Third International Conference on Structural Stability and Stability
June 19-22, 2005 • Gaylord Palms Resort & Convention Center • Kissimmee, Florida

OPTIMIZATION OF A SUBSEA ELECTRONIC HOUSING DESIGN USING LIMIT LOAD ANALYSIS METHODS

Chris Alexander
Stress Engineering Services, Inc.
chris.alexander@stress.com

Glenn Fisseler
Fairfield Industries
gfisseler@fairfield.com

Abstract

A manufacturer of electronic equipment for subsea applications needed to optimize a design to achieve maximum water depths. Using a conceptual design, several analyses were performed using limit load methods to produce a final design. The primary design considerations involved geometric requirements associated with battery sizes and a water depth of 3,000 meters (9,840 feet). Initial efforts to size the dimensions of the design involved the use of a finite element model with shell elements. Once the geometry for the design was finalized, a limit analysis was performed using a model with solid eight-node hexagonal elements. The analysis results demonstrated that the final geometry was adequately designed for the 3,000 meter (9,840 feet) depth requirement. Additionally, full-scale testing with an external pressure corresponding to a water depth of 3,230 meters (10,594 feet) proved the adequacy of the design. This paper describes the analysis methods that were used to optimize the design prior to the fabrication of the prototype subsea housing.

ANALYSIS METHODS

Several analysis techniques were used to determine the final geometry for the 3,000-meter (9,840 feet) subsea housing design. These methods involved the use of classical mechanics, finite element modeling using both shell and solid elements, and integrating the limit load analysis technique to determine a lower bound design load. Discussions are provided on each of these techniques and the contributions that they made to determine the final design.

Classical Mechanics

In many engineering design problems, the use of equations associated with classical mechanics calculations are sufficient. When design geometries and/or the loadings are complex, the application of classical mechanics is limited. A preliminary assessment of the subsea housing design was accomplished by dividing the geometry into sub-regions for detailed study. These regions included wedge-shaped sections of the lid (i.e. ribs), the outer cylinder of the housing, and the individual internally-located vertical support members. However, efforts to incorporate all sections of the housing design required the use of a more rigorous analysis technique, such as the finite element method.

No specific details are provided in this paper on the use of classical mechanics. The initial thicknesses for the lid, outer wall, and ribs were selected using the calculated values using classical mechanics and were used as input into the preliminary finite element shell models.

Shell Model Development

The complex geometry associated with the Fairfield Industries subsea housing required the use of finite element modeling to determine actual stresses and deflections of the structure in response to external pressure. Along the same lines, the limit load method using finite elements is ideally-suited for determining the collapse load for the given structural design. Using these tools, the permissible design geometry and loadings were determined. The limit load method determined the maximum allowable external pressure that the design can withstand. If the calculated lower bound design load was less than the specified design pressure, changes were made to the geometry (i.e. increase wall thickness) to increase the external pressure capacity to an acceptable level. This iterative process continued until a satisfactory design was developed. The advantage in using shell elements during the preliminary stages of the design process is that the thickness of each element can be changed with minimal effort for the finite element model. The disadvantage in using shell elements is that they do not precisely represent that actual geometry of the housing and are unable to account for details such as internal fillet radii, contact between the internal ribs and lid, and any thick-wall mechanics that might exist in certain regions of the design. For this reason, the shell modeling efforts were preliminary steps for determining the general thickness values of the design.

The PATRAN modeling package was used to construct the finite element geometry and define the appropriate boundary conditions and external pressure loads. Processing and post-processing were conducted using the ABAQUS (version 6.3-5) general purpose finite element software with the S4R5 elements.

Figure 1 shows the finite element model that used shell elements. Specific details and results for the shell model are not provided in this paper; however, the following steps were used in integrating the shell model as part of the overall design process.

