FERMILAB-PUB-24-0414-STUDENT

LINAC Electrical Cabinet - Structural Analysis

Efren Blas

Advisor: Kathrine Laureto

Accelerator Division, Fermilab, Batavia IL 60510

August 2024

FERMILAB-PUB-24-0414-STUDENT

Abstract

Along LINAC's downstream path is a floor containing various cabinets above the LINAC. Of the dozens of cabinets are 4 specific cabinet types: PFN (pulse-forming network) Module, PFN Power Supply, Water Skid, and Marx. Each of these cabinets may contain some of the various contents that include electronics, wiring, circuits, and conduits. In addition, high voltage and low conductivity water (LCW) piping are also present within the cabinets. Occasionally, Lab Technicians perform maintenance and inspections of the cabinets and need to access every part, including the top section that is subject to the weight of the technician standing on top. To ensure the cabinet is structurally safe, a CAD model was developed for each cabinet and a Finite Element Analysis (FEA) was performed on each cabinet to confirm it is safe to stand on.

1. Introduction

The accelerator complex houses the LINAC linear accelerator (Figure 1) along with dozens of cabinets that follow along downstream the accelerator path. There are four specific cabinet types that are to be analyzed using the ANSYS structural analysis to determine how much deformation and stress each cabinet experiences from a specific weight. Each of the cabinets are modeled using Siemens NX to create the 3D models that are used to perform the ANSYS analysis. The results of the analysis will provide further insight into the behavior of the cabinet structure under the weight of a person and whether it is safe for an activity to be performed on top of the cabinet.

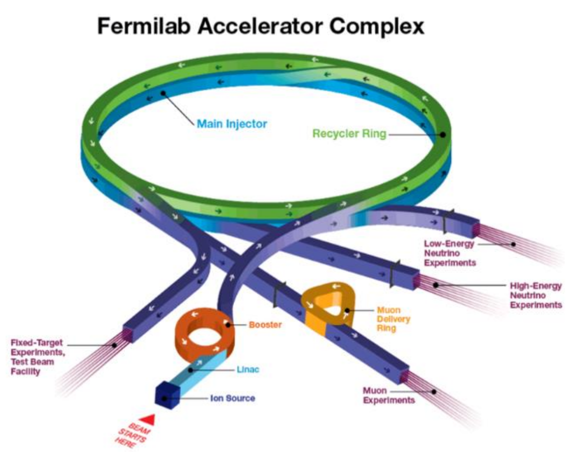

Figure 1. FNAL accelerator complex

2. Measurements

2.1 Panels and Door

To design the cabinets for testing, the CAD software Siemens NX was used to create each of the components from a 2D sketch to a 3D model. The first step of the process began with measurements of the cabinets onsite with the use of a tape measure to measure the total length, width and depth of each panel and door. All the panels and doors on the PFN cabinets and water skid were measured with a depth of ⅞” and ¾”, respectively, and a thickness of 1/16” throughout the body. The panels and doors were attached to rest on the exterior of the frame, leaving them exposed to measure their depths. As for Marx cabinet, the panels and doors were assembled flush with the frame with no exposed body like the other cabinets. However, each of the doors and side panels had cutouts for windows and vents leaving the clear view of the thickness to be 1/16” like the other cabinets. The Marx cabinet couldn't be directly measured but further inspection inferred a ~2” depth.

2.2 Frame

Measuring the frame of the cabinet followed a similar process as the doors and panels but required an inspection of the interior to note additional geometry. From the exterior, the top and bottom faces were noted to be connected to each other with 4 corner columns for all the cabinets. The water skid and PFN cabinets had an additional column between each of the corner columns on the sides where doors were present and the sides that had 2 separate panels. Special cases with the frame were present in the PFN module and Marx, which were 2 identical frames connected via bolts and welds; only the module possessed the additional column between 2 corner columns. The inspection of the PFN cabinets and water skid noted beams and welds along the top edges on the sides where doors weren’t attached and welds at the bottom edge only. In the case of Marx, both the top and bottom edges had beams and were welded where they contacted the columns. Another special case is the module which had a u-beam running from side to side to support the top face. The last notable feature to the cabinets except for Marx was a lip extended out ⅝” of the cutout edge for the doors and panels. All the cabinets had a frame ⅛” thick throughout.

2.3 CAD Drawings

Additional dimensions were referenced from older CAD drawings provided by Kathrine Laureto. The drawings provided the length and width dimensions for all the doors and panels used in the design phase. The measured dimensions of the frame along with the panels and doors were used to determine the cutout dimension on each of the frame sides. The only cabinet to not have a CAD drawing is the water skid; however, its dimensions were attained by tape measure and inspections of the interior.

