FAILURE ANALYSIS OF LARGE COMPOSITE SPACE STRUCTURES USING MULTICONTINUUM TECHNOLOGY

Emmett E. Nelson
Firehole Technologies, Inc.
210 S. 3rd St, Laramie, WY 82070
Emmett@fireholetech.com

Jeffry S. Welsh, Ph.D.
US Air Force Operationally Responsive Space (ORS) Office,
3548 Aberdeen Ave SE, Kirtland AFB, NM 87117-5776

SUMMARY

In a combined experimental and analytical program, finite element predictions are compared to failure tests of large space structures under simulated flight conditions. The unique program provided an opportunity to benchmark structural analysis predictions verses rare experimental data. Results from advanced multicontinuum based failure simulation and traditional failure theories are compared to the experimental data. Additionally, techniques for accurate and efficient finite element representation of large structures are discussed.

Keywords: failure analysis, composites, multicontinuum, space structures

INTRODUCTION

The gap between industry composites design and analysis practices and academic research is truly daunting. While many academics are busy analyzing propagation of a matrix crack through a coupon, industry is consistently using heritage designs based on 30 year old methods using open-hole tension laminate allowables, treating the composite as if it were a black aluminum. This has lead to extreme conservatism in composite space structures and numerous sub-scale testing programs. Conservatism leads to obese structures; the cost of which is paid for by reduced mission performance in terms of payloads and increased launch costs. The cost of sub-scale testing is extremely slow production and extremely expensive development. One primary factor contributing to this extreme conservatism is a general lack of confidence in simulations driven by inaccurate analytical capabilities and an absence of structural failure data to benchmark against. The work described here is an effort to change the way development of composite space structures is viewed.

Pioneering efforts such as the World Wide Failure Exercise [1] have provided a platform for greatly improving composites failure theories in recent years. The exercise proved a general lack of accurate laminate failure predictions by the composites community. Since then, strides have been made in the accuracy of composites analysis, treating the failure progressively rather than as single event [2]. The development cost of composite structures is extremely high, thus, they are rarely tested to failure. With limited failure data to benchmark against, additional factors of conservatism are continually added to analysis.

To that end, a joint program between the Air Force Research Laboratory (AFRL) Space Vehicles Directorate, CSA Engineering, and Firehole Technologies, Inc., that benchmarks failure analysis of large composite aerospace structures against rare experimental failure testing was performed. Three test articles, shown in Figure 1, were chosen as candidates for the program. Each article was designed and manufactured for space flight on a specific launch platform using a traditional design and analysis approach. Prior to failure testing, the articles were subjected to an industry-standard static load qualification test defined as 125% of the specified launch environment (i.e. 125% of the flight loads). Subsequent failure testing involved continuing the loading process beyond 125% until the test article exhibited global structural failure. The analysis of two of the articles, CASPAR and the ISA will be described in this article.

Figure 1. The large structures test suite. A) CASPAR, B) ISA, C) PAF

The primary analytical goal of the program was to evaluate the current ability of industry to generate accurate failure predictions consistently for composite structures, thus improving confidence in results. With that, analysis was performed in an industry setting adhering to the requirements encountered daily by design and analysis practitioners. These include:

- Predictions for failure initiation, propagation, and ultimate failure were provided.
- Materials were characterized using available data generated from standard test methods and previously reported.
- All analysis was performed in a reasonable time frame, measured in weeks. This time included model development, run time, and documentation.

All analysis was performed with Helius:MCT™, a commercial multiscale composites failure simulation technology, in conjunction with the finite element solver Abaqus/Standard. Over the course of the analyses and tests, extremely valuable lessons were learned. In the end, using the combination of multiscale composites progressive failure simulation technology and advanced finite element modeling techniques, accurate failure predictions were achieved.

