Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement

Article · February 2004

Source: NTRS

Mark W. Hilburger
NASA

NASA/TM-2004-213256

Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement

Part 2: Structural Analysis Technologies and Modeling Practices

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

Michael P. Nemeth and Mark W. Hilburger
Langley Research Center, Hampton, Virginia

August 2004

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076-1320

NASA/TM-2004-213256

Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement

Part 2: Structural Analysis Technologies and Modeling Practices

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

Michael P. Nemeth and Mark W. Hilburger
Langley Research Center, Hampton, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

August 2004

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement

Part 2: Structural Analysis Technologies and Modeling Practices*

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems
Chantilly, Virginia

and

Michael P. Nemeth and Mark W. Hilburger
NASA Langley Research Center
Hampton, Virginia

Abstract

A technology review and assessment of modeling and analysis efforts underway in support of a safe return to flight of the thermal protection system (TPS) for the Space Shuttle external tank (ET) are summarized. This review and assessment effort focuses on the structural modeling and analysis practices employed for ET TPS foam design and analysis and on identifying analysis capabilities needed in the short-term and long-term. The current understanding of the relationship between complex flight environments and ET TPS foam failure modes are reviewed as they relate to modeling and analysis. A literature review on modeling and analysis of TPS foam material systems is also presented. Finally, a review of modeling and analysis tools employed in the Space Shuttle Program is presented for the ET TPS acreage and close-out foam regions. This review includes existing simplified engineering analysis tools as well as finite element analysis procedures.

As a result of the review and assessment presented herein, acceptance criteria for finite element modeling and analysis are proposed to assist analysts and managers in their decision making with regard to analysis results for ET TPS systems. The acceptance criteria led to the development of a modeling and analysis plan (MAP) for analytical efforts. The proposed MAP parallels test planning documents and reports required for experimental programs. Recommendations for short-term and long-term structural analysis technology improvements are also identified.

---

* Work of the first author sponsored by NASA Langley through GSA Contract No. GS-00F-0067M, NASA BPA L-71395D (Task Order 4).

This page left blank intentionally.

Table of Contents

This page left blank intentionally.

Introduction

Structural analysis technology has evolved significantly over the past 30 years. Computing technology has made leaps and bounds in the past 25 years – the personal computer first came out around 1981. It is important to keep these facts in perspective as early design work and analysis models are reviewed during accident investigations such as that performed by the Columbia Accident Investigation Board (CAIB). Design decisions were made during the development of the Space Shuttle with what were then state-of-the-art mathematical models of complex systems. Because technology has changed so rapidly in the recent past, these "state-of-the-art" analytical models and results soon lost that title. However, the Space Shuttle project resources apparently were not available or deemed necessary to re-assess the robustness of these modeling and analysis tools by verifying their assumptions as new analysis methodologies and computational technologies became available. Today, modeling and analysis tools and computing infrastructure far exceed what was available to designers of the original Space Shuttle system. As a result, detailed computational models that describe and predict the fundamental physics of the problem (*i.e.,* a high-fidelity analysis model) are not only feasible but also practical. The development of high-fidelity analysis models requires an understanding of the anticipated structural response and engineering judgment in the use of structural analysis tools. While finite element meshing of the structure's geometry is important, it alone does not result in a high-fidelity analysis model. Analysts need to be ever cognizant of the dependencies between accurate representation of the physics and decisions made in developing the engineering mathematical model (*i.e.,* finite element meshing, selection of material models including damage and delaminations, boundary conditions, and loading).

The Space Shuttle system involves three major subsystems: the orbiter, the external tank (ET), and the solid rocket boosters (SRBs) [1]. The main focus of the present study is the ET, which carries the liquid fuel for the Space Shuttle main engines (SSME) and is described in detail in Ref. 1. To date, three versions of the ET have been developed and used; that is, the standard weight, lightweight, and superlightweight tanks. However, the standard weight ET has not been in service for many years. Space Shuttle Columbia mission STS-107, that flew in January 2003, used a lightweight version of the ET designated as ET-93. During re-entry on February 1, 2003, Columbia disintegrated with the root cause being the loss of the left wing of the orbiter. The CAIB report (Ref. 2, pp. 49-58) indicates that the leading cause of the failure was initiated on ascent when a large piece of sprayed-on foam insulation (SOFI) from the ET near the left bipod region (forward attachment point between the orbiter and the ET) broke away and struck the wing leading edge (WLE), creating a hole. On re-entry, this hole provided a path for hot gases that exceeded 2500 °F and melted the internal structural components of the left wing, causing vehicle loss. As part of the Return-to-Flight Program, significant effort is being directed at mitigating the loss of SOFI from the ET (see Refs. 3-6) and at developing a better understanding of the foam behavior, properties, and failure modes. In addition, significant effort is being placed on developing more robust, mechanics-based analysis procedures for the SOFI material from a holistic perspective (thermal, structural, dynamics, gas entrapment and ingestion, material characterization). Improvements to material processing and non-destructive evaluation (NDE) procedures for the ET thermal protection system (TPS) foams are also being developed.

