

Modeling and Sensitivity Analysis of Sandwich Composite Cylinders with Geometric Imperfections

Kyongchan Song¹, Arunkumar Satyanarayana², Adam Przekop³, and
Marc Schultz³

NASA Langley Research Center, Hampton, VA 23681

It is well known that real shell structures can have significantly lower buckling loads and even different mode shapes than the theoretical predictions for perfect structures. Much of this difference can be attributed to geometric and loading imperfections, and geometrically nonlinear effects. The realistic buckling response can be investigated using geometrically nonlinear finite element analyses and including radial imperfections. Such analyses are used in the NASA Engineering and Safety Center Shell Buckling Knockdown Factor Project, which has the goal of developing new analysis-based buckling design recommendations for select classes of cylindrical shell structures under uniaxial compressive load. The approach for modeling several sandwich composite cylinders with two-dimensional general-purpose shell elements and the influence of the element type selection and element size on the buckling load is discussed. The influence of geometric imperfections of various magnitudes on buckling behavior of a sandwich composite cylinder was also investigated.

I. Introduction

One of the goals of the NASA Engineering and Safety Center (NESC) Shell Buckling Knockdown Factor (SBKF) Project is developing buckling design recommendations for a select class of composite launch-vehicle structures. As part of SBKF [1,2], finite element models (FEMs) of sandwich composite cylinders were developed and analyses were performed to predict the buckling responses of geometrically perfect and imperfect sandwich composite cylinders. In order to develop acceptable FEMs, various analysis and modeling approaches [3,4] and analysis tools [5,6] were reviewed in order to develop the SBKF modeling approach for sandwich composite cylinders.

As the first step in the development of a FEM, the model developer has many factors to consider such as the element type, the element size, material property definition, and the methods for application of boundary and loading conditions. These factors were considered and studied during the development of the SBKF finite element modeling approach for sandwich composite cylinders. The modeling approach and verification study discussed herein were focused on FEMs with two-dimensional general-purpose shell elements.

The present paper has three primary objectives. The first objective is to discuss the chosen analysis tool and FEM modeling procedures used to predict buckling response of geometrically perfect and imperfect sandwich composite cylinders. The second objective is to investigate the influence of two-dimensional general-purpose shell element types available in the commercial code Abaqus [5] (S4, S4R, S4R5, S8R, S8R5), and consider the influence of their size on the predicted buckling load. The third objective is to investigate the influence of geometric imperfections of various magnitudes on the predicted buckling behavior of sandwich composite cylinders of several configurations. In Section II, the geometric description of the sandwich composite cylinders, the material system, and the modeling approach used in applying compressive load and boundary conditions to the sandwich composite cylinder is explained. In addition, various analysis types are discussed in this section. In Section III, results from buckling analysis of a cylinder for several shell element types and sizes are presented. In Section IV, the influence of shell element types on buckling

¹ Research Aerospace Engineer, Structural Mechanics and Concepts Branch, Member

² Research Aerospace Engineer, Structural and Thermal Systems Branch, Member

³ Research Aerospace Engineer, Structural Mechanics and Concepts Branch, Associate Fellow

analysis of geometrically perfect and imperfect cylinders is presented. In Section V, the sensitivity of buckling load to geometric imperfections of the sandwich composite cylinder is presented, and conclusions are given in Section VI.

II. Finite Element Model of Sandwich Composite Cylinder

The general-purpose finite element software Abaqus/Standard 2016 was used for this work. MSC Patran [6] and scripts within the framework of Patran were used in the development of FEMs and Abaqus/Viewer 2016 was used for postprocessing of results. The description of the selected sandwich composite cylinder and the modeling approach are discussed in this section.

A. Geometry and Material System of Sandwich Composite Cylinder

Three cylinder lengths, 90 inch, 180 inch, and 331 inch, with a midsurface diameter of 331 inches were considered and are shown in Fig. 1. The cylindrical coordinate system with one axis aligned with the cylindrical axis of rotation is shown in Fig. 2, and the origin in the plane of the bottom circumferential edge was used in this study.