3. Design Phase

3.1 Panel

3D Modeling in NX for each component followed the same process starting from a 2D sketch which consisted of a rectangle using the length and width dimensions obtained from a combination of onsite measurements and old CAD drawings. The extrude command made the sketch into a 3D shape (giving the 2D shape depth) that would allow for the use of the shell command to give the component its thickness. To model the panel, a sketch, extrude, and shell command in the given order was enough to model the real-life panels. Some needed a small cut out on the top and bottom of the panels side to allow space for the lip on the frame.

3.2 Door

The doors use the same process but are accompanied by additional extrude commands to create and extend pieces of the panel. There are 2 main types of doors: left and right, that have their own distinct design. The left door has a solid flap on the right side that extends out ⅞" from the side and acts as a stopper for the right door when closing (Figure 2). As for the right door, its left side has a flap extending into the door forming a U-shape that can rest on the left door flap (Figure 3). Other doors, mainly on the power supply, consisted of a panel with a door handle. The door handles weren't included in the model due to their geometry not contributing to the structural integrity of the whole cabinet and serving only as a mechanism.

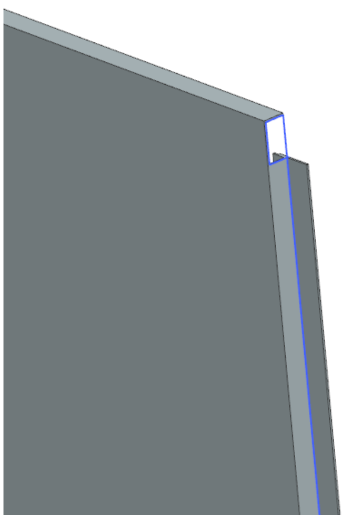

Figure 2. Left-Hand door for PFN power supply front side

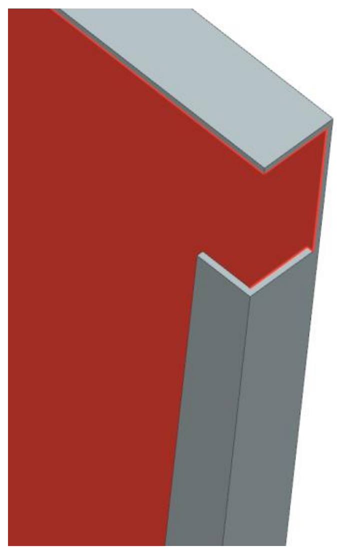

Figure 3. Right-Hand door for PFN power supply front side

3.3 Frame

With the frame, the same process of sketch, extrude, and shell commands are used with extruding. The extrudes create the beam along the interior edge of the frame, subtract large sections off the sides to create the cutouts between the columns and then add the lips on those column edges. The beams are made within the frame itself to eliminate the need to create a separate component that must model the welds during the assembly process (Figure 4). By making the beams within the frame, it simulates the welded beam on the frame as a one whole piece.

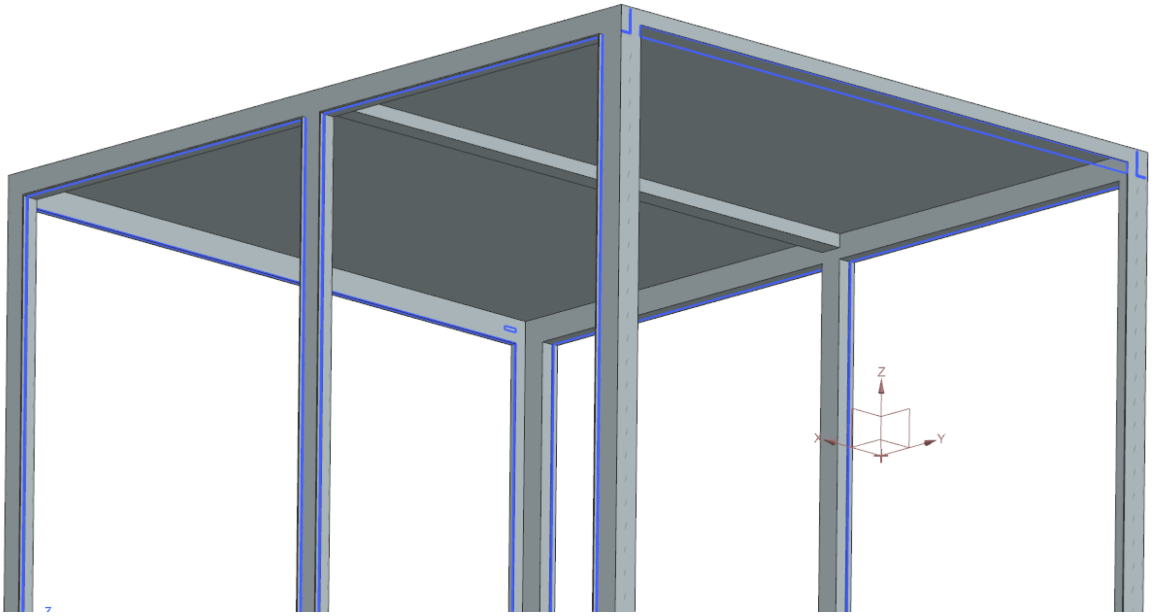

Figure 4. PFN module frame, beams are located on both sides of the frame where 3 columns are present, the beam running across the frame horizontally supports the top section from the middle and connects to the middle columns.

