Nuclear Crane Support Structure Analysis

Jimmy Pare November 27, 2025

1 Summary

Finite element analysis and hand calculations determine that the maximum load the crane can safely lift is **50.3 kip**, assuming rigid connections between beams. This capacity is governed by the deflection limitation of 2 inches rather than material failure. Further information regarding the connection type between beams is required in order to verify assumptions.

2 Assumptions

- Rigid connections between beams.
- Sufficient concrete support at the base of the structure.
- The crane is sufficiently strong and will not fail before the structure.
- The crane is modeled as a distributed load of width 40 inches.
- Static conditions, negligible accelerations.
- A572 Grade 50 Steel material properties.

3 FEA Model Information

3.1 Material and Geometry

ASTM A572 Grade 50 Steel was used as the material of the W18x119 beams for analysis. The assumed material properties were $E = 29,000$ ksi, $\nu = .3$, and $\rho = .283$ lbm/in^3. Throughout this report, the longest beams are labeled as members 1, the uprights as members 2, and the shortest as members 3.

3.2 Mesh

A second order beam element type was used in this Abaqus simulation. Beam element types excel at capturing large displacements while cutting back drastically on computational cost compared to a solid 3D deformable model.

128 B32 elements were used to model the supporting structure. A mesh convergence study was used to verify the number of elements was sufficient.

3.3 Boundary Conditions and Interaction

The base of each vertical member was fixed without displacement or rotation.

Each member was connected together using a tie constraint, meaning that all connections were rigid. This is the largest assumption in the model because steel structures typically fail at connection points. More information about the type of connection at joints is required to improve model accuracy.

3.4 Loading and Step

A distributed load was applied to the middle 40 inches of members 1. The nonlinear geometry feature was used in the static analysis step type.

4 FEA Results

4.1 Bending Moment and Deflection Plots Before Failure

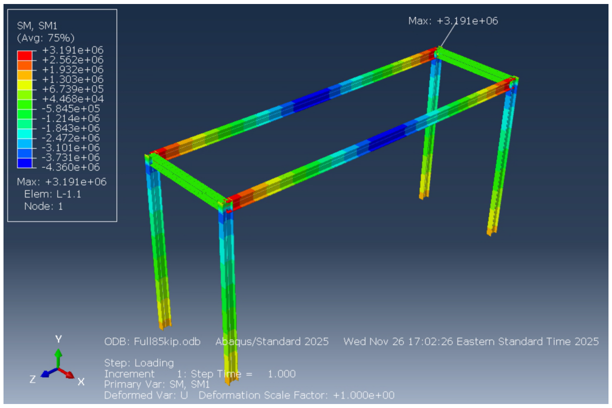

Figure 1: Bending moment contour plot with a maximum of $M_{max} = 3.191 \times 10^6$ lbf in.

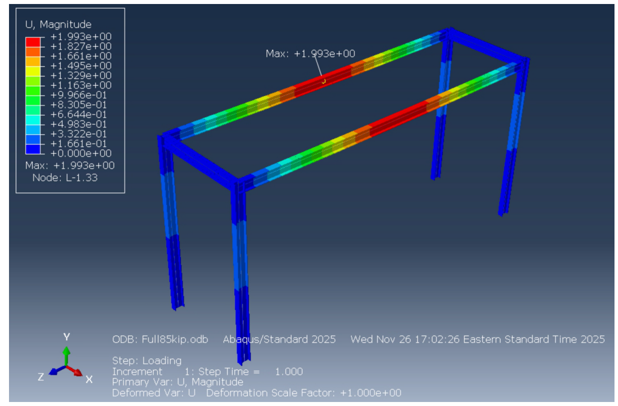

Figure 2: Deflection contour plot with a maximum of $\delta_{max} = 1.993$ in.

4.2 Maximum Payload

The following graph was generated in excel by plotting the maximum von Mises stress and deflection of the model across a range of applied loads.

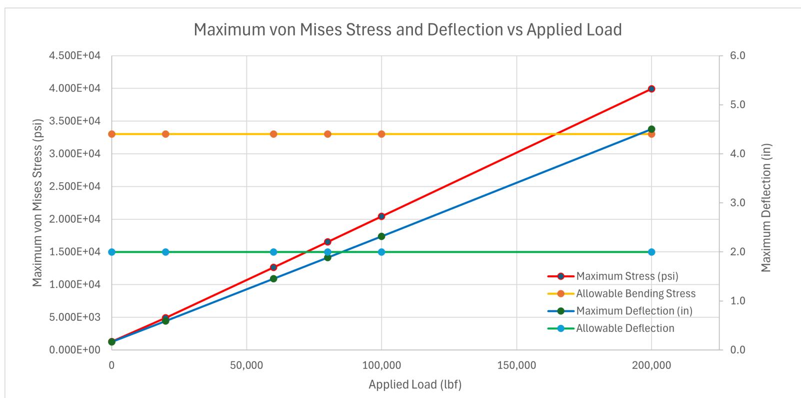

Figure 3: Stress and deflection plotted against force.

From Figure 3, the maximum beam deflection reaches 2 inches before the maximum bending stress reaches 33 ksi. The load at which this occurs was linearly interpolated as $P_{FEA} = 85.3$ ksi. The weight of the crane was subtracted from this number to give the maximum weight the structure can lift.

$$W_{\text{FEA}} = P_{\text{FEA}} - W_{\text{crane}} = 85.3 - 35 = \mathbf{50.3\ kip}$$

5 Hand Calculations

The reaction between members 1 and members 2 is neither a pin connection nor a fixed connection. Both types of reactions were considered and averaged to give a semi-rigid approximation.

Pin-Pin Support

$$P_{\text{pin}} = \frac{48EI\delta}{L^3} = \frac{48 \times 29e3 \times 2190 \times 2}{662^3} = 21.01 \text{ kip}$$

Fixed-Fixed Support

$$P_{\text{fixed}} = \frac{192EI\delta}{L^3} = \frac{192 \times 29e3 \times 2190 \times 2}{662^3} = 84.06 \text{ kip}$$

$$P_{\text{hand calc}} = 2 \text{ sides} \times \frac{21.01 + 84.06}{2} = 105.07 \text{ kip}$$

FEA predicted a maximum weight of 85.3 kip could be applied to the model before deflecting 2 inches. The hand calculations and FEA differ by **18.8%**, indicating that the semi-rigid approximation does not align with the boundary conditions in the simulation. The source of this discrepancy is likely the difference in rotational stiffness at the joints between models. Further information about the connection type would prove useful for future studies.