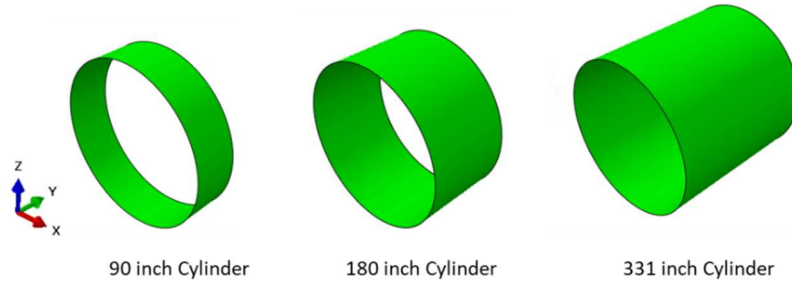


Figure 1. Three different length sandwich composite cylinders.

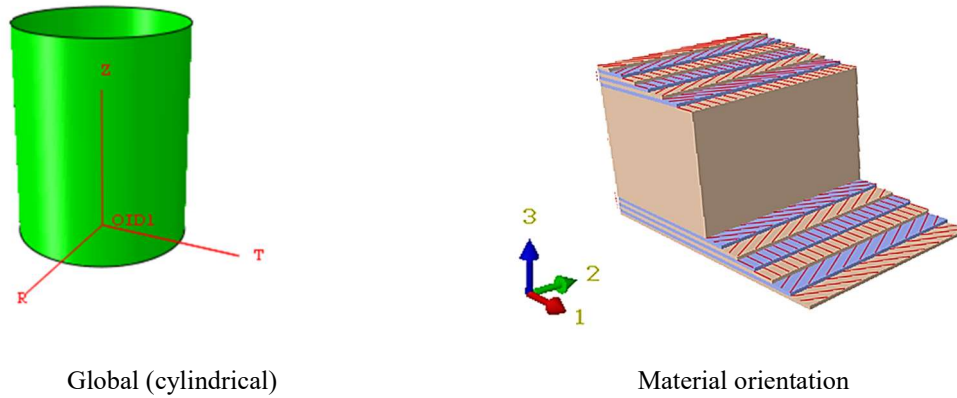


Figure 2. Cylindrical coordinate system and material orientation used in this study.

Three facesheet layups were considered in the present study: quasi-isotropic and axially stiff layups made from IM7/8552 unidirectional tape plies, and a fabric layup consisting of IM7/8552 plain-weave fabric plies. The quasi-isotropic and axially stiff unidirectional tape layups had stacking sequences of $[\pm 45/90/0]_s$ and $[+45/-45/90/0/0/0/-45/45]_T$, respectively, and the fabric layup had a stacking sequence of $[+45/0/0/0/0/+45]_T$. The core material used in this study was a Hexcel 3.1 pcf 1/8-5056-0.0007 aluminum honeycomb with various thicknesses. The material properties of the facesheet and core material systems are provided in Table 1.

Table 1. Material properties.

Properties	IM7/8552 unidirectional tape [7]	IM7/8552 fabric [8]	Aluminum honeycomb core Hexcel 3.1 pcf 1/8-5056-0.0007 [9]
Thickness (inch) [†] , t_f , t_c	0.0054	.00787	0.5, 0.75, 1.0
E_1^t , E_1^c (Msi)	22.99, 20.04	10.585	0.001
E_2^t , E_2^c (Msi)	1.3, 1.41	10.586	0.001
G_{12} (Msi)	0.68	0.696	0.000345
G_{13} (Msi)	0.68	0.595	0.045
G_{23} (Msi)	0.408	0.595	0.020
ν_{12}	0.316	0.35	0.45

[†]Thickness values chosen for this study, note the given references.

The facesheets and core of the sandwich composite cylinder were discretized using two-dimensional general-purpose shell elements. Because general-purpose shell elements were used, the element membrane response was treated with a finite-strain formulation that gives accurate solutions to in-plane bending problems, is not sensitive to element distortion, and avoids parasitic locking [5]. For sandwich shell sections, the transverse shear stiffness values required in the element formulation were calculated by Abaqus. In the analyses, the core was treated as a linear elastic orthotropic continuum and only balanced and symmetric facesheets were considered. Since the cylinders considered here had a large diameter compared to the thickness, the effects of individual shell element curvature were neglected. In the finite element shell models, the midsurface coincided with the reference surface for the perfect geometry model while the inner mold line (IML) coincided with the reference surface for the geometrically imperfect model.

