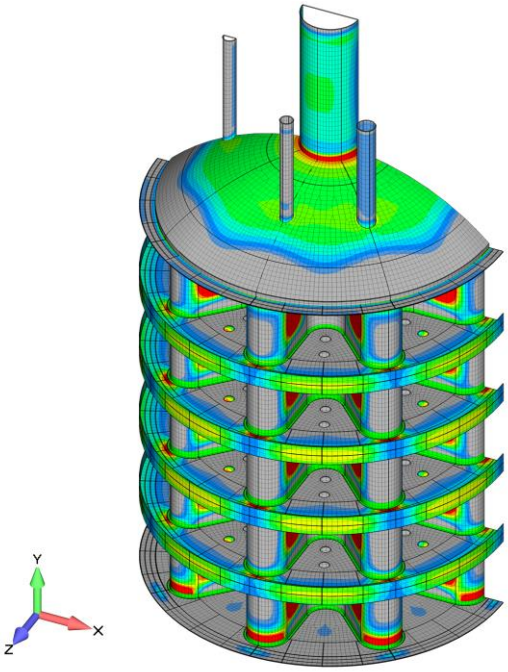
Keywords: Femap, NX Nastran, Pressure Vessel Analysis, ASME Pressure Vessel, ASME Section VIII Division 2, FEA, FEA Consulting, cryogenic

Main Graphic:



Pressure Vessel Analysis

Case Study Section: FEA | ASME Pressure Vessels

Analysis Type: Lifting, ASME Section VIII Div. 2, Linear Static, Stress

Optimization of Cryogenic Pressure Vessel for Rapid Heat Transfer Analysis

It is common for pressure vessels to have non-standard geometry that requires ASME Section VIII, Division 2 analysis. For example, maybe a cone requires an angle that exceeds standard code limits. General sizing of the member can be determined using design guidelines and the component can be shown to meet code allowables using FEA.

What if the majority of the components of the vessel are radically different from the standard components of and ASME Section VIII vessel? How is initial section sizing determined? For the high-pressure vessel featured in this case study, FEA was an integral tool in finding a starting point for component sizing and geometry as well as certifying the final design.

The initial vessel design was driven by heat transfer requirements (see Figure 1). Maximum surface area and minimum wall thickness was the ultimate goal. The initial design was modeled with sheet-solid geometry and meshed with 4-node plate elements. Since the model was efficient and the analysis ran quickly, multiple iterations were performed and the section thicknesses were updated until a workable design was achieved.

Once a workable design had been achieved, there was freedom to explore different shell shapes, stiffener geometries and flange thicknesses. The vessel evolved from something that was simply "workable" to something with optimal wall thicknesses and surface area that met the thermal requirements of the project (see Figure 2).