3.4 Assembly

Of the four cabinet types, two were assembled, that being the PFN module and power supply (Figure 5). The assembly of the cabinets begins with the frame, from there, each of the panels and doors are attached to the frame using 'constraints'. The touch-align and distance constraints were enough to center and attach the doors and panels onto the frame. A special case during the assembly was the PFN power supply's inner panels positioning inside the of the frame (Figure 6) which had to be referenced from the CAD drawings to input the correct values for the distance constraints.

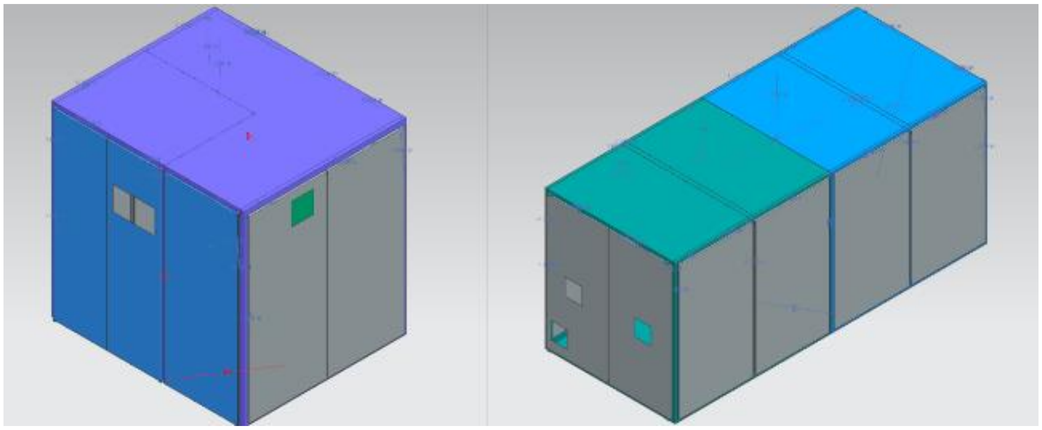

Figure 5. PFN power supply assembly (left), PFN module assembly (right)