B. Application of Boundary and Load Conditions

The boundary conditions applied at the circumferential edges of the cylinder can influence the buckling response of the cylinder in a significant way. For example, a clamped boundary condition generally leads to a larger buckling load than a simply supported boundary condition. Simply supported boundary conditions were used at the circumferential edges of the cylinder in all analyses performed in this study. With these conditions the nodes on the circumferential edges were allowed to rotate freely with respect to tangent of the circumferential edge. The boundary conditions were implemented by coupling nodes on the circumferential edge to the central reference node at each end, which lies in the end plane of the cylinder along the axis of rotation; the degrees of freedom of the reference node were defined by the cylindrical coordinate system. This node configuration was considered here as a *wagon-wheel* configuration and is shown in Fig. 3. All three translational displacements (U_R , U_θ , U_Z) and two rotational displacements (R_R and R_Z) between the center node and the nodes on circumferential edge were linked through the kinematic coupling command defined in the Abaqus input deck, which resulted in a simply supported shell edge. The center node at the bottom (origin) end was fixed in all degrees of freedom (DOF) and the center node at the top (positive Z) end was fixed in all DOF except the axial displacement. An axial displacement was applied to this node as shown in Fig. 4. Note that the orientation identification (CID) of the coupling nodes must be used in the *KINEMATIC COUPLING card for proper coupling of degrees of freedom between the center node and the nodes on the circumferential edge.

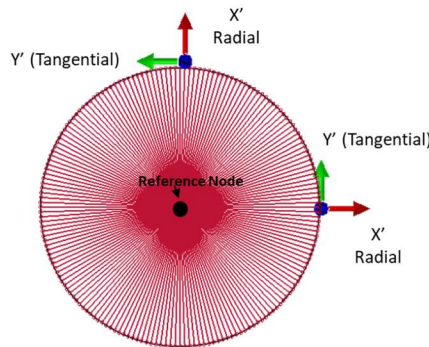


Figure 3. Kinematic coupling between center reference node and circumferential nodes.

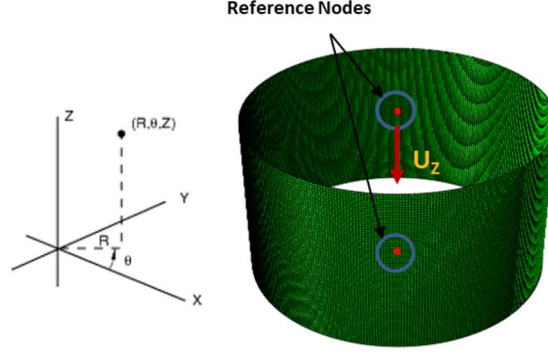


Figure 4. Boundary condition at center reference nodes in cylindrical coordinates.

C. Finite Element Analysis

After the boundary conditions of the sandwich composite cylinder FEM were defined, finite element analyses (FEAs) were performed using the linear-elastic material behavior and uniform compression as described in the previous section. Applied compression was assumed to be uniform over both IML and OML end edges and all of the axial loads were carried by the facesheets.

FEA were performed utilizing four different analysis procedures: natural frequency, linear buckling, nonlinear static, and transient dynamic analysis. Since the analyses were performed to predict the buckling response of sandwich composite cylinders, strength failures of the facesheet, core, and the interface between facesheets and core were not considered.

The FEA solution parameters such as solution convergence tolerance and solution damping were maintained at default value levels for all the analysis methods. While performing frequency analysis, and linear bifurcation (buckling) analysis, the built-in Lanczos solver was used to evaluate buckling load and modes.

The following load and analysis step parameters were used. For nonlinear analysis, the minimum step size was not less than 0.001 and maximum step was not greater than 0.05. In the case of transient dynamic analysis, in order to ensure quasistatic loading, the load was applied in displacement control using constant-rate end shortening up to 2.0 inches over a pseudo time of 1800 sec. Solution parameter “halfotl” was set equal to the approximate buckling load of the cylinder, and the default solution damping value of -0.05 was used in this study.

For each series of analyses, the first FEA to be performed was frequency extraction analysis. The results from this frequency analysis were used to determine the material damping factors for other analysis types as described in the next paragraph.

When performing dynamic analyses, it is important to use appropriate damping coefficients that are large enough to prevent unrealistic numerical vibrations, but small enough so that they will not suppress realistic behavior. Both Rayleigh mass proportional (alpha) and Rayleigh stiffness (beta) proportional material damping were applied to the model during transient dynamic analysis. This process to calculate these damping factors is described in Ref. [10] and the damping factors were calculated using the equations

$$\alpha = 2\pi f \xi \quad \beta = \frac{\xi}{2\pi f}$$

Where f is the natural frequency of the finite element model and ξ the damping coefficient. For this work, a damping coefficient of 0.05 was used.