MULTICONTINUUM COMPOSITE FAILURE SIMULATION

Failure in composite laminates can be summarized by the progression of the following events which must be captured in order to achieve an accurate failure simulation of a composite. Local matrix failure, matrix failure propagation, local fiber failure, fiber failure propagation, and ultimate failure each cause a stiffness reduction, which in turn causes stress to redistribute. The sequence of events varies based upon the loading scenario.

Multicontinuum technology (MCT) is an efficient method for accessing stresses in the fiber and matrix constituents of a composite based on the strain decomposition of Hill [3]. The method was originally proposed by Garnich and Hansen [4], then adapted for failure analysis by Mayes and Hansen [5],[6]. Nelson and Hansen [7] reexamined the failure criteria and provided improved solutions and Firehole Technologies recently developed Helius:MCT[8]; an advanced simulation capability for composites based on these previous works with the following requirements:

- Failure simulation technologies must integrate with the finite element method.
- Material characterization must use only standard input parameters.
- Multiscale failure analysis must address matrix, fiber and ultimate failure.

- Material degradation must be captured appropriately.
- Computational efficiency is paramount.
- Convergence characteristics of the analysis must be robust.

The concept of a multicontinuum simply extends the notion of a continuum to reflect coexisting materials within a material point. Such an extension is natural in any case where there are two, or more, clearly identifiable constituents with drastically different material properties. Given a composite stress and strain field, and assuming linear elastic behavior in an incremental loading scheme, it is a straightforward matter to decompose the composite fields down to their constituent level stress/strain fields for the fiber and the matrix. The specific linear elastic equations applicable to this decomposition may be found in Mayes and Hansen [5].

Simple quadratic stress-based failure criteria (Eqs. 1-2), expressed in terms of transversely isotropic stress invariants of the constituents (Eqs. 3), are used to predict constituent failure within a lamina.

$$^{\pm}A_{1f} I_{1f}^2 + A_{4f} I_{4f} = 1 , \tag{1}$$

$$^{\pm}A_{1m} I_{1m}^2 - {}^{\pm}A_{2m} I_{2m}^2 + A_{3m} I_{3m} + A_{4m} I_{4m} - {}^{\pm}A_{5m} I_{1m} I_{2m} = 1 \quad , \tag{2}$$

where

$$\begin{aligned} I_1 &= \sigma_{11}, \\ I_2 &= \sigma_{22} + \sigma_{33}, \\ I_3 &= \sigma_{22}^2 + \sigma_{33}^2 + 2\sigma_{23}^2, \\ I_4 &= \sigma_{12}^2 + \sigma_{13}^2. \end{aligned} \tag{3}$$

In Eqs. 1 and 2, the $I_{i\beta}$ terms denote transversely isotropic stress invariants for each constituent, $\beta=f$ for fiber, $\beta=m$ for matrix. The coefficients A_{jf} and A_{jm}, leading the invariants, are constituent failure parameters, generally derived from experimentally determined composite ultimate strength data through correlation with the MCT decomposition.

FINITE ELEMENT MODELLING OF STRUCTURES

Industry standard modelling techniques for any of the three structures would use conventional shell or thick shell elements. Often, the laminate would be represented as a single homogenous solid. In areas where deemed important, sub-models would be created to provide additional detail. This iterative sub-modelling process is time consuming and makes performing a progressive failure analysis difficult or impossible.

CASPAR was the first test article. Initially for the blind failure prediction, solid shell elements were used to represent the structure. The failure predictions were not acceptable, as will be discussed in the following section. Perhaps the most insightful outcome of the study deals with element choice. Shell elements are computationally inexpensive and have proven accurate for structural stiffness predictions for years. In this study they were not used for failure analysis of composite structures. Consider the multiaxial stress state shown in Table 1, which was extracted from an element of one of the structure just prior to failure.

Table 1. Comparison of lamina to matrix stresses.