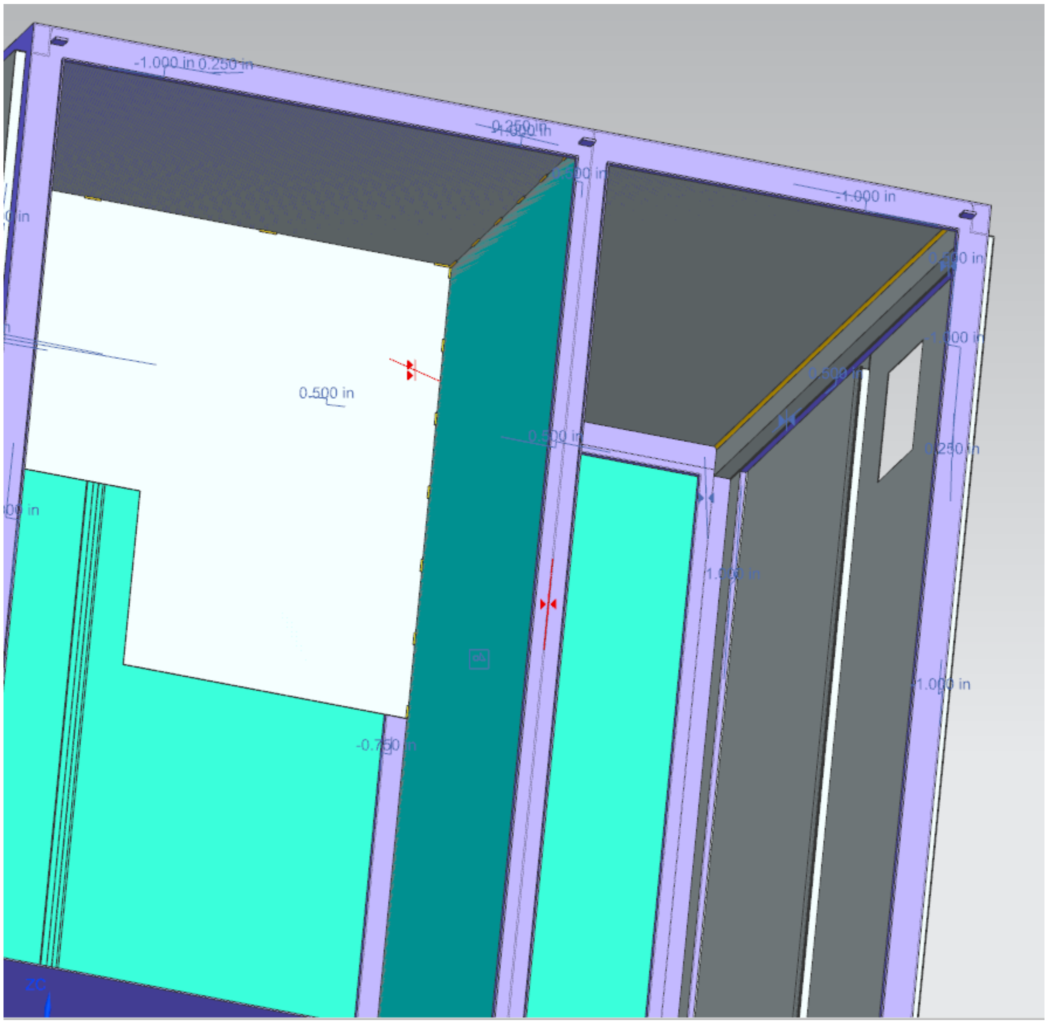

Figure 6. PFN power supply assembly, the inner panels (light grey and dark turquoise), dimensions align inner panel with middle column

4. Test Phase (Set Up)

The FEA performed on the assemblies was an ANSYS Workbench 'static structural' with 7 steps to complete. The first step was choosing which analysis to do. In the second step, the material needs to be specified from an engineering materials library in the software. By default, structural steel is applied in the start, but other materials can be selected; however, the cabinets are made entirely of structural steel, therefore the default option completes the second step.

The third step involves the modeling or 'geometry' of the part within the software. Since the models were completed and assembled on NX, the assemblies were imported into the ANSYS design modeler. While the assembly is in the design modeler, the split face command is used to divide the top face of the cabinet into multiple faces that could be individually selected later when applying a force. A rectangular face (18" x 18") was made using the command to closely represent a person standing within a certain section on the top where it remains largely unsupported.

The remainder of the steps take place in the analysis file where adjustments and additional set up options must be made. For both the PFN cabinets, any welds made in NX were suppressed in ANSYS due to meshing complications that would result in inaccurately high results. Next part was to apply the mesh which is generated on each face in the assembly. The mesh comprises an entire web of shapes that have an 'element' size (Figure 7) that can be changed to a lower number for more accurate results. In this case, both cabinets had a 1" element size for the mesh. After the mesh, certain conditions are applied to the cabinets before running the simulation. In total, 3 conditions are applied: Standard Earth Gravity, Fixed Support, and Force. The standard earth gravity condition is applied to the cabinet where each component experiences $9.8/s^2$ of gravitational force. The bottom face where the cabinet rests on the floor is set as the fixed support. A force of 400 lb pointing down is set on the rectangular face (Figure 8) made using the split command. The final part of the setup is to specify what solutions are needed. For a structural analysis, the total deformation and equivalent stress are needed to see how much stress and warpage the cabinets will experience.