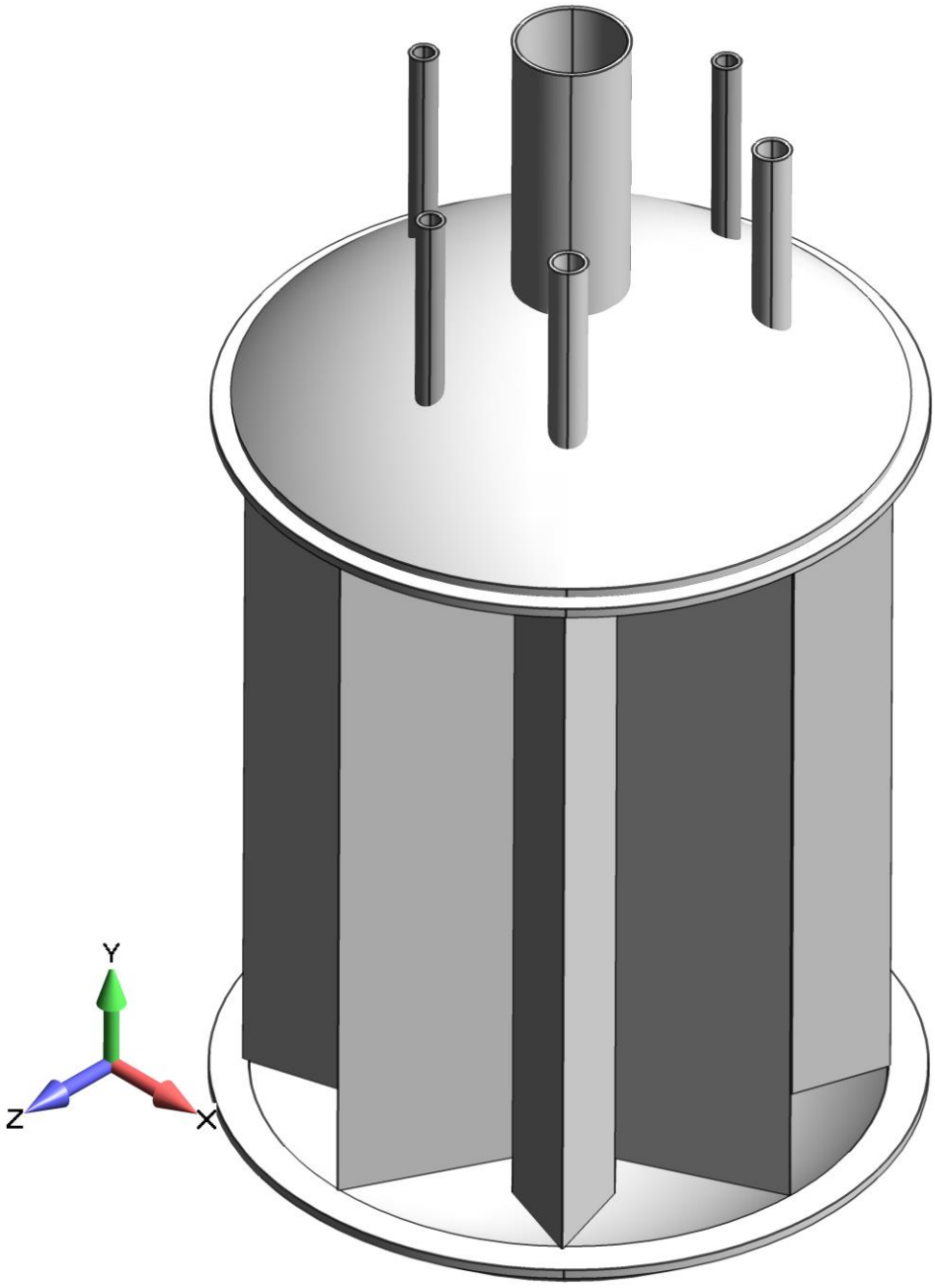


Figure 1: Original vessel design CAD geometry.

