether.

On the other hand, if the failure load due to buckling was lower than that due to stress, then we chose to mitigate this by using the following strategies: increasing the cross-sectional area of the offending beam, changing its orientation, (if it was rectangular) or adding more cross-bracing. The overall optimization process can be described as iterating over multiple designs until the failure load was as close to the maximum of 250 pounds-force as possible.

Buildability was a quality of constant discussion during this iterative design process. We elected not to use I-beams or box beams as we deemed these too difficult to construct. Stacked beams made using multiple plies of the 1/16 inch x 1/4 inch stock were modeled and used for spans under high stress and exhibiting the primary buckling mode. (Figure 3)

At the start of the design improvement process, it was agreed upon that ensuring the structure would contact the brackets would be of the utmost importance. Towards the final stages of developing our design, we chose to use “antennae” beams which would contact the bracket normal to its surface. During the simulation, we assigned SPC123 constraints to each of the nodes contacting the brackets. It is easily understood that the beam will not go through the bracket. In terms of the other two constraints, we assumed that friction between the beam and the bracket would be maximized. (within an assumed friction cone of 17°) This was verified using the simulation results, checking the nodal reaction forces at these contact points. The reaction forces in the directions in-line with the plane of a given bracket were always low in comparison to those generated in the normal direction at the nodes contacting the ground, thus our assumption was validated.

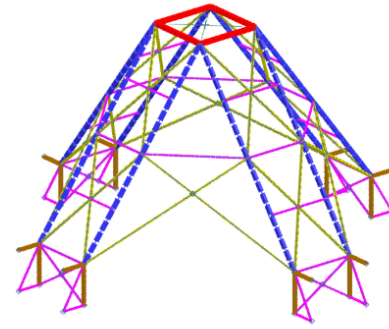


Figure 2 shows the color-coded mesh cross-sectional area. Refer to table 1 for exact area values.

DISCUSSION

The design philosophy was to create a load path dominated by axial forces while minimizing bending in all primary members. This approach was selected because axial loading provides a much higher strength-to-weight performance than bending for slender balsa beams. Throughout the optimization process, the principal design handles were the cross-sectional areas of individual members, (Table 1) the placement and removal of cross-bracing, and the use of stacked beams in regions carrying high compressive load. Each design iteration was evaluated by comparing failure load due to stress and failure load due to buckling. When stress governed, members exhibiting the highest axial stress were increased in area, and bracing members carrying insignificant load were reduced or removed to save mass. When buckling governed, corrective actions included increasing the area of the critical member, reorienting rectangular sections to maximize stiffness about the required axis, and adding intermediate bracing to shorten the effective column length.

This iterative approach ensured that material was concentrated only where structurally necessary, and that excess mass was removed wherever feasible. The combination of axial-focused load paths, strategic bracing, and selective beam thickening allowed the design to approach the highest ultimate load while ensuring a high strength-to-weight ratio. As a result, the optimization procedure directly shaped the final geometry and ensured that both mass constraints and load requirements were satisfied.

The structure is expected to fail by buckling. Using an applied load of 1 pound-force, the buckling analysis produced an eigenvalue multiplier corresponding to a critical buckling load of 258.3 pounds-force. (Figure 3) The stress-based failure load, obtained by scaling the 1 pound-force stress results, was higher than the required 250 pound-force limit. Since the failure load due to stress is higher than the failure load due to buckling, the problem is buckling-driven.

Solution 1 Result : 26d16055_sim1
 Subcase - Buckling Method, Mode 1, 268.34
 Displacement - Nodal Magnitude
 Min : 0.000, Max : 1.252, Units = in
 CSYS : Absolute Rectangular
 Deformation : 10% Model, Displacement - Nodal Magnitude

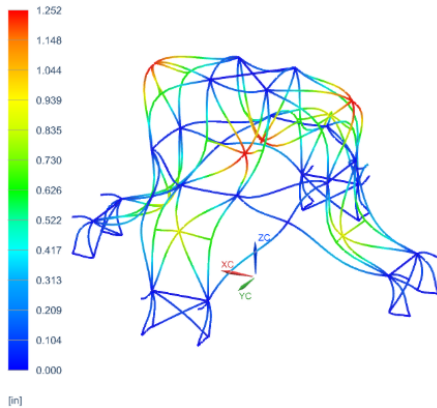


Figure 3: First buckling mode. Note how the Nastran FEA predicted the buckling of the cross-braced beam mid-height of the structure first.