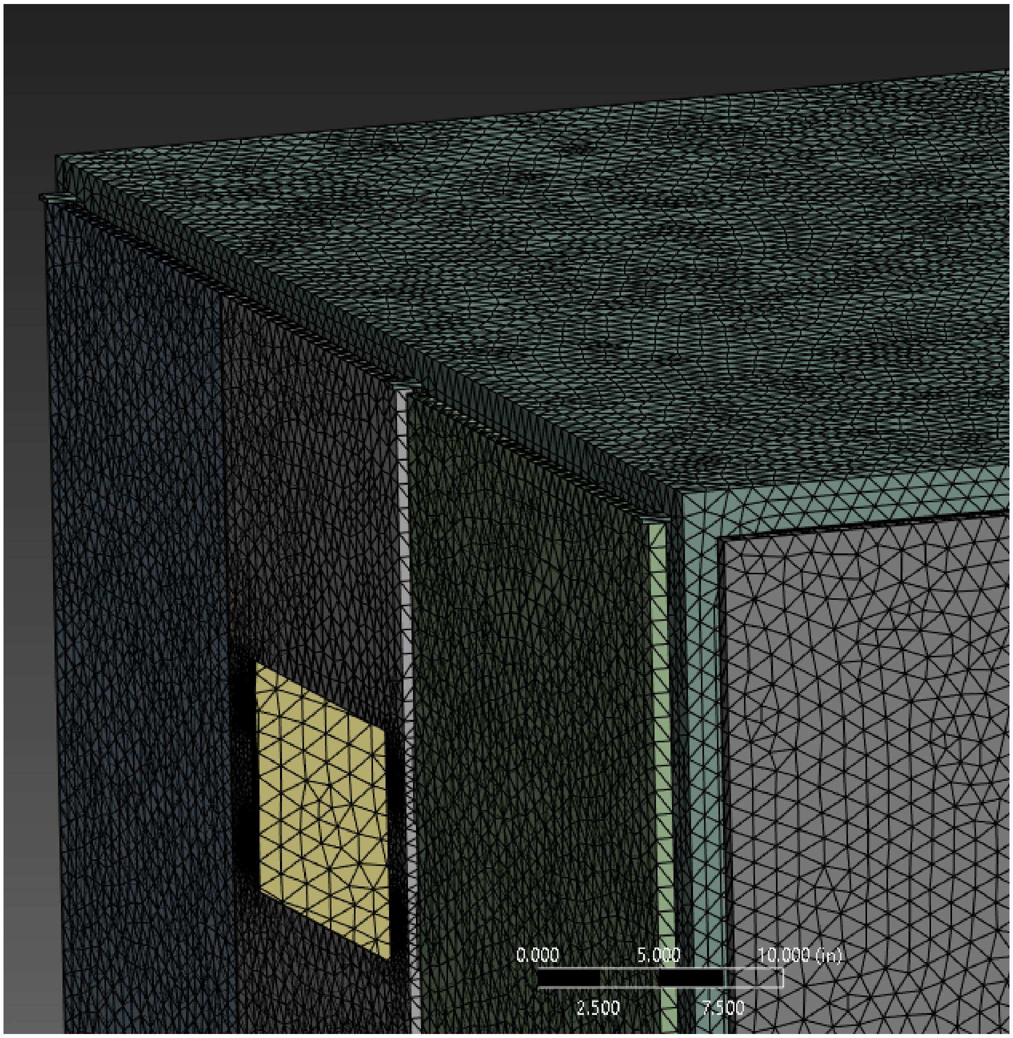

Figure 7. A mesh (characterized by a web of triangles) on the PFN power supply cabinet

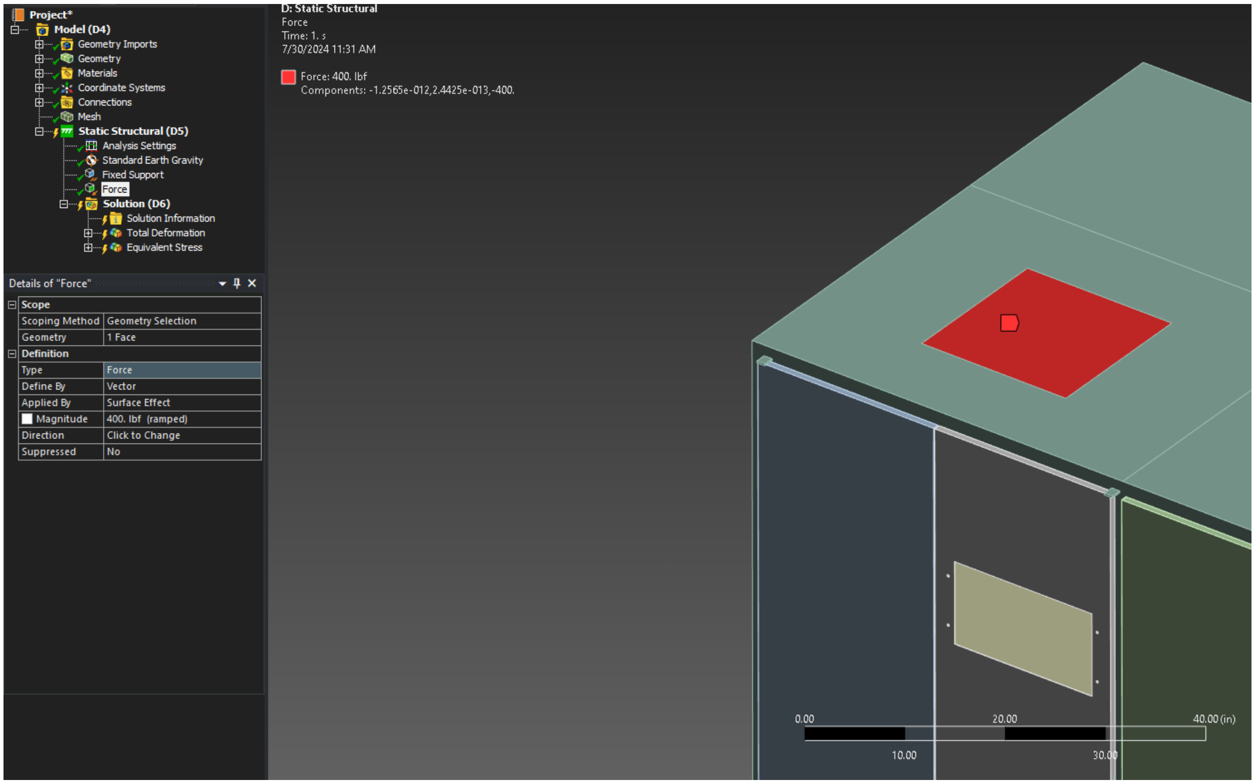

Figure 8. Red square is the section where the force is applied

5. ANSYS Results

5.1 Total Deformation

With the total deformation, it can be observed that the force applied on the square section for both cabinets causes a miniscule max deformation that does not exceed 0.25". For the power supply, the inner panels positioning at approximately the center provides excellent support to the top section that doesn't have beams across the middle section like the module. As a result, the max deformation of 0.098” (Figure 9) remained well below to be anything significant towards the cabinet. Another observation to be noted is the behavior of the sides and panels which remain largely unaffected from the force, showing nearly no sign of bending.