- The shell model geometry was based upon dimensions provided by the manufacturer. From an AutoCAD drawing a two-dimensional line drawing was made that was then extruded to a specified depth to create the specified shell elements.
- Wall thickness was specified for individual elements (or groups of elements) within the model using dimensions provided by the manufacturer.
- Material properties were applied to the finite element model in the form of elastic modulus and the input of a perfectly-plastic stress strain curve. For the 6061-T6 aluminum a yield strength of 35,000 psi (241 MPa) was used.
- Non-linear strain-displacement relations were used in the finite element model.
- Boundary conditions involved the imposition of a symmetry plane on each edge of the model (see details in **Figure 1**). For the finite element models used in this design analysis, the imposed symmetry plane prevents nodes on the surface from displacing in the 3-direction and prevents rotations about the 1-direction and 2-direction.
- Loading on the finite element model involved applying external pressure on the outside surface of the cylinder portion of the housing. Loads generated by the lid due to contact with the main body of the housing were not included as part of the shell analysis.
- The limit load analysis was conducted using the prescribed model geometry, boundary conditions, and external pressure loading. The failure criterion of the limit

analysis method is the inability for equilibrium to be maintained between the internal and applied forces. External pressure was increased until convergence of the finite element solution was no longer possible. The final load point obtained is defined as the ***lower bound limit load***. In other words, the structure fails to support any additional load.

Numerous finite element models were used in optimizing the final dimensions for the subsea housing. Once a satisfactory lower bound limit load was obtained for one design, dimensions (e.g. wall thicknesses) were reduced until a converged solution was no longer available. This iterative process produced the preliminary design that served as the foundation for the solid model geometry.

As a point of reference, **Figure 2** provides the displaced shape and stress contour plot for one of the shell finite element models. This figure demonstrates the displacement of the outer cylinder and shows that high stresses were generated in the same sections of the model where maximum deflections occurred.