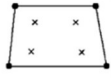
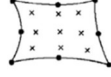
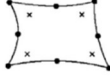
III. Finite Element Type and Mesh Convergence Study

For modeling a sandwich composite shell that contains a soft nonisotropic core material, the Abaqus *User's Guide* recommends using a general-purpose shell element or continuum shell elements [5]. The Abaqus general-purpose shell elements are S4, S4R, S4R5, S8R, and S8R5. The description of general-purpose shell element and integration points in first and second order shell element are shown in Tables 2 and 3, respectively.

Table 2. General-purpose shell elements available in Abaqus.

Element	Description
S4	Four-node general-purpose shell, finite membrane strains
S4R	Four-node general-purpose shell, reduced integration with hourglass control, finite membrane strains
S4R5	Four-node thin shell, reduced integration with hourglass control, using five degrees-of-freedom per node, no drilling degree-of-freedom
S8R	Eight-node doubly curved thick shell, reduced integration
S8R5	Eight-node doubly curved thin shell, reduced integration, using five degrees-of-freedom per node, no drilling degree-of-freedom

Table 3. Integration points in Abaqus first- and second-order shell elements [5].

	Full integration	Reduced integration
First-order interpolation S4		
Second-order interpolation S8		

The purpose of this study was to obtain a converged mesh size for a sandwich composite cylinder subjected to a compression load for a given geometry definition and to investigate the influence of the general-purpose shell-element type in Abaqus on the predicted buckling load and load-displacement response. Three different meshes were generated for the mesh convergence study of 180-inch-long cylinder:

- 0.5-degree mesh: 720 elements along circumference, 1.45-inch square element size
- 1.0-degree mesh: 360 elements along circumference, 2.90-inch square element size
- 2.0-degree mesh: 180 elements along circumference, 5.80-inch square element size

This study consisted of performing bifurcation buckling and nonlinear static buckling analyses. The default bifurcation buckling analysis solver in Abaqus is the subspace iteration eigensolver. However, the Lanczos solver can be used to calculate buckling responses similar to the buckling responses obtained from the subspace iteration eigensolver with less computing cost. The bifurcation buckling analysis utilized the Lanczos solver with at least 10 eigenvalues requested.

Linear bifurcation buckling analyses were performed on FEMs of a cylinder with quasi-isotropic facesheets and a 0.5-inch-thick core to investigate the influence of shell element type and mesh size on the predicted buckling load and modes. The predicted first buckling loads from the linear buckling analysis of FEMs are listed in Table 4. The variation of the buckling load for all shell element types considered in this study was negligible (with buckling loads well within 1% of each other) for the 0.5-degree and 1.0-degree meshes. The predicted buckling loads from the 2.0-degree mesh do not agree as well, with differences up to more than 3% from the 0.5-degree mesh. Therefore, only the 0.5-degree and 1.0-degree meshes with S4R element, i.e., meshes and elements with least computational cost, were considered for further analyses.

Table 4. Predicted linear buckling loads (in lbf) as a function of element type and size

Mesh	Abaqus Element Type				
	S4	S4R	S4R5	S8R	S8R5
0.5-degree	2.325 E+6	2.325 E+6	2.325 E+6	2.330 E+6	2.326 E+6
1.0-degree	2.341 E+6	2.335 E+6	2.335 E+6	2.331 E+6	2.327 E+6
2.0-degree	2.403 E+6	2.332 E+6	2.332 E+6	2.341 E+6	2.329 E+6

Linear bifurcation and nonlinear static analyses were performed on FEMs of a sandwich composite cylinder for three facesheet stacking sequences and two core thicknesses (0.5 inch and 1.0 inch) as part of the mesh convergence

study. Linear buckling and nonlinear peak loads are shown in Figs. 5-7 for quasi-isotropic, axially stiff, and fabric layups, respectively.