The present study has three primary objectives. The first objective is to evaluate past and present analysis tools and modeling procedures used to analyze the ET TPS. The second objective is to identify analysis tools and modeling procedures that may be applicable to ET TPS, including near-term capabilities to be pursued. The third objective is to assess the current understanding of failure modes for complex loading environments; to evaluate the capability of analytical tools, test databases, and rationale; to recommend analysis and testing needed; to identify near-term procedures and rationale to be implemented; and to identify long-term capabilities to be developed.

These objectives are accomplished as follows. First, background and overview information are presented for the ET TPS system that includes a description of known SOFI failure modes and damage mechanisms as they relate to structural modeling and analysis. Second, a literature review on modeling and analysis of foam material systems is presented. Third, a review of modeling and analysis tools employed in the Space Shuttle program for the structural and thermal analysis of the ET TPS foam acreage and closeout regions is performed. This last review includes examination of existing simplified engineering analysis tools and finite element analysis procedures. Trends and future directions for ET SOFI modeling and analysis are discussed. As a result of these reviews and assessments, acceptance criteria for finite element modeling and analysis for the ET TPS foam system, and for engineering systems in general, are proposed and described. The acceptance criteria are defined in terms of a modeling and analysis plan (MAP). The MAP is intended to assist, not burden, analysts and managers in their decision-making, with regard to the accuracy of analysis results, and to provide engineers with a systematic procedure for defining and documenting the pedigree of analysis models. Finally, short-term and long-term recommendations are identified for ET TPS structural analysis technology investment.

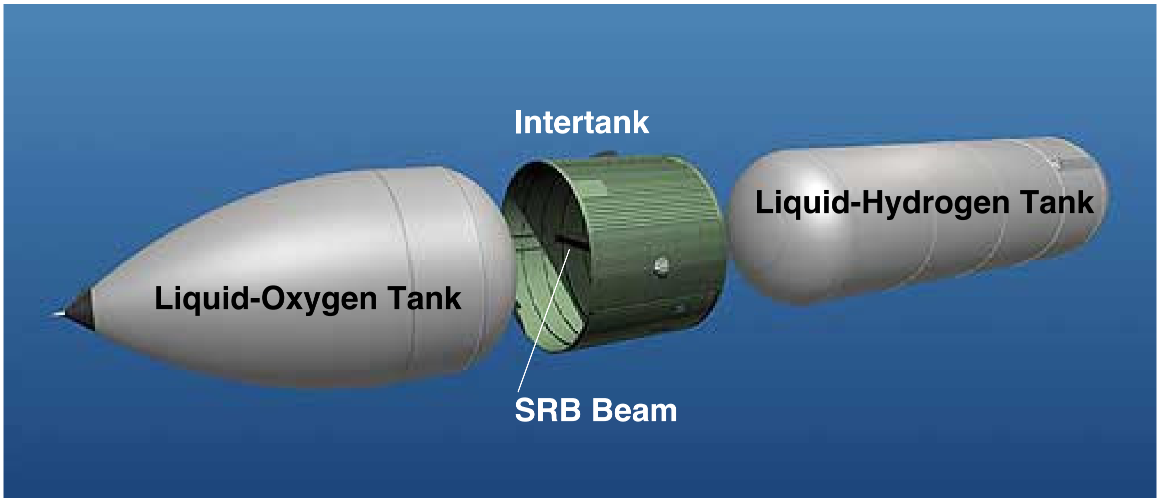


Figure 1. Major components of the external tank.

Background

The major ET components are shown in figure 1 and include the 385,265-gallon liquid-hydrogen (LH_2) tank (aft lower tank), the 143,351-gallon liquid-oxygen (LO_2) tank (forward upper tank), and a stiffened cylindrical structure, referred to in the Space Shuttle program as the intertank, that connects the two liquid-fuel tanks. The ET is approximately 153.8 feet long and 27.6 feet in diameter. The basic ET structure is made of 2024, 2195, 2219, and 7075 aluminum alloys.

Figure 2. Cutaway of the Y-shaped joint between the liquid hydrogen tank and the intertank structure.