	σ_{11}	σ_{22}	σ_{33}
Lamina	-286.0	-20.0	-4.6
Matrix	-20.5	-18.5	-6.4

Examination of Table 1shows that the σ_{33} component is only 1.6% as large as the σ_{11} lamina stress, generally considered a negligible value. In the matrix, however, the σ_{33} stress is 31% of the σ_{11} lamina stress and certainly not negligible in this situation. Properly accounting for matrix failure and the resulting material degradation allows stress redistribution to be accurately captured. This is the foundation for ultimate failure predictions in a structure. With this in mind, finite element models must use multiple three-dimensional layered solid elements to represent the thickness of the laminate.

Although representing the thickness of a structure using multiple elements is not an advanced idea, in the sense of failure analysis of a large shell like aerospace structures, it represents a departure from industry standards. Generally more effort is required to build such a model, but recent advances in computer aided engineering makes this effort much more manageable. Detailed global finite element models of any of the three candidate structures were developed in under 80 hours of analyst time, from personal experience this is a significant improvement over the global-local methods.

LARGE STRUCTURES FAILURE ANALYSIS AND TEST PROGRAM

The following section contains details of the failure analysis and test of three structures. Certain details of each structures definition are missing from this paper due to proprietary information considerations specific to each structure's designer or manufacturer. Please contact the Air Force Research Lab, Space Vehicles Directorate, Kirtland, NM or the authors for further details.

CASPAR ANALYSIS AND EXPERIMENT COMPARISON

The current program started with qualification and failure testing of the Composite Adapter for Shared Payloads (CASPAR). This structure was designed to carry multiple payloads aboard the Minotaur IV launch vehicle by optimizing the available payload envelope. CASPAR consists of two symmetric, 74-inch diameter solid composite laminate cylinders. The composite laminate is approximately 60 plies thick, with each ply made from an industry standard carbon/epoxy pre-preg.

The major diameter tapers to an all-composite flange at both the forward and aft flanges. Each shell contains a single access door in the cylindrical section and four vent holes evenly spaced in the tapered regions. The failure load test case for CASPAR was designed based upon the worst-case condition from the qualification test, placing maximum compression opposite the access doors.

Blind Analytical Predictions

Prior to the experimental failure test, blind numerical predictions were provided using industry standard shell model and MCT. Matrix failure was predicted to initiate at 1296% of flight limit load (FLL), fiber failure was predicted at 1944% of FLL. The CASPAR structure was successfully tested to failure on April 14, 2008. The failure progression of the structure was a highly nonlinear event, with an abundance of localized matrix and fiber failures beginning at approximately 230% of flight limit load and continuing until the ultimate failure of the structure at 847% of flight limit load.

Model Description

In order to reconcile the differences between the test and analysis, a detailed finite element model of the CASPAR structure was created in Abaqus/CAE using 122,146 3-dimensional layered solid elements (C3D8R). Half-symmetry was used to reduce the size of the model. A picture of the CASPAR test assembly finite element model is shown in Figure 2.

Although the model is more detailed than an industry-standard finite element model, the entire progressive failure analysis ran in only 36 hours of continuous run-time on a standard 8-cpu desktop PC. The loading in the experiment was design to linearly scale worst case qualification loads until ultimate failure. Since the loading and structure are symmetric about the same plane, symmetric boundary conditions are applied to the FE model. To simulate the bolted connection between the aft adapter and the load frame, the nodes at the approximate locations of the bolt holes were pinned. A surface contact interaction was defined between CASPAR and the load frame at the bolted interface. The bolted connection between the forward adapter and the load head was modelled using nodal tie constraints.

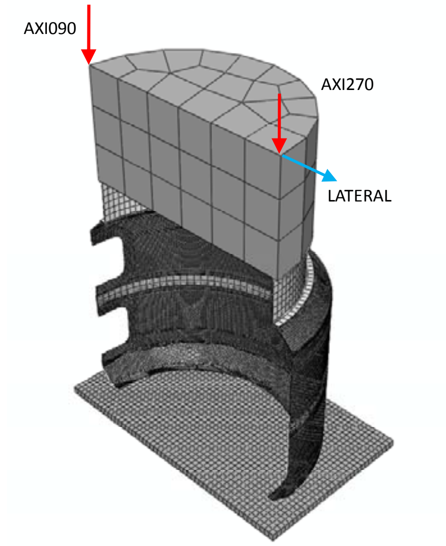

Figure 2. CASPAR finite element model.