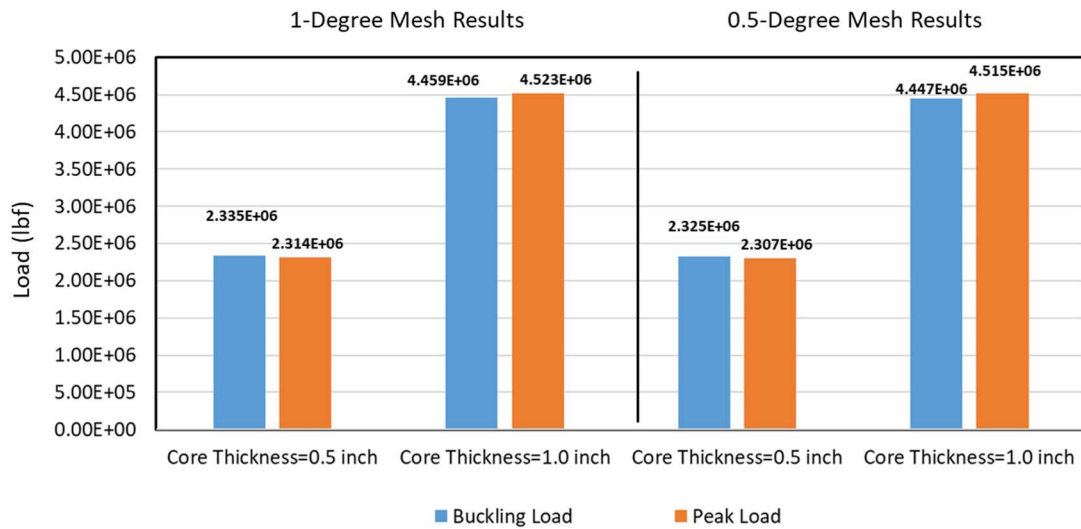


Figure 5. Linear buckling and nonlinear peak loads of FEM with quasi-isotropic facesheets.

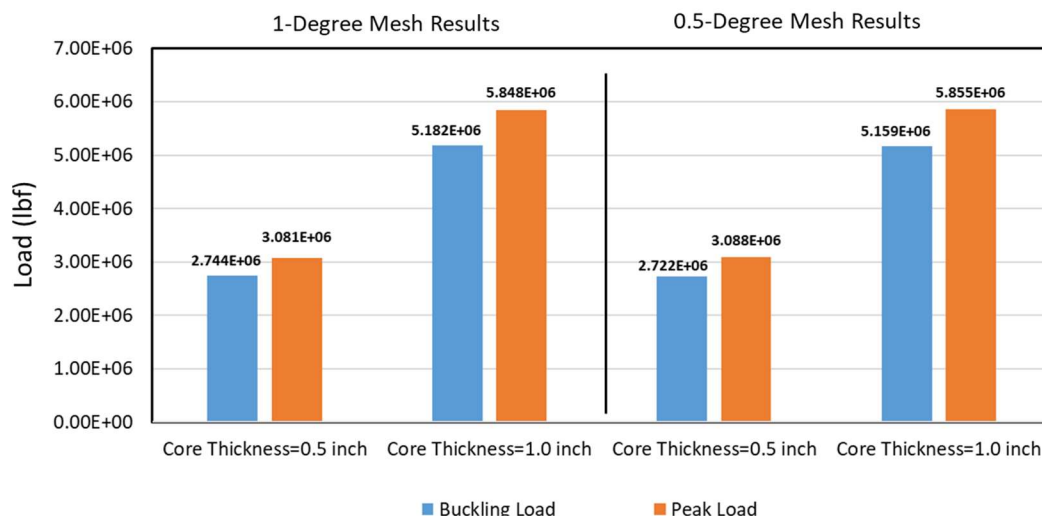


Figure 6. Linear buckling and nonlinear peak loads of FEM with axially stiff facesheets.

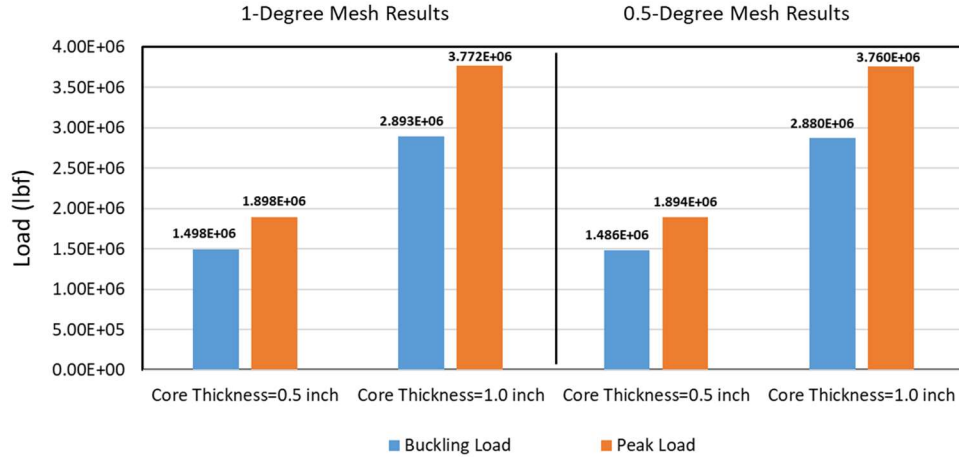


Figure 7. Linear buckling and nonlinear peak loads of FEM with fabric facesheets.