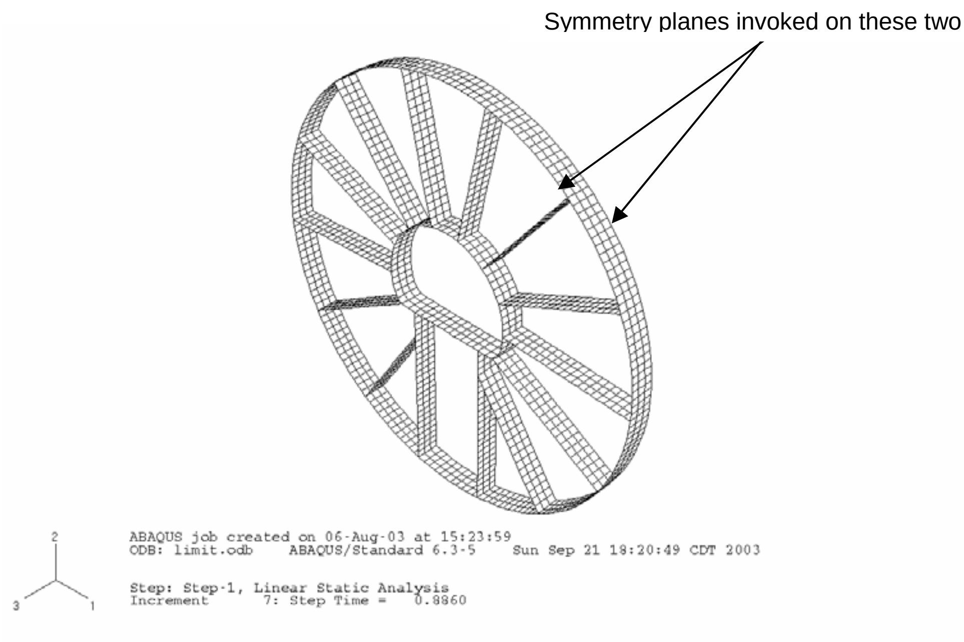

Figure 1. Finite element model based on shell elements

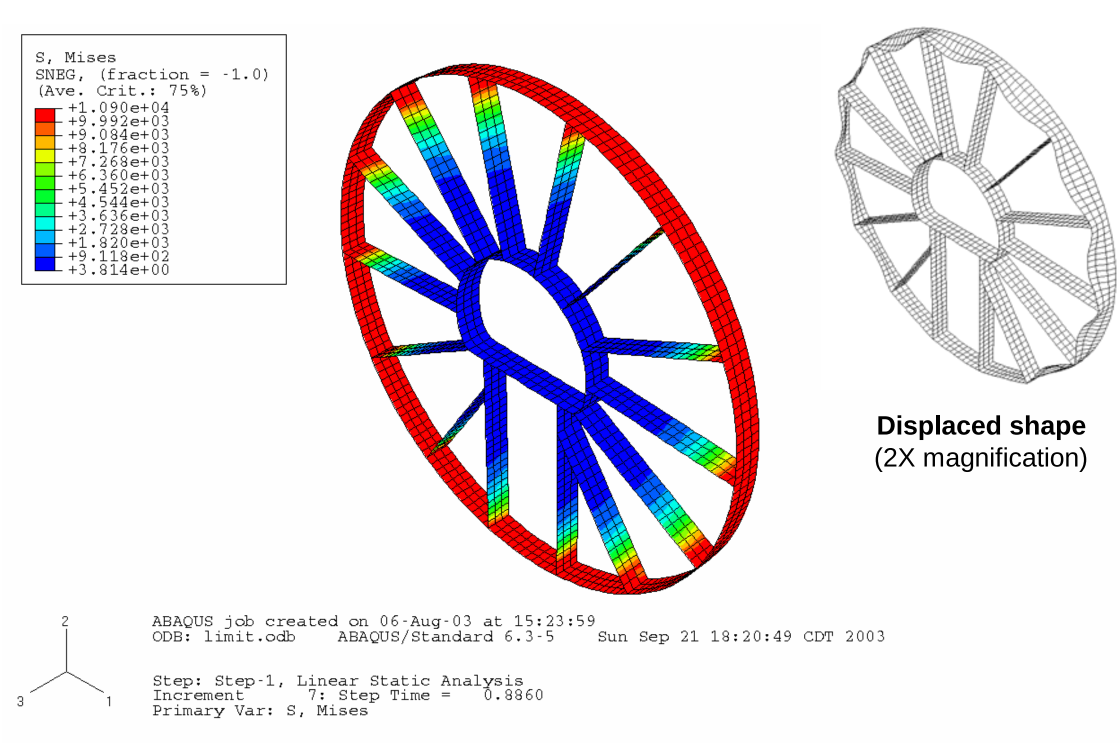

Figure 2. Von Mises stress contour plot for shell finite element model (units of *psi*)

Solid Model Development

Once the basic geometry for the subsea housing was selected, a design was selected to satisfy both the structural stability and on-board electronic battery pack storage requirements. A three-dimensional finite element model using solid continuum elements was generated. **Figure 3** provides an isometric view of the solid finite element model that shows the details on the symmetry plane and components associated with the housing. Spring elements were used to prevent rigid body motion of the model and are indicated by the four black dots located on the symmetry plane (two are visible in this figure). **Figure 4** shows the internal components of the model with the lid removed.

The solid model incorporated the following details.

- Geometry included the outer cylinder of the main body housing, internal ribs oriented radially outward, 0.85-inch (21.6 mm) thick internal plate, and a 0.90-inch (22.9 mm) thick lid.
- Contact was modeled between the lid and the main body housing. Contact on the main body was generated on top of the ribs and in the recessed portion on each end of the housing.
- Bolting to attach the lid to the housing was accomplished by connecting nodes on the lid and body. This connection method was defined at eight (8) regions on the housing located 45 degrees apart circumferentially.

- A symmetry plane was invoked half-way between the ends of the housing. This cut the 0.85-inch (21.6 mm) internal plate in half. As with the shell model, this boundary condition prevents nodes on the symmetry plane from displacing in the 3-direction.
- External pressure was applied to all outside surfaces of the housing including the lid and the cylinder. An external pressure of 10,000 psi (68.8 MPa) was applied to the model. This value exceeds the design requirement of 3,000 meters (9,840 feet), but was deemed high enough that convergence of the finite element model would be unlikely (i.e. pressure magnitude sufficient to ensure that lower bound limit load will be determined).
- As with the shell models, the lower bound limit load was obtained by increasing external pressure on the finite element model to the point where the structure fails to withstand any additional load (i.e. convergence of the finite element solution was no longer possible).

The main body of the housing is manufactured using 6061 T6 aluminum with a yield strength of 35,000 psi (240.7 MPa), while the 0.9-inch (22.9 mm) thick lid is to be fabricated from 7075 material. A yield strength of 61,250 psi (421.3 MPa) was used for the 7075 material.

The following section provides details on the results obtained for the solid finite element model as well as interpretation of results as they relate to the specified design requirements.