Material Characterization

CASPAR was manufactured using an aerospace grade intermediate modulus carbon epoxy pre-preg tape. Material characterization for the analysis was performed using unidirectional lamina properties provided by the manufacturer [9]. Upon inspection of the test article, it was realized that wrinkling of the pre-preg tape existed in the region of the forward and aft radii. The wrinkling certainly creates fiber waviness, but at that time it was unclear the extent of the waviness and if it effects the entire thickness or only the outer plies. Although the fiber waviness may have a significant effect on the strength or stiffness of these regions, its effect was not included in the analysis presented here due to the uncertainty. Work is currently on-going to further characterize the effects and distribution.

Comparison of Analysis to Experiment

The analytical failure events reflected by the updated model are compared to the experimental events in Table 2. Analytical results show CASPAR's failure initiating with matrix failure in the lower radius region at ~260% of the flight limit load (FLL). With increased loading, failures continued to progress around the lower radius and also began to occur around the bolted joint at the bottom of the structure. The first fiber failure occurred at 740% FLL, and has progressed through the lower radius by 800% FLL. At 980% FLL, ultimate failure of the structure is predicted by a substantial discontinuity in the global load vs. displacement curve for the structure.

Table 2. Experimental and Analytical Failure Events for the CASPAR structure.

% FLL	Experimental Failure Event	% FLL	Analytical Failure Event
234	Initial matrix cracking noise	260	Initial matrix failure
319-469	Occasional matrix cracking noise	261-480	Matrix failure progression
470 +	Continuous matrix cracking noise	500 +	Rapid matrix failure progression
658	Fiber failure noise	740	First fiber failure
792	Lower radius failure	800	Fiber failure in lower radius
847	Ultimate failure	980	Ultimate Failure

During the experiment, the matrix cracking began at ~234% FLL and continued with increasing frequency until the ultimate failure of the article. The lower radius failed due to fiber failure at 792% FLL, with ultimate failure or collapse of the structure occurring at 847% FLL. Figure 3 shows the lower radius failure experimental and analytical results comparison; the red circle highlights the lower radius failure in the experiment.

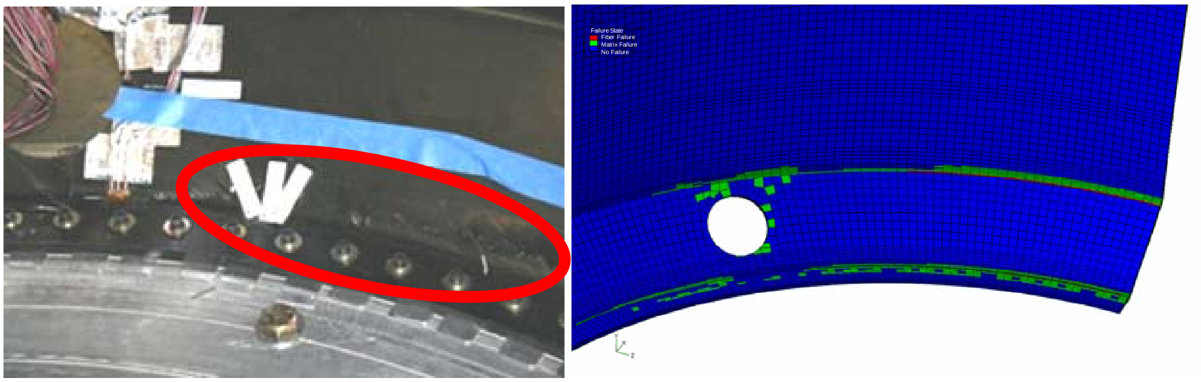

Figure 3. CASPAR lower radius failure, experimental (left) and analytical (right) results.