The LO_2 tank is a welded assembly of machined and formed panels and rings and is maintained at –297 °F prior to launch. The LH_2 tank is a welded assembly of barrel sections, I-shaped ring frames, and dome sections and is maintained at –423 °F prior to launch. The juncture for the LH_2 tank and the intertank is a circumferential Y-shaped joint, as shown in figure 2. The intertank is a 22.5-foot-long hollow cylinder made of eight stiffened aluminum alloy panels bolted together along longitudinal joints. Two of these panels, referred to as thrust panels, are integrally machined, blade-stiffened panels that react the SRB thrust loads. In addition, the intertank is spanned diametrically by a massive beam, referred to as the SRB beam (see figure 1), that also reacts the SRB thrust loads. The intertank also provides reaction structure for the SRB mounting points and orbiter mounting points (*i.e.,* bipod region). Orbiter and SRB attachment points are located also on the ET aft end. Empty, the superlightweight and lightweight ETs weigh approximately 57,800 and 66,000 pounds, respectively, and when loaded with propellants at launch they weigh approximately 1,674,000 pounds. The ET separates from the orbiter after main-engine cutoff (MECO) at approximately eight minutes and 30 seconds after liftoff. The ET is the only fully expendable element of the Space Shuttle system.

During propellant loading, the intertank, connecting the LH_2 and LO_2 tanks (see figure 1), is purged with gaseous nitrogen GN_2 to prevent inadvertent, catastrophic mixing of the LO_2 and LH_2. During this process, liquefying of the GN_2 is likely to occur in the lower, uninsulated portion of circumferential Y-shaped joint (see figure 2). If liquid nitrogen accumulates in the bottom of the Y-shaped joint prior to launch, it could be cryoingested through leak paths into the adjacent SOFI and possibly solidify. Then, at launch and during ascent, as LH_2 fuel is burned, the LH_2-tank temperature rises in regions where the fuel level drops. This temperature increase causes solidified nitrogen to liquefy and liquid nitrogen to "flash" evaporate, producing a pressure increase within the SOFI subsequently leading to shedding of the SOFI material.

Prior to Space Shuttle Columbia flight STS-107, SOFI used for the ET TPS was not intended to be a structural member. The TPS was used to control the propellant boil-off rates, minimize formation of frost and ice, protect the ET against aerodynamic heating, and ensure propellant quality. Thus, the understanding of the ET TPS foam failure modes and damage mechanisms was limited, and primarily focused on, bondline delamination failure for the ET acreage TPS foam. Bondline delamination failure was believed to be the primary contributor to observed losses in thermal protection for the acreage TPS [6]. In contrast, spallation or small-scale cohesive failure (known as "popcorning") of the SOFI caused by entrapped and ingested gases (water vapor, Freon, carbon dioxide, and air) was identified and described by del Casal [7] using a one-dimensional model for flow through a porous media. This early paper (1983) suggested that spallation due to vapor-pressure buildup in the SOFI was a significant source for debris generation.

Since the Space Shuttle Columbia flight STS-107 accident, the ET SOFI has been studied both structurally and thermally by using large-scale finite element analysis models. These detailed two- and three-dimensional analyses are dependent on the characterization of the foam material – characterization of mechanical properties, as well as, failure modes and damage mechanisms. Considerable testing of coupon- and element-level specimens has been recently performed and continues in order to obtain these data. The challenge to the engineering teams involved in these studies is increasing the confidence level of the ET TPS system through additional testing and results obtained using new computational models.

ET TPS Overview

The ET TPS is applied at the NASA Michoud Assembly Facility (MAF) in New Orleans using a combination of automated-spraying and manual-spraying operations. Most of the SOFI is applied using automated-spraying procedures for the acreage TPS. Manual spraying procedures are used for close-out regions between and on automated-spray regions such as the LO_2-tank/intertank flange and LH_2-tank/intertank flange regions, the bipod ramps, the LO_2 feedline and supports, and the protuberance air-load ramps (PAL ramps) on the LO_2 and LH_2 tanks. The SOFI is an evolving material system with many process-related variables to control. As described in Ref. 2 (page 51), three spray-on foams are used on the ET for thermal protection. The NCFI-24-124 foam (a polyisocyanurate foam applied with an HCFC-141b hydrochlorofluorocarbon blowing agent) is applied to most areas of the LO_2 and LH_2 tanks (*i.e.,* most of the acreage area of the ET). The NCFI-24-57 foam (another polyisocyanurate foam applied with an HCFC-141b blowing agent) is applied to the lower LH_2-tank dome. The BX-250 foam (a polyurethane foam applied with a CFC-11 chlorofluorocarbon blowing agent) is applied to domes, ramps, and areas where the foam is applied manually (*i.e.,* close-out regions).