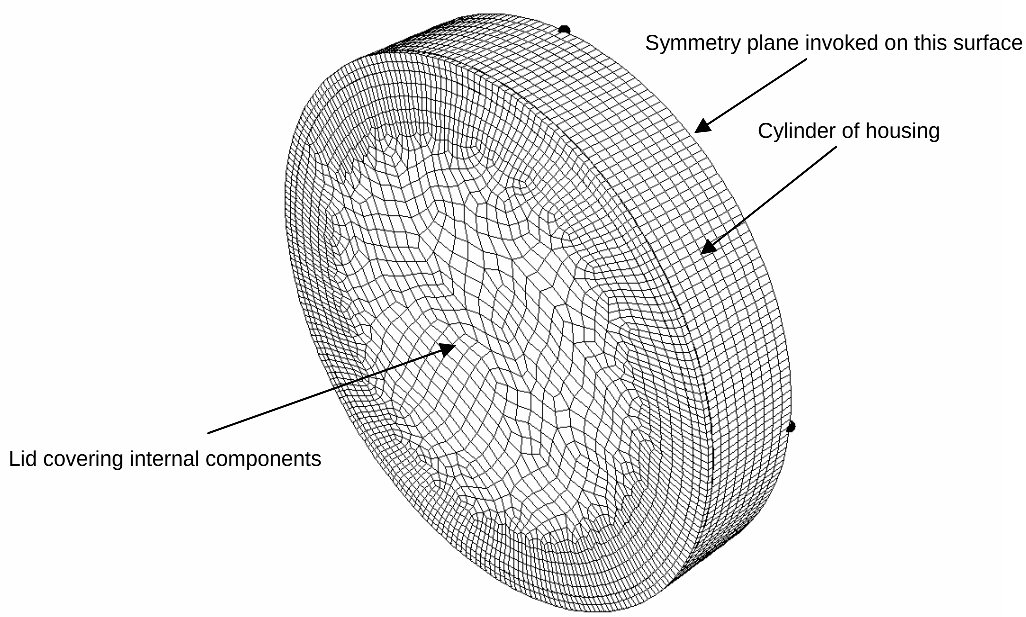

Figure 3. Isometric view of solid finite element model

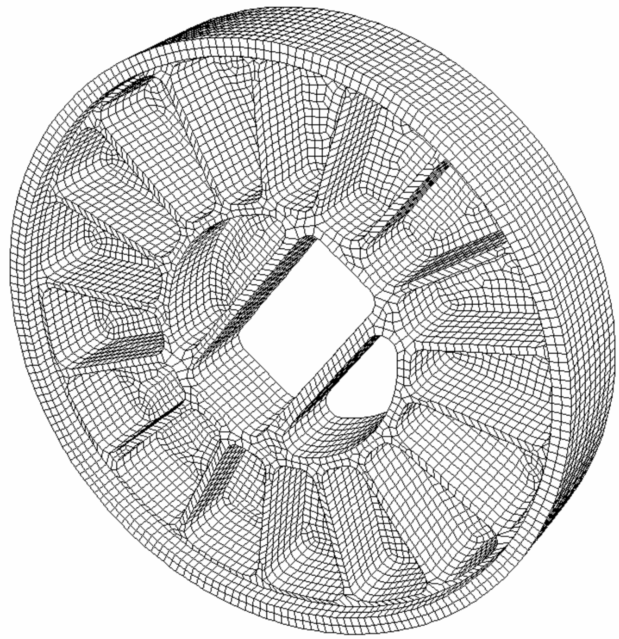

Figure 4. Isometric view of solid finite element model with the lid removed

ANALYSIS RESULTS

Using the finite element method to determine a lower bound limit load was ideally-suited for the design of the subsea housing. Modeling involved using the appropriate design geometry, invoking the appropriate boundary and loading conditions, and using the appropriate material models. An external pressure was applied to the model that exceeded the design requirements by a certain margin. If this value was too low, the undesired convergence of the solution will be achieved. As shown in this section of the paper, the applied 10,000 psi (68.8 MPa) external pressure was appropriate as the finite element solution stopped at 94.1 percent of this value, or 9,410 psi (64.7 MPa).

Provided in this section are the results that involve the following.

- Finite element convergence data
- Displacement and stress contour plots
- Interpretation of results and application of the calculated limit value

Finite Element Convergence Data

When ABAQUS solves a finite element problem, it produces a status file (e.g. *model_input.sta*) that reports the convergence parameters for the respective model. When performing a limit analysis, specific information contained within this file is useful. **Figure 5** provides the output data obtained for the solid finite element model. In this figure two columns are important.