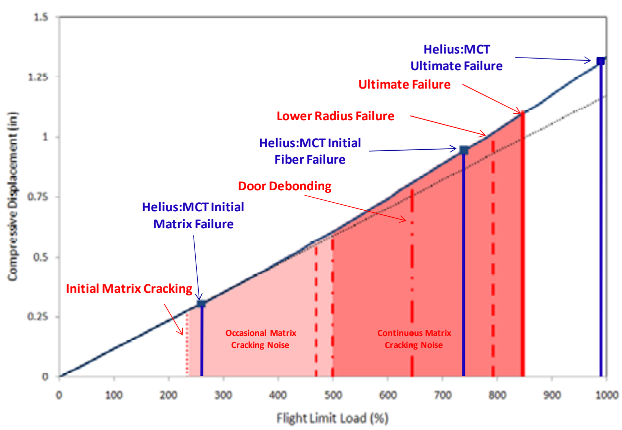

Figure 4. CASPAR failure progression. Red: Experimental observation, Blue: Analytical predictions.

The MCT analytical and experimental results show excellent correlation; initial matrix cracking is predicted within 11%, gradual global softening of the structure is predicted with the increase of matrix failures, failure of the lower radius is predicted within 1%, and the ultimate failure of the structure is predicted with 16%. Figure 4 shows the failure progression of CASPAR plotted on a MCT load vs. displacement curve (blue). Note the global softening of the structure when compared to a linear elastic response (dotted line).

Comparison to Industry Standard Modelling Techniques

The CASPAR analysis was also performed using the Hashin failure criteria and damage evolution algorithms provided by Abaqus/Standard. The model used in the analysis provided as direct of comparison to the Helius:MCT results as possible. A key difference is that continuum shell elements (SC8R) were substituted in the place of the layered solid elements because the Hashin criteria is not support for use with 3-D criteria.

Initial localized matrix failure was not predicted using the Hashin criteria until 1100% FLL, a load 30% greater than the ultimate failure of the structure. A linear response CASPAR was predicted past the first fiber failure (1300% FLL) until 1950% FLL where the analysis fails to find a converged solution.

Atlas V Inner Stage Adapter Analysis and Experiment Comparison

The second test article chosen was the Atlas V inner stage adapter (ISA). Functionally, the conical ISA sits atop the Atlas V common core booster, adapting the booster's 12.5-foot diameter to the upper stage 10-foot diameter. Both the forward and aft flanges are metallic and are attached to the composite cone via fasteners. Two thin composite face sheets surround an aluminium honeycomb core in a sandwich structure to make up the shell of the ISA. The composite used in the ISA is an industry standard carbon/epoxy co-cured on the core.

Two identical access doors exist opposite of each other in the structure. The first has a doubler containing a 3x build up of plies surrounding it. The second door was retrofitted using the original tooling and does not include the doubler region. The failure load on the ISA was designed based upon the worst-case condition from the qualification test, placing maximum compression over the second, retrofitted door.

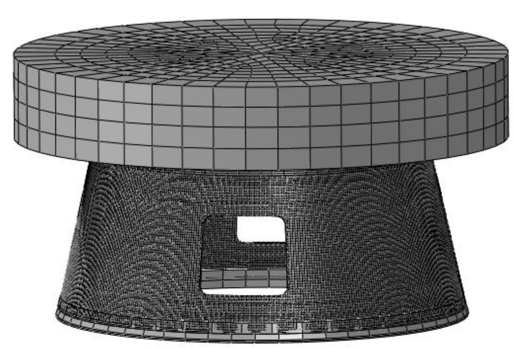

Figure 5. ISA finite element model.

Model Description

The ISA test assembly model included a composite conic adapter, a forward and aft attach ring, and a load head. The finite element model of the test assembly included 143,907 layered solid elements which represented the composite conic adapter. The remaining parts were represented with 13,696 regular solid elements. The FEA model of the ISA assembly is show in Figure 5.