The linear buckling and peak-load results presented in Figs. 5-7 indicate that the 1.0-degree mesh model which has 360 elements along circumference was a sufficiently refined model because corresponding results between the 0.5-degree mesh and the 1.0-degree mesh all agree within 1% for three varying facesheet layups and two different core thicknesses. Based on these results, FEMs for more extensive parametric studies of the buckling response of sandwich composite cylinders were developed with S4R elements and the 1.0-degree mesh density along the circumferential direction.

IV. The Influence of Finite Element Type for Geometrically Perfect and Imperfect Cylinders

The buckling loads and modes of sandwich composite cylinders are sensitive to imperfections in the geometry, thickness, and loading. In this regard, different modeling approaches can be used to impose geometric imperfection on a FEM. The geometric imperfection of the first SBKF 8-ft-diameter cylindrical composite test article (designated CTA8.1) was measured and is presented in Fig. 8. This geometric imperfection was obtained by measuring the inner mold line (IML) and the outer mold line (OML) cylinder surfaces and averaging the two measurements. This imperfection was applied as a midsurface imperfection to the FEM. Imperfection sensitivity analysis was carried out to investigate the influence of the general-purpose shell element types on the predicted peak load and post-buckling response of geometrically imperfect sandwich composite cylinder models of a quasi-isotropic facesheet with a 1.0-inch-thick core. Transient dynamic analysis was performed on FEMs with 1.0-degree mesh, for different shell element types and four different amplitudes of the CTA8.1 geometric imperfection (no imperfection or perfect, 0.4-inch, 1.0-inch, and 2.0-inch peak-to-peak imperfections). Predicted linear buckling and nonlinear peak loads obtained from linear bifurcation buckling and transient dynamic analyses are presented in Table 5.

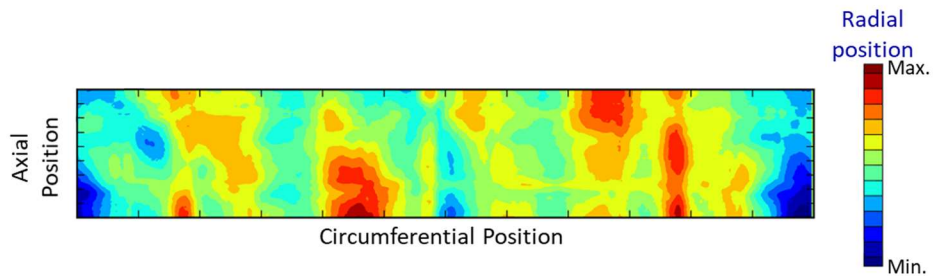


Figure 8. CTA8.1 test article midsurface geometric imperfection input.

Table 5. Predicted linear buckling and nonlinear peak loads (in lbf) for each element type for transient dynamic analysis.

Peak-to-peak imperfection	Abaqus Element Type				
	S4	S4R	S4R5	S8R	S8R5
Perfect (Linear Buckling)	4.464E+06	4.461E+06	4.437E+06	4.443E+06	4.439E+06
Perfect	4.540E+06	4.524E+06	4.421E+06	4.237E+06	4.371E+06
0.4 inch	3.739E+06	3.734E+06	3.698E+06	3.723E+06	3.715E+06
1.0 inch	3.374E+06	3.374E+06	3.336E+06	3.346E+06	3.343E+06
2.0 inch	2.870E+06	2.870E+06	2.837E+06	2.834E+06	2.698E+06

In Table 5, there is some inconsistency in the predicted peak loads of the perfect shell from different shell element types. In Fig. 9, the predicted load-displacement responses of FEMs with the geometrically perfect 1.0-degree mesh using the different shell element types are presented.