Preparation and application of the SOFI involves several steps that are subject to many variables. Constituent chemistry, mixing ratios, application environment (humidity, barometric pressure, temperature), technician skill and procedures, foam layer curing, aging (time from application to flight, storage environment), exposure to environment prior to flight and other factors are representative of these variables. The SOFI material exhibits a cellular structure characteristic of many foam materials (*e.g.,* see Ref. 8). The foam is a closed-cell structure with small pores, filled with air and blowing agents, that are separated by a thin-membrane cell wall made of the foam's polymeric component [2]. However, unlike some commercially available foams, the SOFI material system exhibits significant variability as a result of processing and application to the ET.

Thickness variations of the SOFI along the ET occur because multiple passes are required in the spraying procedure (automated or manual). An epoxy primer is applied between the first foam coat and the metallic ET substrate to enhance bonding. Where the foam is applied over cured or dried foam, a bonding enhancer called Conathane is first applied to aid adhesion between the two foam coats [2]. Each pass or foam coat generates a "knit-line" marking the boundary between sprayed layers, as indicated in figure 3. Additionally, the SOFI material rises the most in the direction with least resistance (the spray direction), which causes material anisotropy. Regions with voids and other flaws can result during these multi-pass applications and set-up or curing of each foam layer. A description of twelve different flaws that have been found is found in Ref. 9. As such, the final ET TPS material system is anisotropic and non-homogeneous. In addition, its mechanical behavior is highly temperature dependent. The SOFI is essentially a non-continuum in its material structure, which includes cell walls, pressurized pores, and voids. The internal cell structure of the NCFI foam material is a closed-cell foam, as shown in figure 3. However, tensile tests indicate that the SOFI material does appear to respond in a near-continuum manner and, as a result, ET TPS modeling and analysis efforts conducted to date simulate the SOFI by using a continuum-modeling approach.

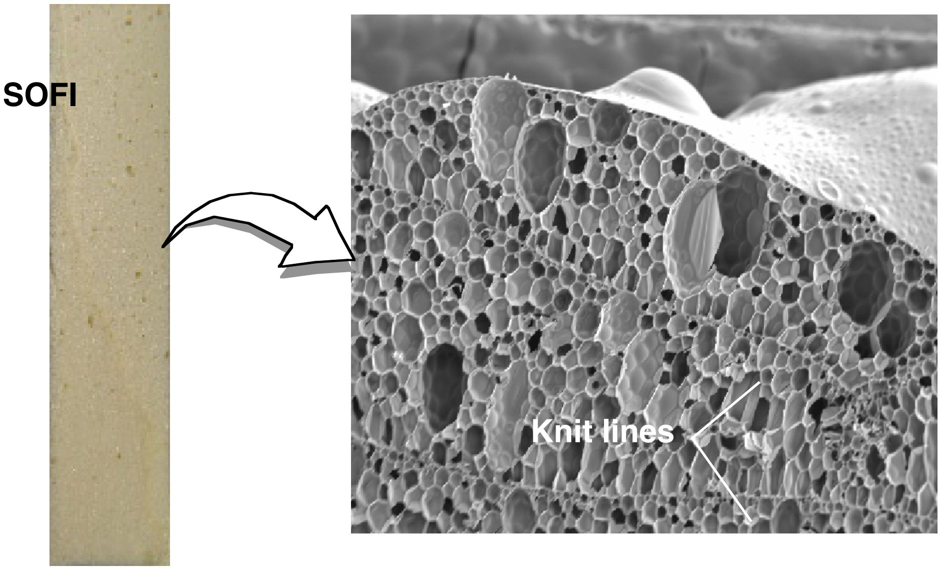


Figure 3. Scanning electron microscopy photomicrograph (30X), from Ref. 6, p. 94.

ET TPS Failure Modes

The operating environment of the ET is a complex combination of thermal conditions, aerodynamics heating, aerodynamic and acoustic loads, and mechanical loads. Internally, the ET shell structure contains various amounts of cryogenic liquid fuels, while externally the ET is exposed to ambient air temperature and pressure prior to launch and aerodynamic heating and pressure, vibro-acoustic loading, and external pressure gradients during ascent. For weight savings, the structural capacity of the ET shell material is highly exploited. The structurally tailored shell walls of the LH_2 and LO_2 tanks are quite thin – in some regions the ratio of shell radius to shell thickness is much less than 0.001. As a result, the ET shell wall (substrate) deformations can be significant and may be a secondary contribution to many of the ET TPS failure modes.