The body of the ISA is a composite sandwich structure containing thin carbon/epoxy composite face sheets with an aluminum honeycomb core. In the model, the geometry is represented to the level of tolerance in the drawings. Composite plies were modeled on a ply-by-ply basis, allowing for accurate representation of material orientations. The honeycomb core was represented as a homogeneous orthotropic material using homogenized material properties. Loads in the experiment were designed based on a linear scaling of the worst case qualification loads for the structure. The aft ring of the ISA was bolted directly to the base plate. For modeling purposes, the base of the aft ring is constrained with fixed displacement in all 3 directions ($x = 0$, $y = 0$, $z = 0$). Nodal tie constraints were used to attach the multiple parts together in the assembly.

Material Characterization

Several coupon tests of the co-cured sandwich laminate were performed and the data was provided for the material characterization. Lamina material elastic constants and strengths were determined from these coupons using classical laminated plate theory.

Blind Analytical Predictions

Prior to the test of the modified ISA, a progressive failure analysis of the ISA was completed using the model as described above. The simulation completed 64 increments and 808 equilibrium iterations in ~ 39 hours of continuous run-time on an 8 CPU desktop PC.

Ultimate failure of the ISA was predicted to occur at 187% of the flight load. Figure 6 shows the load vs. displacement prediction with failure events. The predicted response of the structure remains linear through initial matrix failure (110%) and initial fiber (180%) until ultimate

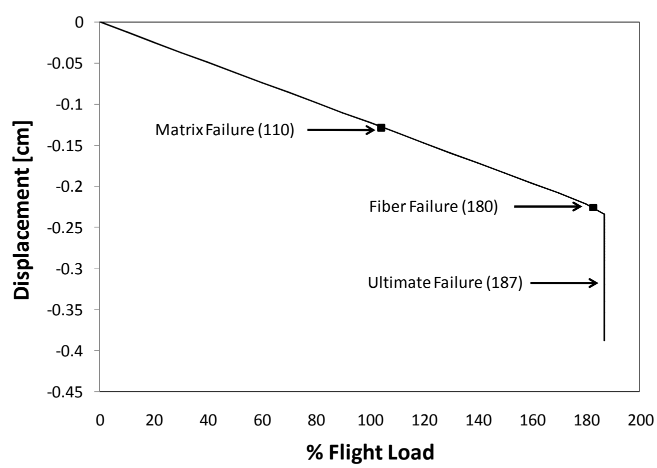

Figure 6. Helius:MCT blind failure predictions of the ISA.

failure of the structure was predicted at 187%.

Figure 7 shows a contour plot of the failure state at 187.68% of flight load. Blue is unfailed material, green indicates matrix failure, and red indicates fiber failure. In the analysis noticeable failure initiates at the upper door corners at 186.72%, progresses horizontally and rapidly around the circumference at 187.52%, and by 187.68% has propagated to the opposite door. Similar failures occurred in both the inner and outer face sheets. Failure of the structure as predicted by the MCT analysis was a rapid, nearly instantaneous event.

Experimental Results

On October 24, 2008 the ISA was successfully tested at the AFRL LASS Laboratory, Kirtland AFB, NM USA producing a catastrophic failure of the component at 183% of the normalized flight load. Failure initiated at the upper corners of the modified door and immediately propagated around the circumference of the article. Both the inner and outer face sheets experienced similar failures. Figure 7 shows the article after failure.