- The data plotted in **red** constitute an incremental fraction of applied load. A value of 0.10 implies that 10 percent of the total load has been applied. As noted in **Figure 5**, the model stopped when an increment fraction of 0.941 was reached. For the problem at hand this means that the lower bound limit load is 94.1 percent of the total applied load (i.e. 10,000 psi). Consequently, the calculated lower bound limit load is 9,410 psi (64.7 MPa) that corresponds to a subsea depth of 6,461 psi (44.4 MPa). To this value a design safety factor is applied.

- The data plotted in **blue** constitute deflection of a tied node where a bolt was assumed to exist. Although not necessarily applicable for the problem at hand, deflection data is often useful for generating load-deflection plots. At the end of the load step, large deflection of the structure takes place with small increases in load.

From the data provided in **Figure 5**, a permissible design load is selected as a function of the lower bound limit load. Greater details are provided in the ***Interpretation of Results*** section of this paper.

```
ABAQUS VERSION 6.3-5                                DATE 21-SEP-2003  TIME 17:09:37
 SUMMARY OF JOB INFORMATION:
 MONITOR NODE:   25752  DOF:  2
 STEP   INC ATT TOTAL  TOTAL      STEP       INC OF      DOF
                ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR
                       FREQ

   1      1   1     6  0.100      0.100      0.1000     -0.00267
   1      2   1     5  0.200      0.200      0.1000     -0.00481
   1      3   1     5  0.350      0.350      0.1500     -0.00782
   1      4   1     5  0.575      0.575      0.2250     -0.0124
   1      5   1    11  0.913      0.913      0.3375     -0.0241
   1      6   2     6  0.934      0.934      0.02187    -0.0359
   1      7   2     5  0.940      0.940      0.005469   -0.0469
   1      8   3     4  0.941      0.941      0.001000   -0.0574

 THE ANALYSIS HAS NOT BEEN COMPLETED
```

Figure 5. ABAQUS status file output for finite element model

Displacement and Stress Contour Plots

The displacement and stress contour plots are useful for post-processing the finite element model to show the overall response of the structure to applied loads. **Figure 6** shows the displaced shape of the finite element model subject to an external pressure of 9,410 psi (64.7 MPa). This figure does not have any magnification on the plotted displacement of the model.

Figure 7 and **Figure 8** show two views of the model with contour plots of the Von Mises stresses. All values plotted in red exceed 35,000 psi (yield strength for the 6061 T6 material). A more detailed study of the finite element models indicated that the highest stresses in the main body of the housing occurred in the vessel wall located between the ribs. The other members within the housing that experience high stresses are the ribs. The primary purpose of the rib is to provide additional support to the lid, although a secondary function is to provide stiffness to the cylindrical portion of the housing.

Interpretation of Results
Having calculated the lower bound limit load, it is appropriate to discuss a design criterion to establish the allowable safe operating depth for the subsea housing. The calculated lower bound limit load is 9,410 psi (64.7 MPa), or 6,461 meters (21,192 feet). Division 3 of the ASME Boiler & Pressure Vessel Code permits a factor of 2.0 on the lower bound limit load without restrictions. Using this design factor, a design pressure of 4,704 psi (32.4 MPa) is calculated. This corresponds to a sea depth of 3,230 meters (10,594 feet), which exceeds the minimum design requirement of 3,000 meters (9,840 feet).

In addition to the analysis work, actual full-scale physical testing was performed involving external pressure. An external pressure of 7,000 psi (48.1 MPa) was applied to the housing. This pressure exceeds the design pressure of 4,704 psi (32.4 MPa) that corresponds to a water depth of 3,230 meters (10,594 feet). The prototype housing was able to withstand the applied external pressure.

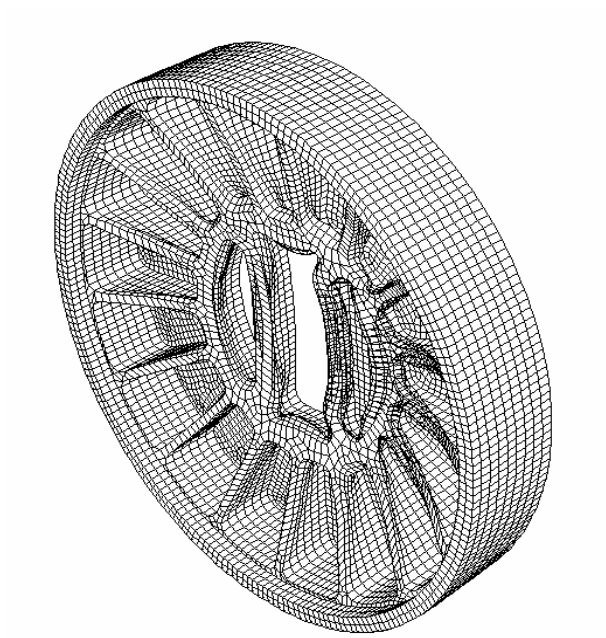

Figure 6. Displaced shape for the solid finite element model
(1X magnification at 9,410 psi)