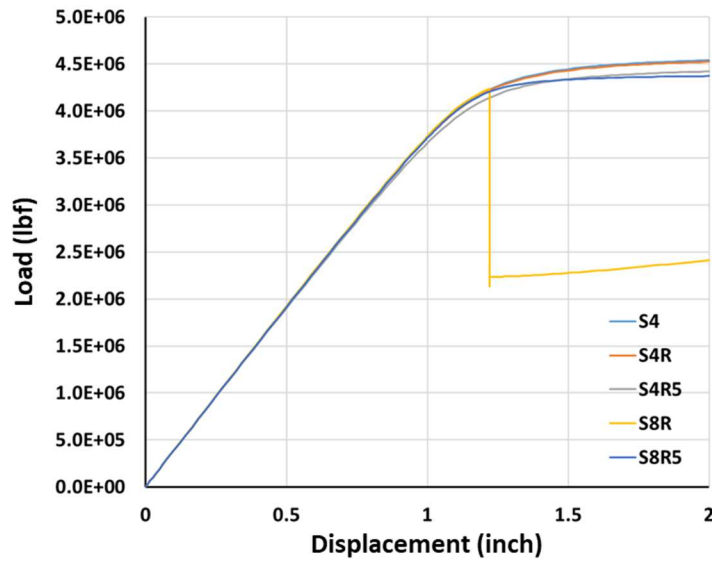


Figure 9. The load-displacement response of FEM for geometrically perfect shell.

As shown in Fig. 9, the peak load and post-buckled response of shell with S8R elements are different that the responses predicted with other shell elements. The elements with five degrees of freedom per node (S4R5 and S8R5) show a similar nonlinear response to the responses from the S4 and S4R elements. In contrast to the predicted responses of the perfect shell, the predicted peak load and postbuckled responses when geometric imperfections are included are all very similar for different element types. The peak loads and postbuckled responses of the FEMs with the 0.4-inch peak-to-peak amplitude of CTA8.1 imperfection are shown in Fig. 10. The predicted buckling loads and subsequent load drops are all very similar, however, the eight node elements (S8R and S8R5) show unstable responses and convergence problems after the load drop that lead to early termination of these analyses. These unstable responses and convergence problems are more apparent and occur earlier in the displacement for the 2.0-inch peak-to-peak imperfection as presented in Fig. 11.

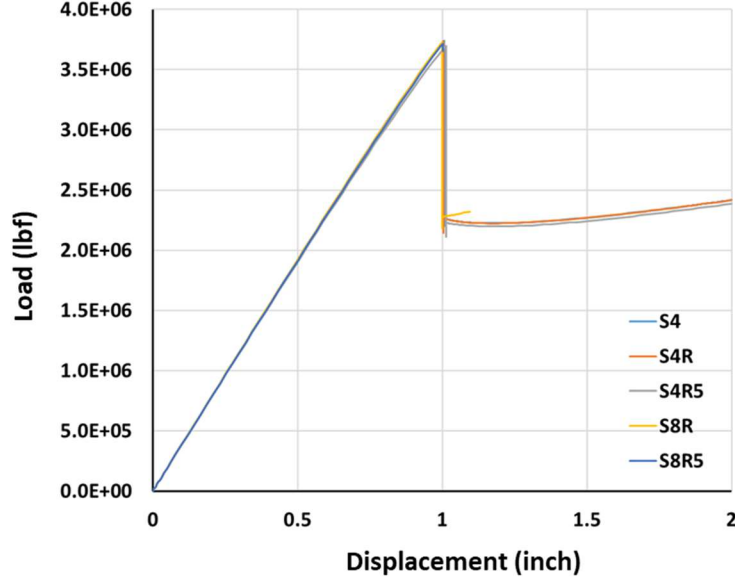


Figure 10. The load-displacement response of FEM with 0.4-inch peak-to-peak CTA8.1 imperfection.