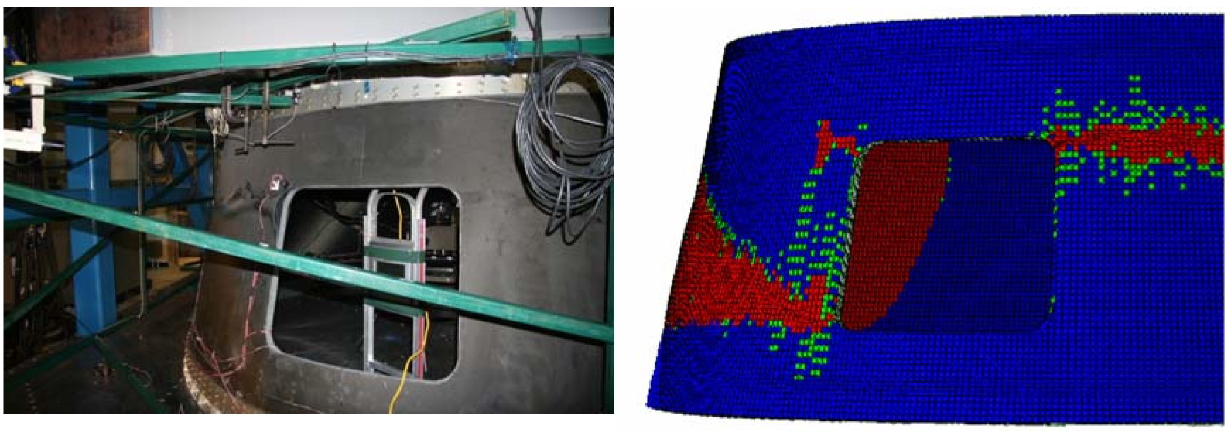

Figure 7. Experimental failure of the ISA (left), MCT failure predictions (right) blue: unfailed, green: matrix failure, red: fiber failure.

The blind failure prediction of the ISA using MCT successfully predicted the value of ultimate failure within 2.5%. The initiation location and propagation were also correctly predicted.

Comparison to Industry Standard Modelling Techniques

For comparison, progressive failure analysis was also performed in Abaqus/Standard using industry standard failure criteria: Hashin, Tsai-Wu, and Maximum Stress. Figure 8 shows the load vs. displacement response of the experiment compared to the predictions using the various techniques. The Hashin, Tsai-Wu, and Maximum Stress results were produced using industry standard continuum shell elements in Abaqus/Standard rather than the layer solid elements. The reason for this is these failure criteria are not supported for use with the layered solid element. Examination of

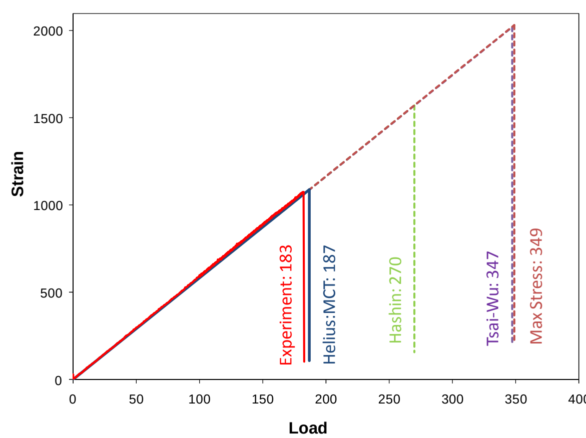

Figure 8. Comparison of MCT and traditional failure analysis.

the figure shows the failure predictions using industry standard techniques are very different than the experimental results. The Hashin progressive failure analysis predicted ultimate failure at a value of 270, 147% greater than the experimental value. Progressive failure analysis using the Tsai-Wu and Max Stress criteria produced similar results, predicting ultimate failure at 347 and 349, respectively, nearly twice the actual failure load.

DISCUSSION

Two large, flight qualified, composite space structures were failed under simulated worst case flight conditions. The ultimate failure event sequence for the two structures was dramatically different. One structure, the ISA, experienced a linear response until ultimate failure occurred in a single catastrophic event. This was because the primary load path in composite face sheets was in-plane. CASPAR experienced a long drawn out failure process due to a net bending load on the all composite flange. Initial failure began at a load of approximately one-forth of the ultimate failure load and continued to progress until ultimate failure was reached.

