

Figure 1: Radial Displacement Under Submersive Load

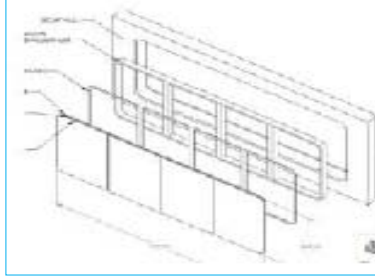


Figure 2: Hull Window Design

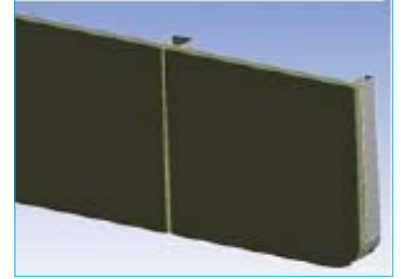


Figure 3: Finite Element Model of Hull Window

Boat Hull Multi-Paneled Window Stress Analysis

A leading manufacturer of boating equipment had to meet strict deflection and stress requirements for a multi-panel boat hull window within very limited time constraints. Hand calculations had been initially performed as a design aid, but our client was concerned with some of the oversimplifications of the approach for this demanding application.

CAE Associates was chosen for this project based on our experience and expertise for projects of this type, and on our commitment to produce the results within the narrow timeframe allocated for the project.

The goal of the simulation was to develop a simplified, yet accurate representation of the deformations and component stresses in the window within a very short time period. Using the ANSYS Mechanical FEA software, CAE Associates performed the simulations by explicitly modeling the multi-panel laminated glass, adhesive, and aluminum frame. All components were tied together using contact elements. Deflection in the laminated glass panels and the stress in the aluminum window frame were evaluated against design specs, confirming the hand calculations. A subsequent design iteration was performed, which allowed for a reduction in the aluminum frame size while maintaining the required safety factor. This was an important benefit of the analysis since any excess weight means additional fuel consumption for the vessel.

Continues >

Boat Hull Multi-Paneled Window Stress Analysis / *Continued*

CAE Associates performed the simulation and the design iteration in less than a week after the initial customer contact. This provided our client with a solution which allowed them to meet their tight design deadlines along with saving both weight and cost.