Several procedures were performed to ensure that the NASTRAN model was accurate and reflected the physical structure as closely as possible. Element connectivity and beam orientations were checked repeatedly to eliminate unintended rigid body modes, and all SPC entries were reviewed multiple times to confirm that the applied boundary conditions matched the intended support configuration. The same balsa stock required by the course was used for construction, ensuring that the material properties in the simulation were representative of the actual components.

When building the cross-braced members, the first diagonal was glued at the required angle, and the second diagonal was cut in half and sanded carefully so that its two segments matched the exact lengths from the FEM. All members were sanded to match the modeled geometry in NX as closely as possible. The main beams (running directly from the top of the structure to its bottom) were sanded at their end at 53° to replicate the CAD geometry and to ensure that the top loading surface remained level. The stacked beams were assembled using the same cross-sectional combinations defined in the FEM. The only necessary modification was a minor adjustment of the overall structure angle (between approximately 1° and 3°) to account for a measured 0.2 in discrepancy between the real test apparatus brackets and the provided CAD model. This adjustment preserved alignment with the brackets while maintaining consistency with the intended load path. Taken together, these steps ensured that the physical structure conformed closely to the FEM assumptions and that the simulation results could be taken confidently.

The simulated model assumes idealized conditions, perfectly straight beams, exact cross-sectional dimensions, and joints with no added stiffness or mass. In the physical model, a significant amount of glue is required not only to form the joints but also to laminate multiple pieces of stock into larger composite beams. For example, three $1/4$ inch x $1/16$ inch members were combined to create the $1/4$ inch x $3/16$ inch main beams, and four of the same stock beams were stacked to form

the beams making up the square frame at the top. These stackings introduce additional adhesive mass that is not accounted for in the FEM, increasing the overall weight of the real structure relative to the simulation. The cured adhesive also increases joint and member stiffness, causing the physical structure to behave less flexibly than the beam-element model predicts. Variability in the actual material properties of the balsa stock further contributes to differences between simulated and experimental results. If the real material exhibits a lower ultimate stress than assumed, buckling and failure will occur at loads below the predicted value. If the actual value is higher, the structure may sustain greater loads before failure. Small geometric imperfections and slight misalignments introduced during construction generally reduce buckling capacity, so the experimental failure load is expected to fall below the simulation prediction.

The boundary conditions will be demonstrated by one of the four feet contacting ground and the bracket because of the symmetry of the structure. In figure 4, point 1, 2, 3, and 4 are touching the bracket, point 5 and 6 contact the ground. We constrained point 1, 2, 3, 4 in x, y and z direction, and point 5 and 6 are constrained in z direction. Because beam 1, 2, 3, 4 are perpendicular to the bracket, the axial force of the beams is also perpendicular to the bracket which gives the highest normal forces resulting high friction. Thus, we assumed that point 1 and 2 are specified in the x and z direction. By showing the nodal reaction forces, we confirmed that there are no forces in the z and x direction with our specifications, and the only reaction force is in the y direction which is constrained by a fixed bracket, so the assumption is valid. For point 3 and 4, they are already constrained in y and z direction because they are touching the ground and bracket. By the nodal reaction plot, there is no x direction forces shown at point 3 and 4, so we can also constrain it in the x direction. For point 5 and 6, they are only constrained in the z direction because of the contact with ground. We cannot constrain these points in the x and y direction because the cross-bracing beams are over 17 degrees which might cause sliding, and the deformation in beam 3 and 4 would also allow moving in the y direction. However, the nodal reaction forces plot showed that there are no forces in the x and y direction for point 5 and 6, so it is allowed to specify them in the x and y direction. In order to simulate the real situation, we decide not to constrain them in the x and y direction, but the simulation results are the same with the two boundary conditions.