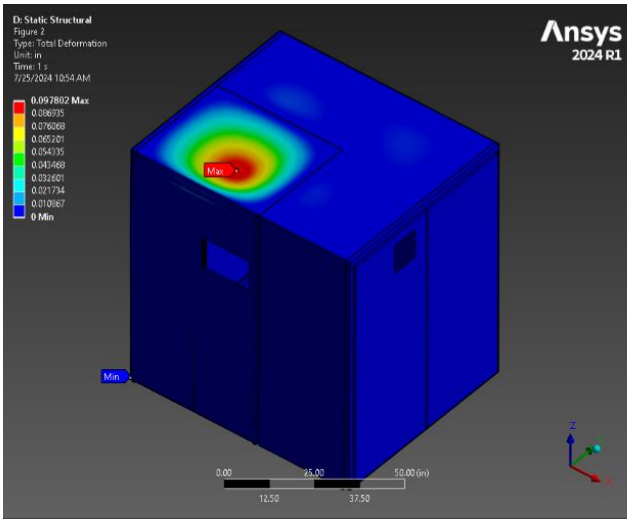

Figure 9. Total deformation result for PFN power supply

As for the module, its max deformation comes in higher than the power supply at 0.19174", about a 0.1" difference. Unlike the power supply which had the benefit of an inner panel that connected from the top to bottom, the module has beams spanning the shortest length (side to side) to support the top section. Additionally, given the nature of the module being two identical frames assembled as one large cabinet, the extended lip from the frames end opposite to doors acts as another support similar to the beams. While the max deformation is higher than the power supply, the area it was applied was closest to the center between the beam and lip supports (Figure 10) which may be the weakest position if comparing to the area between the beam and doors. Like the power supply, the behavior of the sides and columns don't visibly appear to show any bending.

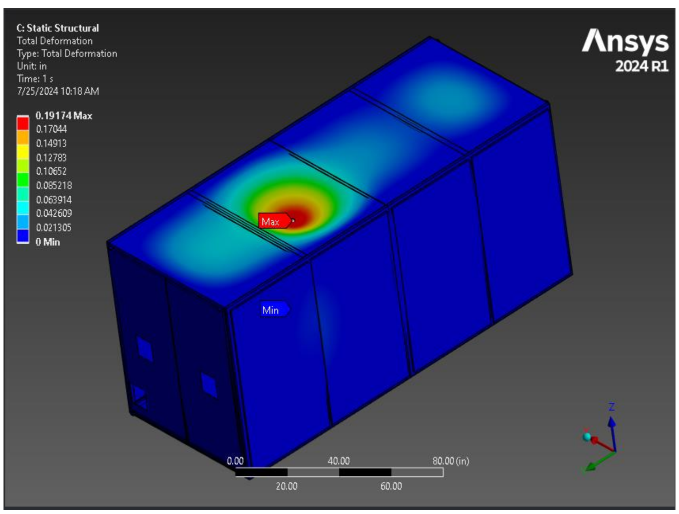

Figure 10. Total deformation result for PFN module

5.2 Equivalent Stress

One of the properties of a solid material that is considered is the yield strength, which can be taken in a tensile and compressive yield strengths. For structural steel, the yield strength is 36259 psi for both. For the cabinets, ensuring the max stress < yield strength is an important factor in analyzing the integrity of the structure. If the max stress is higher than the yield strength, permanent deformation will result on the structure, in the given location that it occurs.

With the power supply, it can be noted that the area where the square section experiences ~2500 psi up to ~4000 psi in some spots (Figure 11). In addition, the 2000 psi range is concentrated on the entire square section and follows a diagonal path to the four corners where the section of the top is supported by the inner panels and frame edges. The edge where the beam is connected to also experiences the 2000 psi trend and higher where the beam connects to the frame's interior. However, the max stress is not in the square section of the cabinet, instead, the corner where the two inner panels and top section connect experiences the max stress (Figure 12). Another thing to note is that the max stress is a small concentration within the corner that spreads out ~0.5" before reaching the 500 psi region.