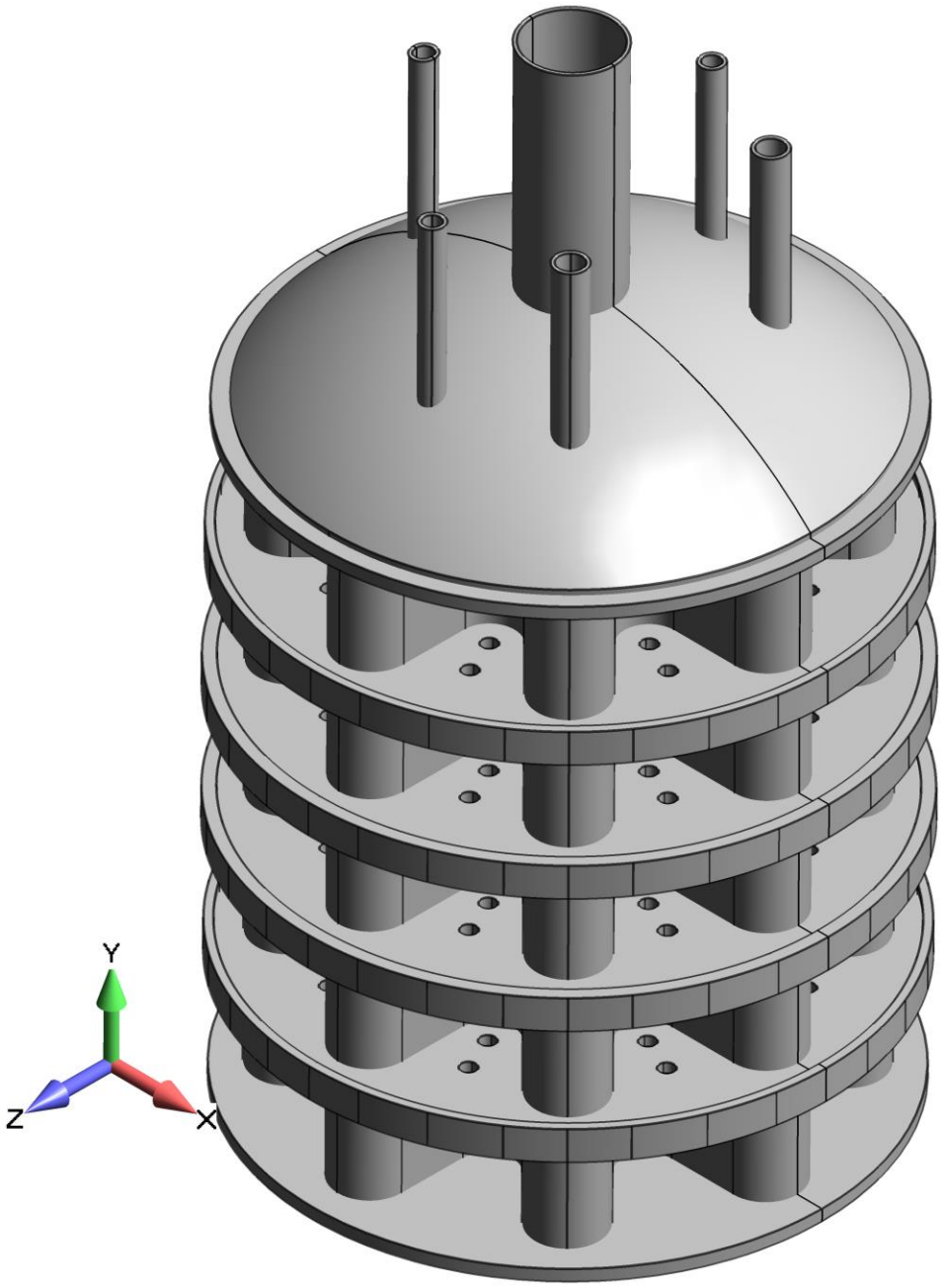


Figure 2: Final vessel design CAD geometry.