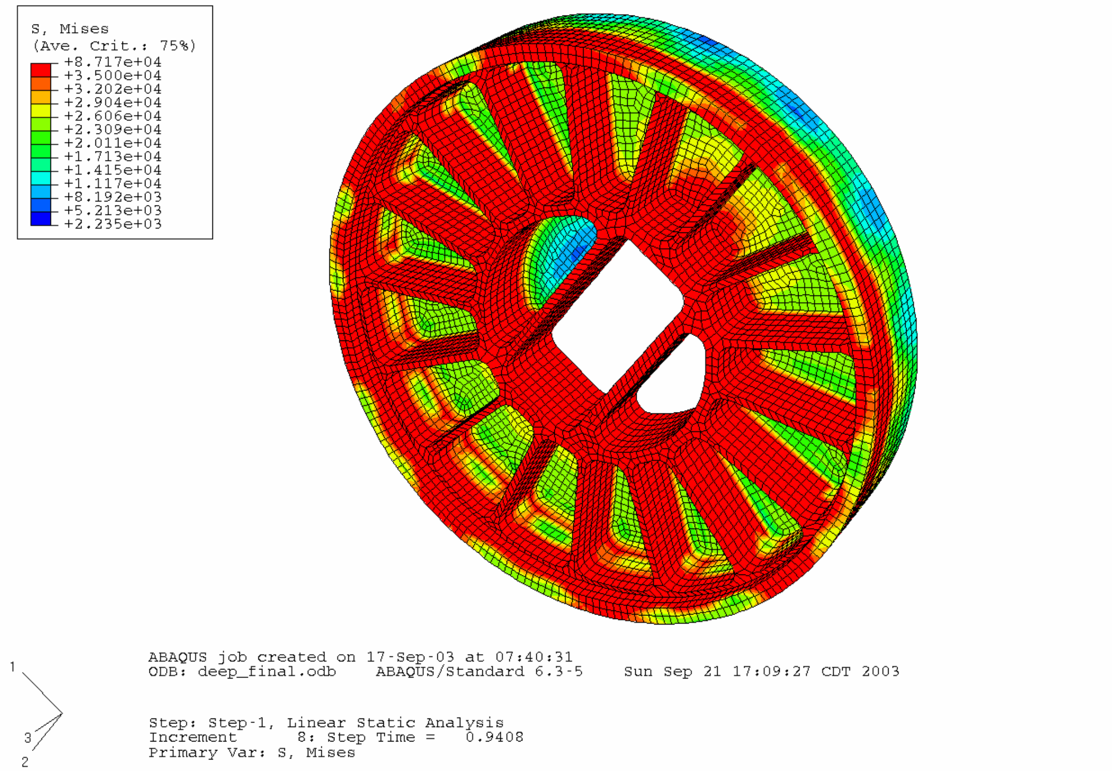

Figure 7. Von Mises stress contour plot showing internal components (units of *psi*)