The primary ET TPS failure modes appear to be a result of consequences stemming from the SOFI application process and/or the overall ET system-level design. Known failure modes for the ET SOFI system include:

- *Substrate debond* – This failure mode (see figure 4) can result from poor adhesion of the SOFI to the ET aluminum substrate surface. Moreover, peel stresses caused by the thermal gradient and mismatch in thermal expansion between the substrate and the TPS, large-magnitude ET shell wall deflections (*i.e.,* hoop stretching of the cellular structure), and rapidly varying severe ET shell wall deflection gradients ("accordion mode") in the axial and/or hoop directions are major contributors to this failure mode. A key concern of an accordion mode is the high curvature of the local, short-wavelength deformation modes. These deformation modes can result in local compression of the foam and potentially create pockets of crushed foam that debond from the substrate. Additional effort is needed in the near term to better characterize substrate/SOFI delamination and debond.

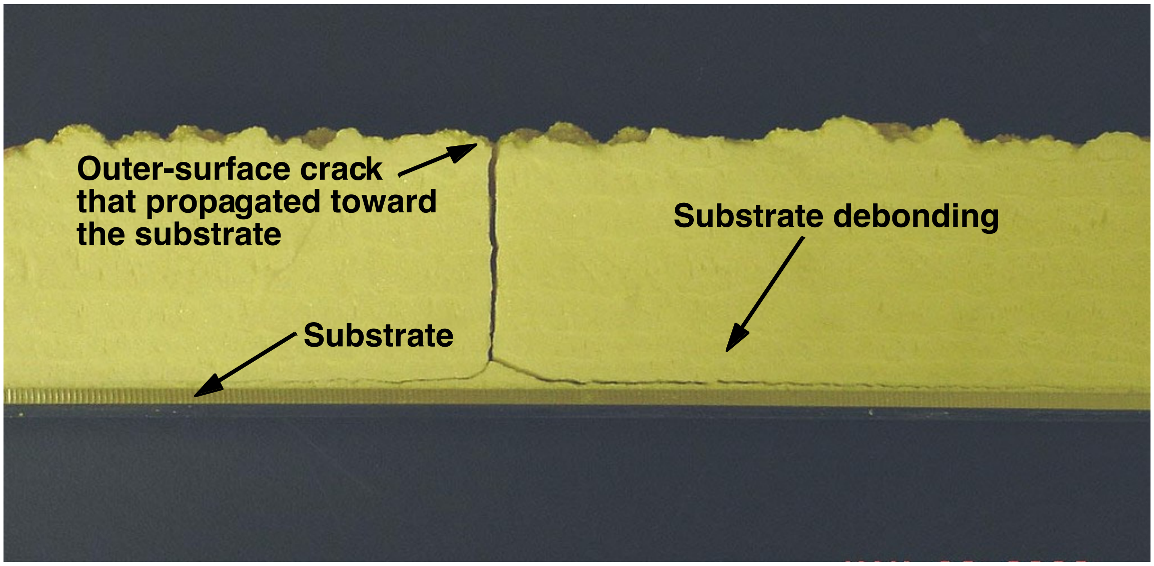


Figure 4. Substrate debonding failure mode.*

- *Divoting* – This mode is a large-scale, cohesive failure mode that has been observed on several flights (*e.g.,* see figure 5). This failure mode can result from entrapped gas within the foam cellular structure, cryoingestion† of condensed liquid nitrogen from the intertank, and cryopumping‡ of energy sources into local voids in the SOFI or substrate debonds near the substrate surface. As the ET ascends, it experiences aerodynamic heating, heating of the substrate from the drop in propellant level, and a reduction in external pressure, caused by the reduction air density, that tends to hold the SOFI to the ET. The heating causes rapid expansion of the entrapped gases and energy sources that build up pressure that leads to foam shedding.

---

[*] From SDS 6113 TPS Verification Team Technical Interchange Meeting, August 13-14, 2003, chart 20.

[†] *Cryoingestion* (see Ref. 3) refers to the ingestion of liquid nitrogen from the intertank through the circumferential "Y" joint where the LH_2 tank mates to the intertank. Pooled liquid nitrogen may come into contact with the LH_2 tank causing it to solidify. Then at launch and ascent, as the LH_2 fuel is burned, the LH_2 tank temperature rises causing the solidified nitrogen to "flash" evaporate and possibly causing the foam to break off or divot.

[‡] *Cryopumping* (see Refs. 3 and 6) refers to the process where air fills voids or debonded regions through transverse cracks. As this air reaches the cryogenic tanks, it solidifies and then on launch and ascent, the entrapped air expands rapidly possibly causing the foam to break off.