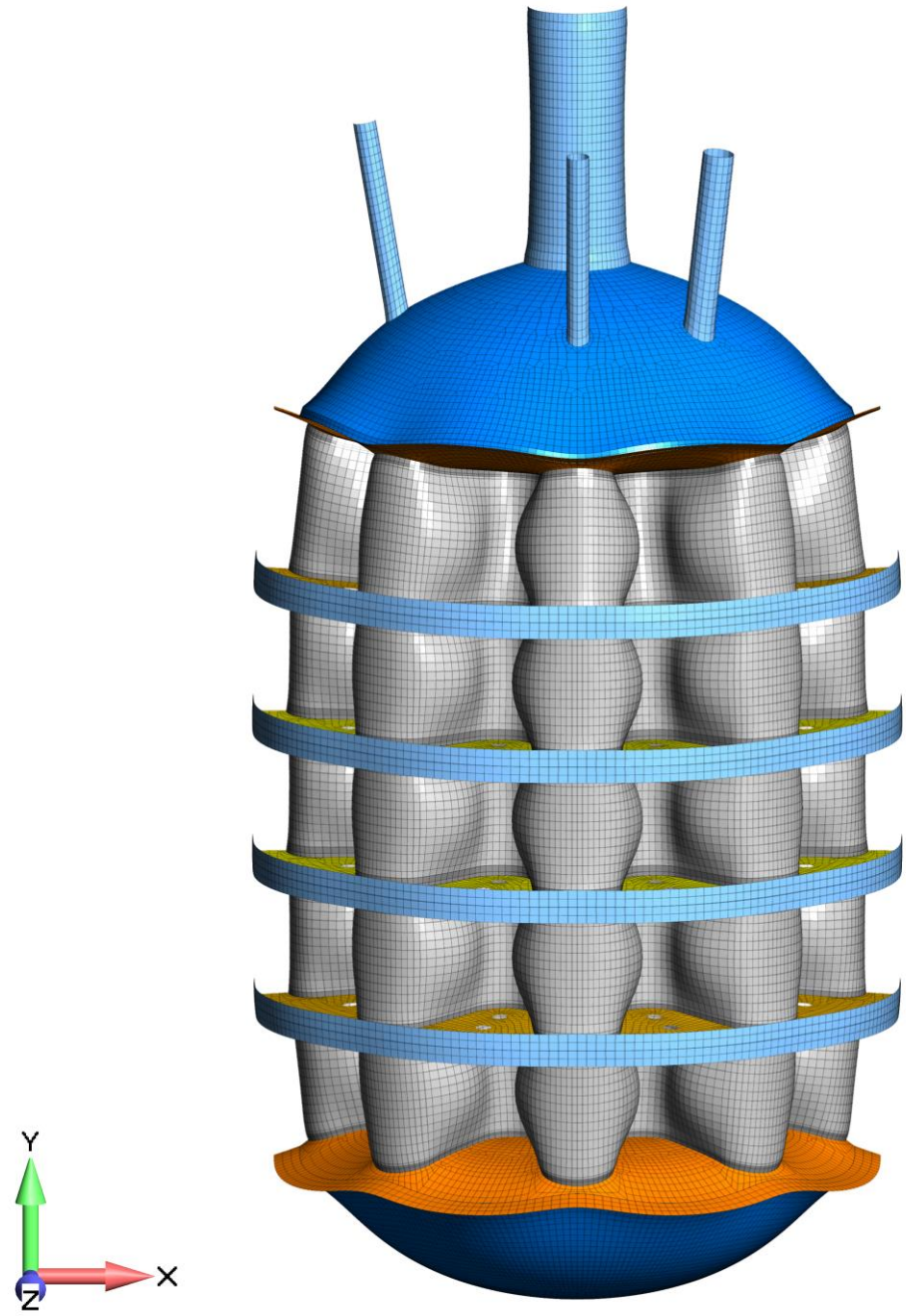


Figure 3: An exaggerated deformation of the vessel under high pressure loading.

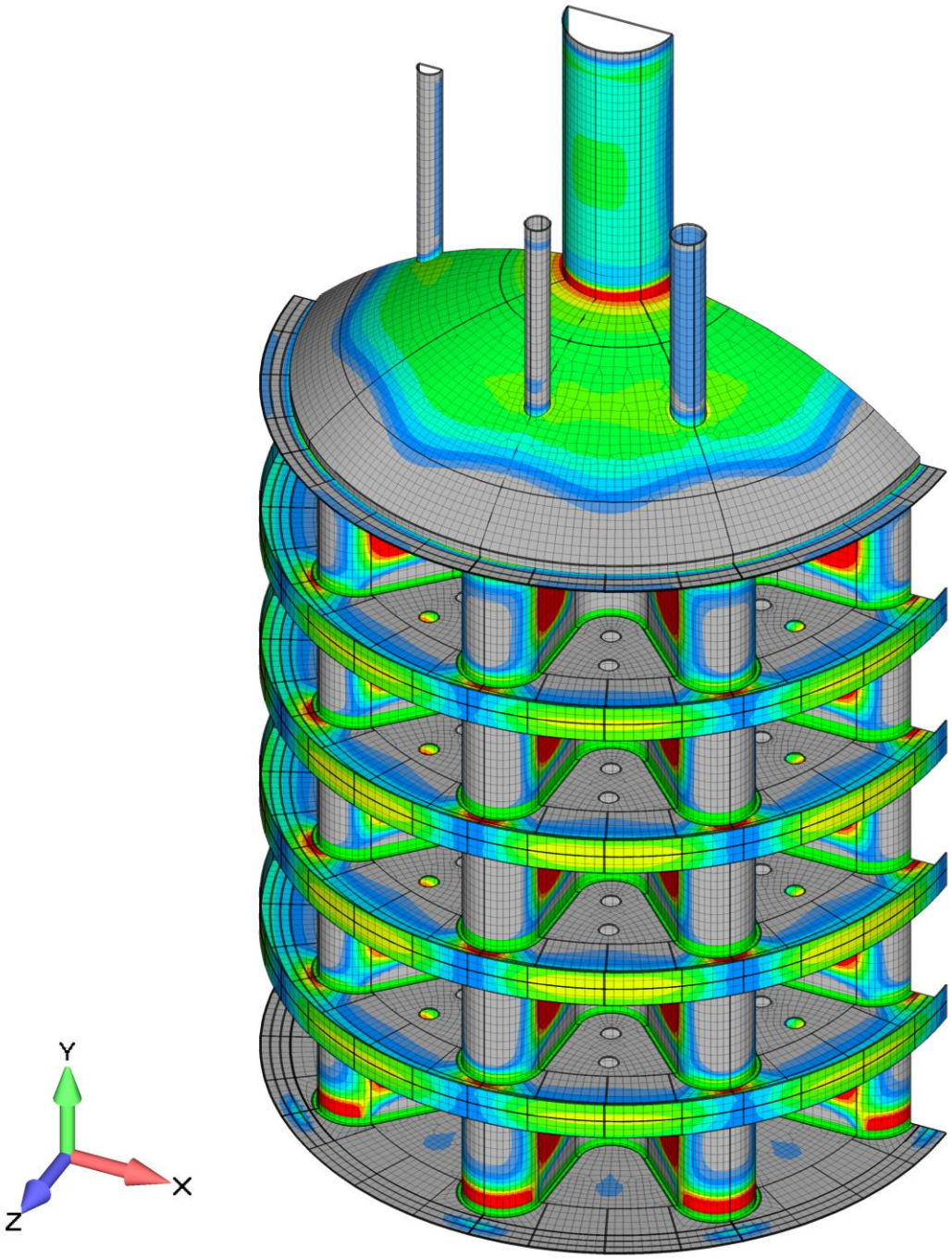


Figure 4: Plate element mesh with stress contour.