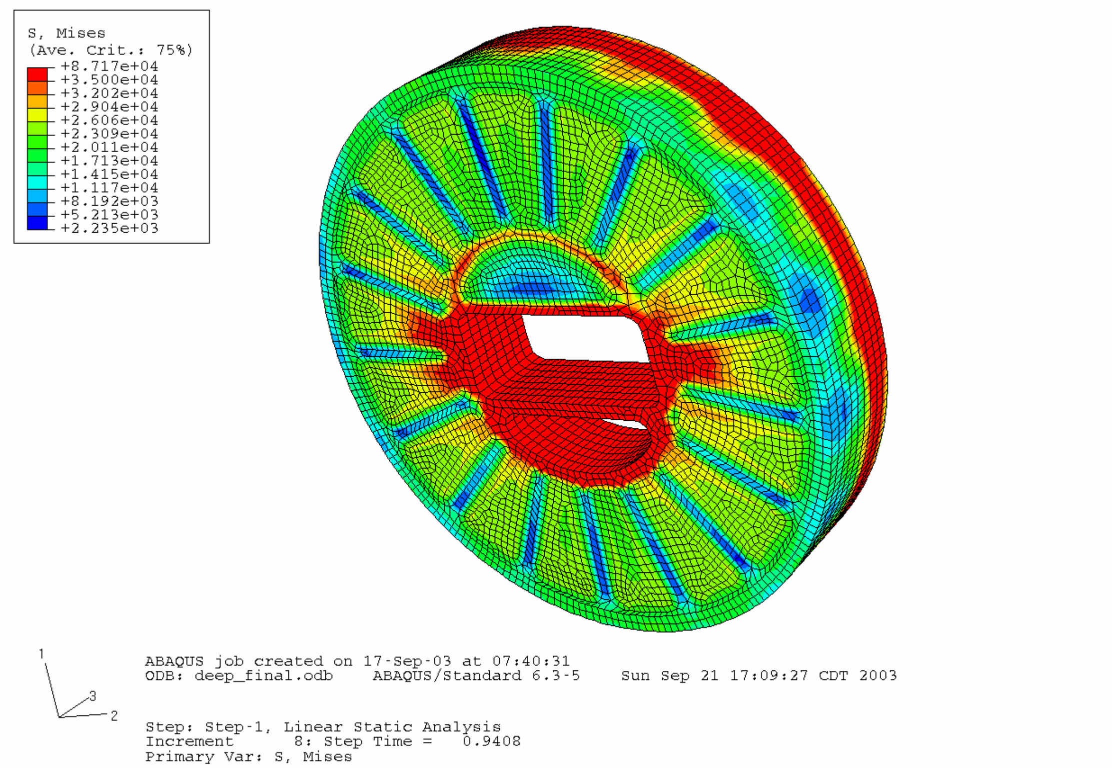

Figure 8. Von Mises stress contour plot showing symmetry plane (units of *psi*)

CONCLUSIONS

Finite element-based limit analysis methods were used to assist a manufacturer in the design development of a subsea vessel to house electronic devices. Models using shell and solid elements were integrated into this design process. The shell models were used as a preliminary means for sizing the design, with specific emphasis on the thickness of the cylinder and internal components such as the ribs. Once the basic geometry was defined, a finite element model using three-dimensional solid continuum elements was created. This model was used for calculating the final lower bound limit load.

External pressure was increased until convergence of the finite element solution was no longer possible. The model stopped when an increment fraction of 0.941 was reached. This results in a lower bound limit load that is 94.1 percent of the total applied load. Consequently, the calculated lower bound limit load is 9,410 psi (64.7 MPa), which corresponds to a subsea depth of 6,461 meters (21,192 feet). To this value a design safety factor of two (2) is applied, resulting in a permissible design depth of 3,230 meters (10,594 feet). This value exceeds the specified minimum design requirement of 3,000 meters (9,840 feet).

The analysis design work was validated by experimental testing that involved the application of external pressure of 7,000 psi (48.1 MPa). This test pressure is well in excess of the design pressure of 4,704 psi (32.4 MPa) that corresponds to a water depth of 3,230 meters (10,594 feet).

The methods presented in this paper indicate that limit load analysis methods can be used to assist manufacturers in the design of structures, especially those designs that are directly influenced by potential instabilities. Along the same lines, an iterative design process permits manufacturers to assess the effects of changes in design variables prior to the development of prototype components used during proof testing. The alternative to using analysis design methods involves testing full or sub-scale samples and making changes based on experimental observations. This process can be expensive and time consuming, and almost always fails to generate the insights gained when using the iterative design tools such as finite element methods.

REFERENCES

ASME, Boiler and Pressure Vessel Code, Section VIII, Division 2, *Rules for Construction of Pressure Vessels, Alternative Rules*; and Section II, Materials, Part D, Properties, The American Society of Mechanical Engineers, Three Park Avenue, New York, NY 10016.

Kalnins, Arturs, WRC Bulletin 464, *Guidelines for Sizing of Vessels by Limit Analysis*, August 2001, Welding Research Council, Inc. Three Park Avenue, New York, NY, 10016.

ABAQUS/Standard, Hibbitt, Karlsson & Sorensen, Inc. 1080 Pawtucket, RI 02860-4847.