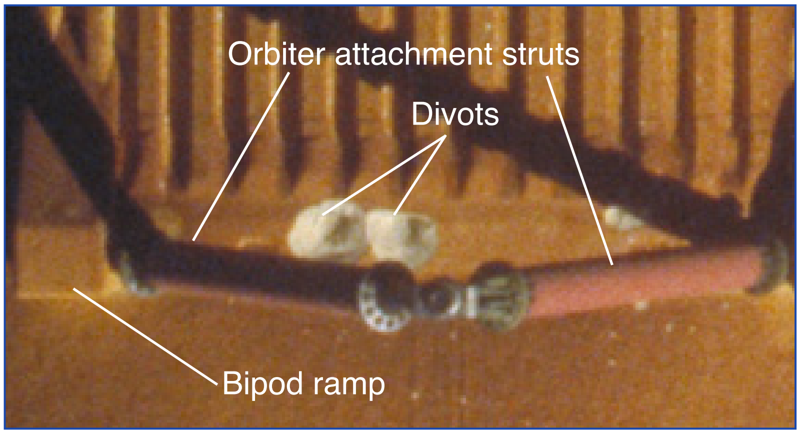


Figure 5. Divoting failure mode shown on post-separation photographs.*

- *Popcorning* – This failure mode is a small-scale, cohesive failure mode that can result from small voids near the surface of the SOFI. In this case, if a vent path is absent or insufficient to relieve the gas pressure, a void located away from the substrate, in the through-the-thickness direction, that has entrapped gas in the foam cells causes a small popcorn-size piece of foam to "pop" off as the external pressure drops during ascent. Divoting and popcorning are related failure modes (see figure 6) stemming from rapid expansion of entrapped gases during heating. They typically differ in the through-the-thickness location of the void and the nature of the entrapped energy source. For divoting, the voids are usually near the substrate surface and generate debris with larger mass than the debris shed by a single popcorning event. For popcorning, the voids are typically small and near the free surface of the SOFI. This failure mode was identified and studied in Ref. [1].

- *Delaminations* – This failure mode can result from SOFI disbond along knit lines between layers of the foam or from a coalescence of local failures of the SOFI cellular structure.

- *Transverse cracking* – This failure mode, shown in figure 7, can result from local SOFI failures, substrate flexure, or from entrapped gas within the foam cellular structure. This mode can also serve as a relief mechanism for the delaminations, divots, and substrate debond wherein a transverse crack provides a leak or vent path for entrapped gas to escape.

---

* Provided by the NASA Independent Technical Assessment Team for the External Tank.

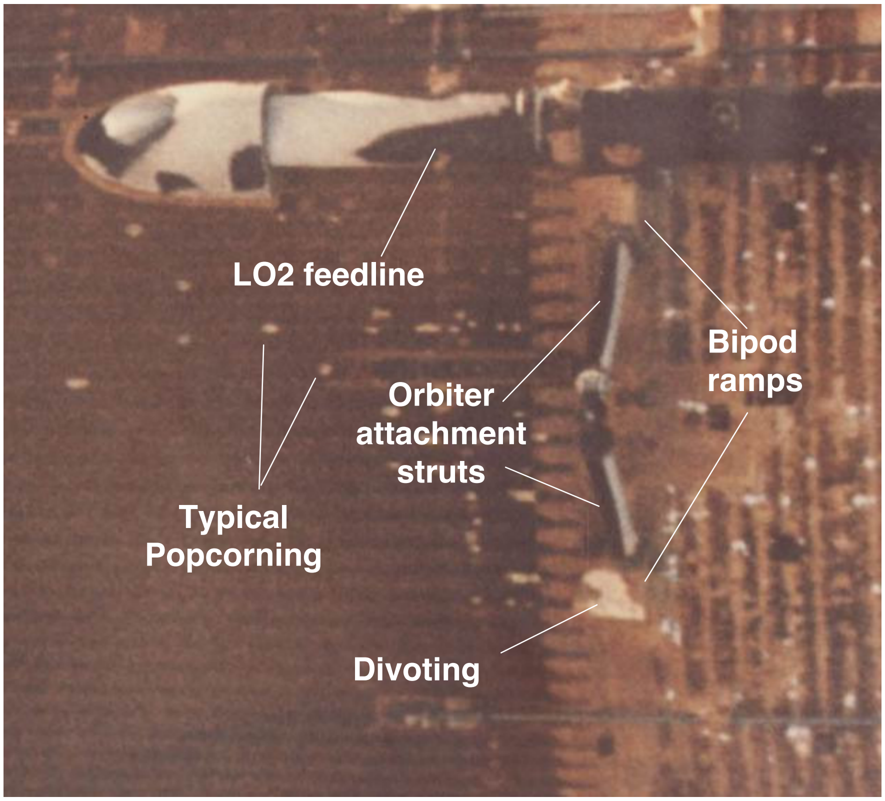


Figure 6. Popcorning and divoting failure mode shown on post-separation photographs.*

Figure 7. Transverse-cracking failure mode.*

---

* Provided by the NASA Independent Technical Assessment Team for the External Tank.