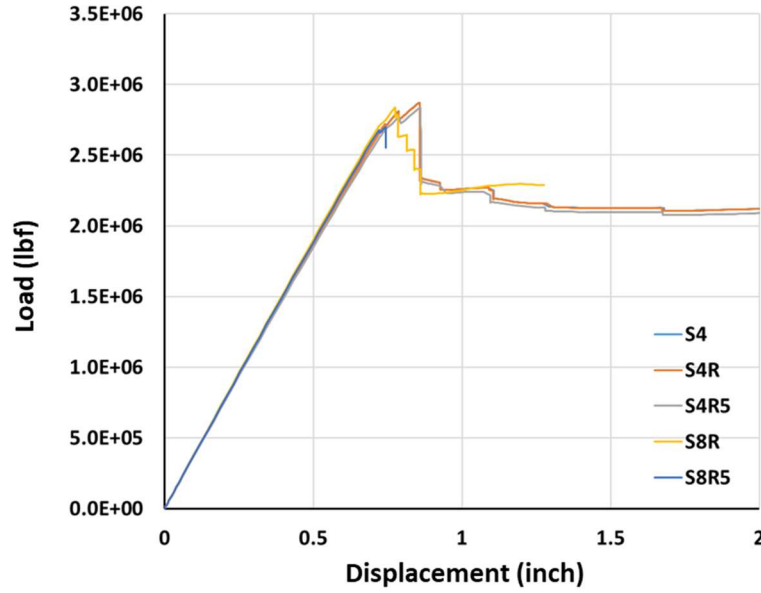


Figure 11. The load-displacement response of FEM with 2.0-inch peak-to-peak CTA8.1 imperfection.

Considering the postbuckling response, modeling effort, and the computational time, four-noded, reduced integration with hourglass control, finite membrane strain, general-purpose shell element, S4R, is recommended for discretizing the FEM of sandwich composite cylinder models.

V. Geometric Imperfection Sensitivity Analysis

In the proposed paper, a study of the sensitivity of the buckling response to a geometric (radial) imperfection with varied amplitude will be described. FEMs of sandwich composite cylinders with the three different cylinder lengths, shown in Fig. 1, were developed utilizing selected general-purpose shell element, S4R, and converged 1.0-degree mesh. Transient dynamic analyses were performed on FEMs for three different facesheet layups (quasi-isotropic, axially stiff, and fabric layups), different core thickness, and four different imperfection amplitudes (no imperfection, 0.4-inch, 1.0-inch, and 2.0-inch peak-to-peak imperfection) of the CTA8.1 geometric imperfection shape. The results

obtained from linear bifurcation buckling and transient dynamic analyses using the FEMs described above will be presented in the proposed paper.

VI. Concluding Remarks

The buckling response of shell structures is important for many aerospace design and analysis problems. This response can be simulated using finite element analyses provided such analyses are set up properly. During the presented study, the influence of general shell element type and mesh density on the predicted linear and nonlinear buckling responses were studied. Based on these findings and observations of the FEA results, a modeling approach was chosen and used to perform a sensitivity study of sandwich composite cylinders to geometric imperfection amplitude; results of this study will be presented in the proposed paper.

References

- [1] Hilburger, M.W., "Developing the Next Generation Shell Buckling Design Factors and Technologies," Proceedings of the 53rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference, AIAA Paper 2012-1686, Honolulu, HI, April 2012.
- [2] Schultz, M.R., Sleight, D.L., Myers, D.E., Waters, W.A. Jr., Chunchu, P.B., Lovejoy, A.E., and Hilburger, M.W., "Buckling Design and Imperfection Sensitivity of Sandwich Composite Launch-Vehicle Shell Structures," Proceedings of the American Society for Composites: Thirty-First Technical Conference, Williamsburg, Virginia, September 19–21, 2016. DEStech Publications, Inc., CD-ROM.
- [3] Knight, N.F., Nemeth, M.P., and Hilburger, M.W., "Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement," NASA TM-2004-213256, 2004.
- [4] Adan E. Stockwell, "A verification procedure for MSC/NASTRAN Finite Element Models," NASA CR- 4675, Hampton VA, June 1995.
- [5] Anon., ABAQUS™ 2016 Analysis User's Guide, Vols. I-V, Version 2016. Dassault Systèmes, Providence, RI, 2016.
- [6] Patran User's Manual, Version 2014, MSC Software Corporation, 4675 MacArthur Court, Suite 900, Newport Beach, CA 92660.
- [7] Marlett, K., Ng, Y., and Tomblin, J., "Hexcel 8552 IM7 Unidirectional Prepreg 190 gsm & 35% RC Qualification Material Property Data Report", CAM-RP-2009-015 Rev A, April 23, 2011 report for IM7/8552 Unidirectional tape.
- [8] Krueger, R., and Ratcliffe, J., "Analysis of Composite Panel-Stiffener Debonding Using a Shell/3D Modeling Technique," NASA/CR-2007-214879, NIA Report No. 2007-07, June, 2007.
- [9] Anon., "HexWeb Honeycomb Attributes and Properties," Hexcel Composites, TSB 120, Pleasanton, CA, November 1999.
- [10] Knight, N.F., and Rankin, C.C., "STAGS Example Problems Manual," NASA CR-2006-2144281, March, 2006.