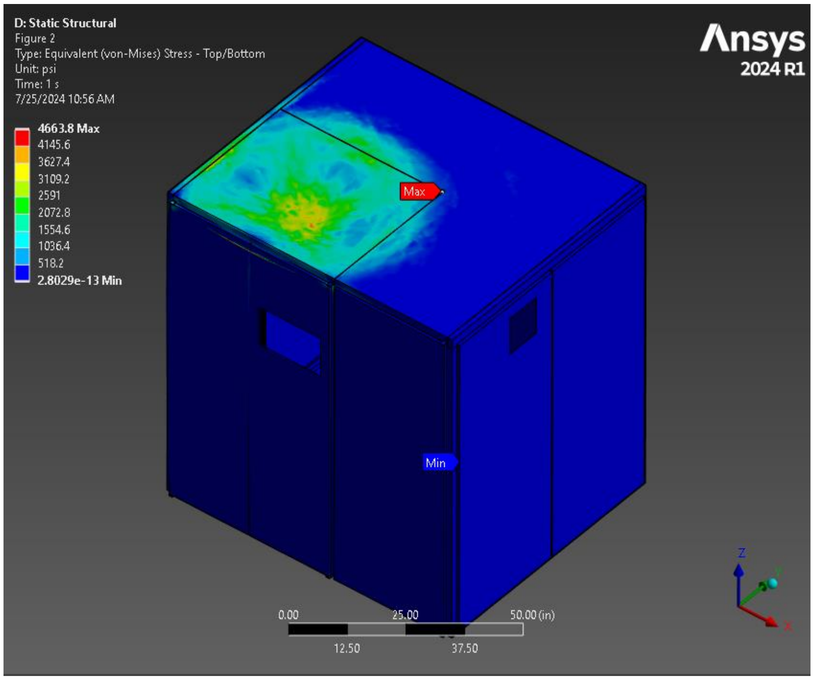

Figure 11. Equivalent stress result for PFN power supply

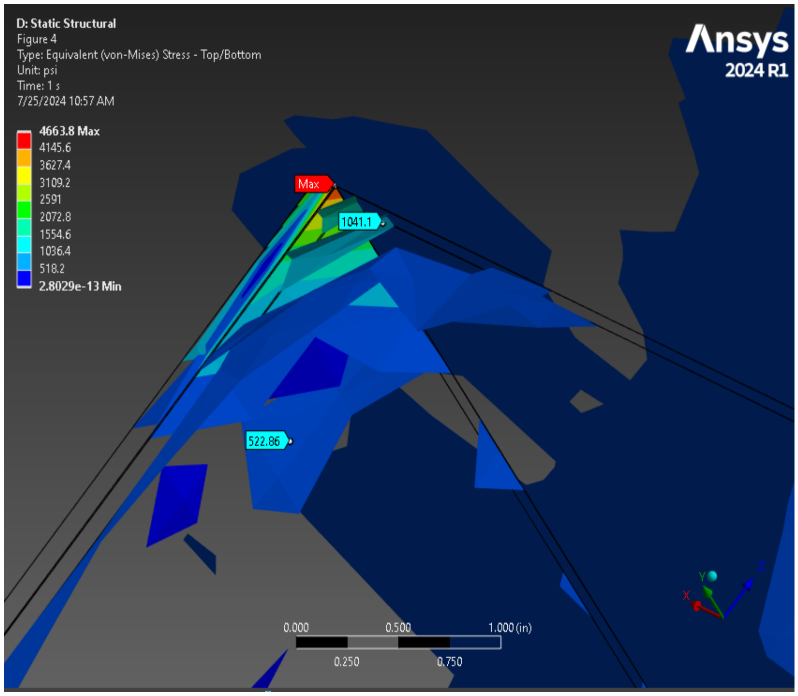

Figure 12. Close up of the stress concentration where the inner panels and top section meet

On the module, the beams dividing the top face into four sections managed to contain the stress within the section where the force was applied. Within the square section, most of the stress experienced lies between the 5000 psi and 3000 psi range while some scattering of 2000 psi is present within the section (Figure 13). The beam closest to the doors experiences a similar stress range comparable to the square section throughout most of its length but concentrated in the middle. As for the max stress, the center beam made by the lips of both frames has a stress concentration near the corner with the side wall (Figure 14). Most of the stress spans outward along the beam for ~4” from its max to 1500 psi but is also mimicked on the opposite end of the beam as well. The larger side beams that run against the sides of the frame in both vertical and horizontal directions have stress of ~1500 psi with a higher concentration near the frame's lip.