- *Fragmentation/crushing* – This failure mode can result from external local impact of free-stream debris or ice that strikes the ET TPS and breaks off a SOFI fragment. This failure mode is dependent on the incident angle and geometric shape of the impacting debris. An impact normal to the ET surface would most likely compress the SOFI and not cause a fragment to be released. A glancing impact could potentially "plow off" a sizeable SOFI fragment. A related event is the local crushing of the SOFI during the close-out process when technicians are required to stand on the ET itself. Padding is installed for the workers to access the closeout regions; however, local surface crushing of the foam may possibly occur.

- *Strength failure* – This failure mode is essentially embodied in every failure mode and can result from external loading (discrete or distributed) that causes a material strength failure of the cellular foam system. This mode is singled out primarily to identify the strength-failure mode of the cell wall itself. Vibro-acoustic loading and aerodynamic pressure loading are two sources of the external loading that potentially contribute to such a strength failure. The load paths for a cellular material are essentially the cell walls of the foam. Processing controls to maintain uniform cell structure would reduce the variability of material strength by controlling cell-wall size and cell-wall structure over the area of application.

- *Aero-shear failure* – This failure mode is indicated in figure 8 and can result from external aerodynamic loading caused by TPS protuberances in the flow field. Protuberances such as the bipod ramps, PAL ramps, ice/frost ramps, and feedlines are inherent to the vehicle and can result in a surface shear loading on the TPS. In addition, ET TPS surface roughness, manufacturing/processing variations, and small surface defects result in surface shear loading. Efforts to reduce and/or eliminate TPS protuberances into the flow field are under consideration – in particular the bipod and PAL ramps.

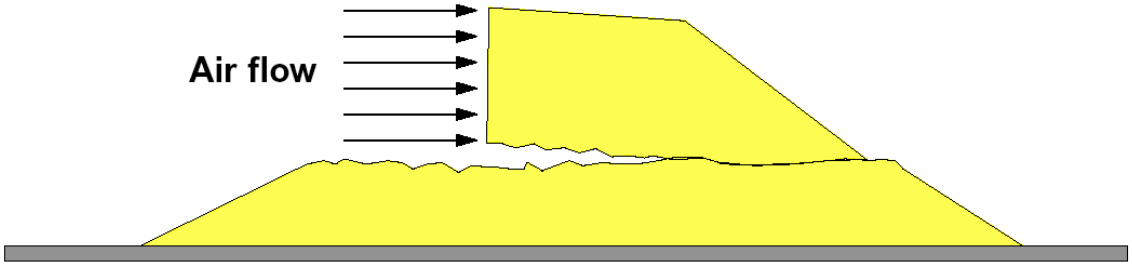


Figure 8. Depiction of aero-shear failure mode.

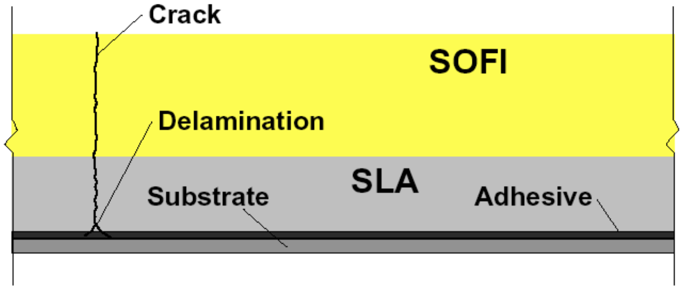
- *Fatigue* – This failure mode is related to the high-frequency loading caused by pulsating external loads. Possible sources of such loading are in the vicinity of the attachment points between the ET and the orbiter or between the ET and the solid rocket boosters (SRBs). Flight conditions generate a time-varying high-frequency loading condition that can potentially contribute to a rapid flexing of the ET substrate. This flexing can cause fatigue of the bond between the foam and the ET substrate or cause local cellular fatigue failures that create voids. Such voids could then result in “popcorning” or “divoting” as a result of the external loading and a cascade of local material-strength failures. Another source of low-cycle fatigue that may precipitate failures is the propellant tanking/de-tanking process that often occurs prior to launch.