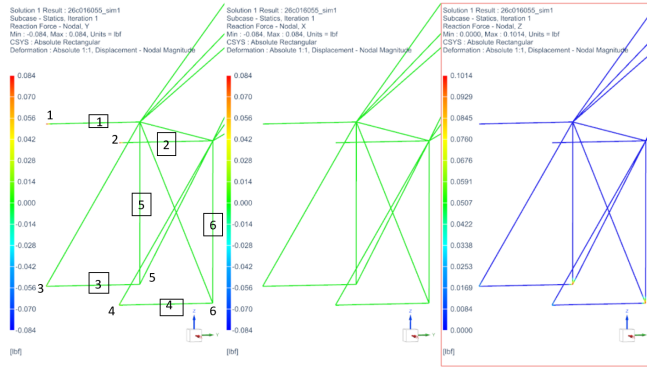


Figure 4: Nodal Reaction Forces. The views are synchronized.

ACKNOWLEDGMENTS

We want to acknowledge all the help and efforts we received from Professor Christopher Muir, Teaching assistants Kyle Christensen and Angel Bermudez, Bill Mildenger, and all the technical staff that helped us optimize the design and build the structure.

TABLES

Color	Cross-Sectional Area (in. x in.)
Red	1/4 x 1/4
Blue	3/16 x 3/16
Yellow	1/8 x 1/8
Pink	3/32 x 3/32
Brown	1/8 x 1/4

Table 1 shows the cross-sectional areas corresponding to the colored beams.

P _{CR} (lbf)	P _{ULT} (lbf)	P _{FAILURE} (lbf)	Weight (lbf)	S/W (actual)	S/W (used)
282.805	258.34	258.34	0.0788	3280	3173

Table 2 shows the calculation for the strength-to-weight ratio using both the actual predicted failure load and a maximum specified load of 250 pounds-force.

INITIAL DESIGN ONE-PAGER (BEFORE OPTIMIZATION)

Project 04 – Design 1

Abdulwahab Sayes

S/W = 1260

Sol105 – Linear Buckling

Load = 1 lb in -Z
Mass = 1.2889e-04 lbf-s²/in
MinStress = -10.954 psi
MaxStress = 9.0568 psi
firstEignevalue = 62.8171

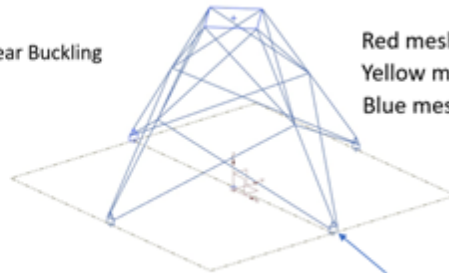
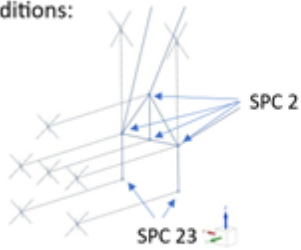
Pf = 91.2883 lbf

Pcr = 62.81719 lbf

Weight = 4.983e-02 lbm

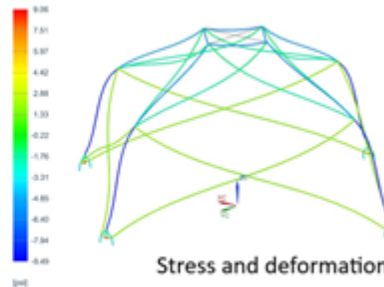
$$S_f/W = 1260.633$$

Example Boundary conditions:

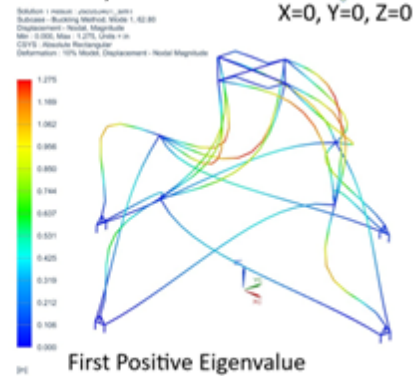
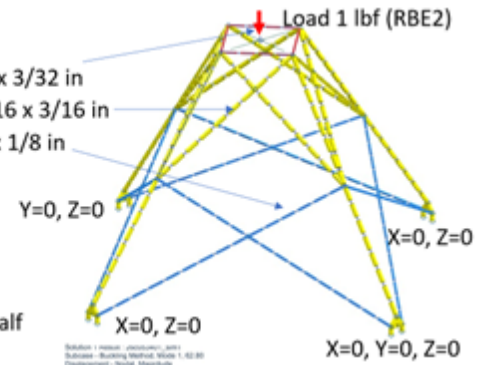


Not touching the half round element.

Solution 1 Result: 26x55x601_vant
Subcase = Static, Iteration 1
Stress = Element-Nodal, Unaveraged, 11
Beam Section = Maximum
Min.: -0.49, Max.: 0.08, Units = psi
CIVS: Nodal
Deformation: 10% Model, Displacement-Nodal Magnitude

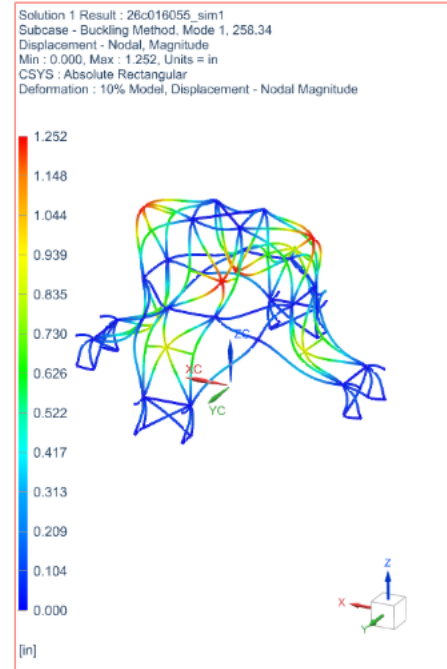
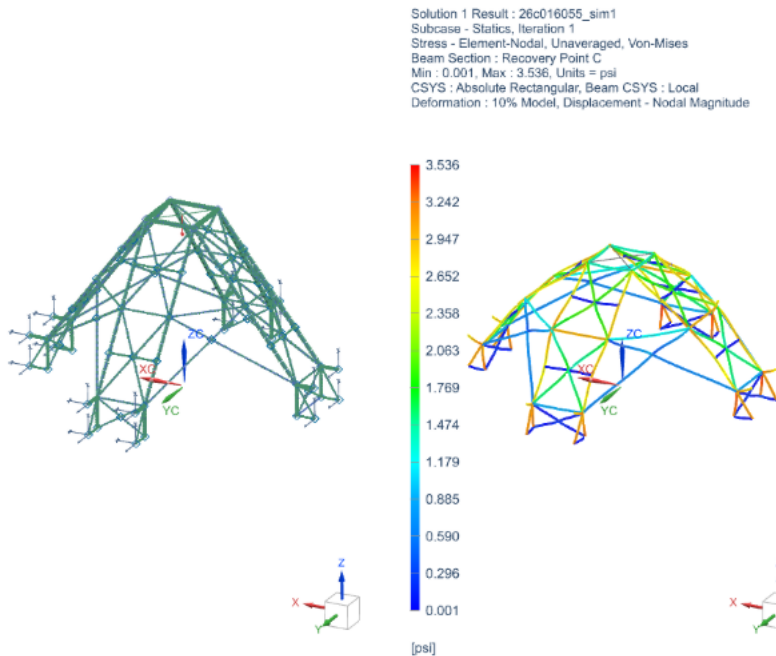


Stress and deformation



First Positive Eigenvalue

FINAL DESIGN (AFTER OPTIMIZATION)



FINAL BALSA MODEL