Analysis using Helius:MCT provided ultimate failure predictions within 2.5% for the ISA and 16% for CASPAR, proving the method is capable of realistically simulating the two different responses. The significance of these results should not be understated as it represents very accurate correlation to extremely complicated failure mechanisms found in large space structures. Moving forward it should give engineers confidence in analysis and provide a platform for improving structural design with composite materials. The authors attribute the excellent correlation to the decade long development of multicontinuum technology, the use of advanced modeling techniques, the appropriate use of a material degradation model, and accurate material inputs.

The differences between the CASPAR experiment and Helius:MCT analytical results may come from either fiber waviness that exists at the flange radii or the damage model used cannot appropriately capture the laminate stiffness change caused by the opening of a delamination failure. In future work, material properties that account for fiber waviness in the flange radii will be included in the analysis and cohesive elements that can capture delamination response will be used in areas where this failure mode is predicted.

Analysis using aerospace industry standard techniques provided inconsistent results in both cases. Several key features contribute to these results: the traditional failure criteria do not account for interactions between the fibers and matrix, traditional failure criteria can only be applied to in-plane stresses thus must be used with two dimensional elements, and the progressive failure models do not capture the failure progression robustly or accurately.

CONCLUSIONS

Failure analysis of two large space structures was conducted in an industry setting and the results are presented and compared to experiments. Results generated using an advanced composites failure technology Helius:MCT are compared to aerospace industry standards. The Helius:MCT ultimate failure prediction provide good correlation with the experiments while the industry standard analysis deviates significantly. The authors provide reasons for deviation between the two methods. Also provided is reasoning for differences in Helius:MCT results and the experiments and possible avenues to improve the analysis.

As expected, the program proved that the majority of composite aerospace structures are overdesigned and that traditional analysis techniques are not able to capture a composite structures' response to loading. However, it was also shown that a multiscale, progressive failure analysis is capable of capturing its response. The program provides a platform for improving confidence in analysis of composite structures, eliminating the black aluminum influence. If the approach is adopted, it will lead to mass optimization of structures and a reduction in sub-scale testing, providing improved structural performance and greatly reduced development time.

ACKNOWLEDGEMENTS

The research was sponsored by the Air Force Research Laboratory Space Vehicles Directorate under contract number FA 9453-07-C-0191 under the direction of Dr. Thomas Murphey.

REFERENCES

[1] M. Hinton, A. Kaddour, and P. Soden, *Failure Criteria in Fibre Reinforced Polymer Composites: The World-Wide Failure Exercise*, Oxford, UK: Elsevier, 2004.

[2] E. Nelson, A. Hansen, J. Welsh, and J. Mayes, "Recent Advances in Failure Predictions of Composite Laminates Using Multicontinuum Technology," Schaumburg, Illinois: 2008.

[3] R. Hill, "Elastic properties of reinforced solids: Some theoretical principles," *Journal of the Mechanics and Physics of Solids*, vol. 11, 1963, pp. 357-372.

[4] M.R. Garnich and A.C. Hansen, "A Multicontinuum Approach to Structural Analysis of Linear Viscoelastic Composite Materials," *Journal of Applied Mechanics*, vol. 64, Dec. 1997, pp. 795-803.

[5] J.S. Mayes and A.C. Hansen, "Composite laminate failure analysis using multicontinuum theory," *Composites Science and Technology*, vol. 64, Mar. 2004, pp. 379-394.

[6] J.S. Mayes and A.C. Hansen, "A comparison of multicontinuum theory based failure simulation with experimental results," *Composites Science and Technology*, vol. 64, Mar. 2004, pp. 517-527.

[7] E. Nelson, A. Hansen, and J. Mayes, "Failure analysis of composite laminates subjected to hydrostatic stresses: A multicontinuum approach," *Accepted as part of the Second World Wide Failure Exedrcise*, 2009.

[8] *Helius:MCT*, Laramie, WY: Firehole Technologies, Inc., 2009.

[9] G. Mahle, "CASPAR Materials properties," email communication, Jan. 2008.