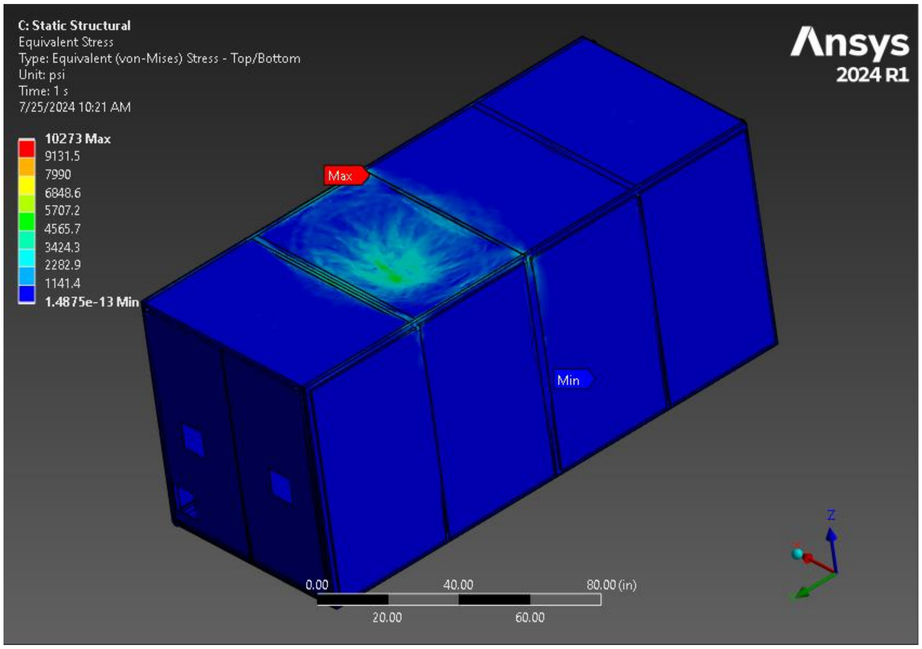

Figure 13. Equivalent stress results for PFN module

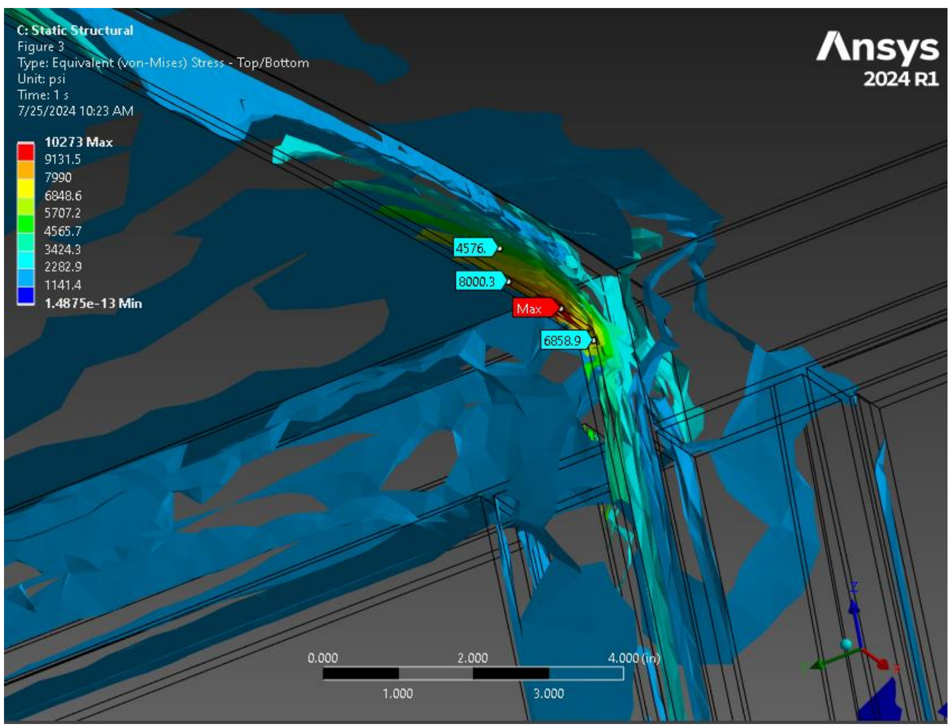

Figure 14. Close up of the stress concentration on the middle beam of the cabinet where the frames connect

6. Conclusion

Overall, the current design of the cabinets analyzed in ANSYS has demonstrated the capacity to withstand the load of a person on areas with minimal support. The max deformation results are extremely minimal, no noticeable deformation can be observed or cause any concern over the cabinets structure. As for the max stress values, both values remain well below the yield strength to cause permanent deformation that could impact a person's safety while on top of the cabinets. Given the area that needs to be supported by the cabinets, an inference can be made that the other cabinet, Marx and the water skid, may also be able to support the same weight application. For the future, further analysis needs to be performed and discussions for a fall protection plan will entail potential ways to install safety devices around or near the cabinets to address the height hazard.

7. References

[1] ANSYS, Inc., ANSYS Mechanical (Version 2024), ANSYS, Inc., 2024

[2] Siemens Digital Industries Software, Siemens NX (Version 2206), Siemens Digital Industries Software, 2022.

Acknowledgement

- This manuscript has been authored by Fermi Research Alliance, LLC under Contract No. DE-AC02-07CH11359 with the U.S. Department of Energy, Office of Science, Office of High Energy Physics.
- This work was supported in part by the U.S. Department of Energy, Office of Science, Office of Workforce Development for Teachers and Scientists (WDTS) under the Community College Internship (CCI)