ET TPS Analysis Review and Commentary

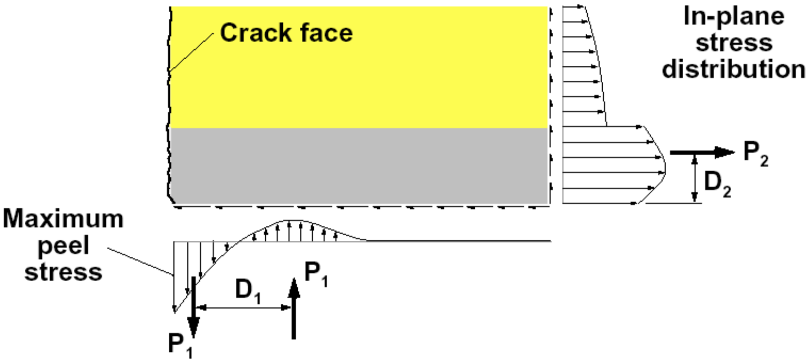
As mentioned previously herein, the ET SOFI was originally not considered as a structural element of the ET system and did not receive a thorough structural characterization during the original design process. It was primarily viewed as a thermal protection system only, with the underlying assumption that the SOFI adhered to the ET substrate. Consequently, prior to the Space Shuttle Columbia flight STS-107 accident, only simplified engineering analyses were performed for the ET TPS material, and then only to the acreage SOFI, not the close-out regions. Since then, finite element models that include the SOFI have been developed and solved as part of the CAIB investigation and subsequent redesign effort to return to flight in a safe manner. This section presents a brief overview of the pre-STS-107 ET SOFI structural analysis effort and the post-STS-107 design and analysis efforts that are primarily finite element based.

Simplified Engineering Analyses

The simplified engineering analysis tool used for the acreage SOFI is a tool developed by Lockheed-Martin named TPSMOM - an acronym for TPS moment calculation (see Refs. 10 and 11). This analysis tool postulates that a through-the-thickness surface crack or free edge extends through the TPS foam to the ET substrate, causing a bondline delamination as shown in figure 9. The resulting in-plane normal-stress distribution caused by the mechanical and thermal loads varies through the SOFI thickness and gives rise to a moment depicted by the resultant in-plane force P_2 acting at a distance D_2 above the substrate. For equilibrium, this moment must be balanced by a couple generated from the interlaminar peel stress distribution P_1D_1. The in-plane shear stress distribution provides the equilibrium force balance for the in-plane normal-stress resultant and the interlaminar peel-stress resultant. Once the moment P_2D_2 exceeds the test-determined allowable value, bondline failure is assumed to occur. This failure implies that the moment P_2D_2 exceeds the strength of the SOFI that is represented implicitly by the moment P_1D_1.



(a) Schematic of crack and delamination in ET TPS for TPSMOM.



(b) Moment analysis of bondline delamination in TPSMOM.

Figure 9. Overview of TPSMOM analysis model.[*]

Recent studies on pull-off tests [12] and wedge-peel tests [13] for adhesive materials offer alternative testing methods to characterize adhesive strength. These studies also propose analytical methods for their study. Developing accurate, closed-form analytical tools for the ET TPS that account for substrate curvature, gaps or free edges, debonds, and biaxial effects would significantly enhance the design process. Selected references are described next that have potential to being extended for ET TPS SOFI analysis.

Sun, Wan, and Dillard [12] describe a closed-form analytical solution for the strain-energy release rate for a thin film in a pull-off test configuration. Extensions to thicker regions or the isolation of the ET TPS adherent may provide a useful tool for assessing the influence of local debonds.

Ferracin, Landis, Delannay, and Pardoen [13] present results from a numerical study of the wedge-peel test to study the cohesive-zone properties of an adhesive layer. A fracture-based criterion is used in the assessment and accounts for local curvature effects. Cui *et al.* [14] studied the finite element modeling requirements for an accurate peel-stress prediction by using the von Mises critical strain as the failure criterion. Extensions

---

[*] SDS 6113 TPS Verification Technical Interchange Meeting, August 13, 2003.

of these approaches to the ET TPS appear to have merit and warrant further consideration.

Sun and Tong [15] examined curved beams with debonded piezoelectric sensor/actuator patches. Their analytical approach for treating curvature and debonds has potential for extensions to plates and shells. While the piezoelectric aspects of their paper are not of direct interest, the paper does describe an analytical approach for adhesively bonded curved beams.

Olia and Rossettos [16] present a plane strain analysis of adhesively bonded joints with gaps and subjected to bending. This paper presents an analysis procedure for bonded joints that should have potential in the application to the ET TPS configuration. Concepts from this paper may be carried over into a new approach applicable to ET TPS analytical models with extensions provided for local defects and curvature. Other approaches used in the analysis of adhesively bonded joints may also provide insight for debond and delamination modeling and analysis efforts and should be examined for applicability.

Finite Element Analyses

Finite element analysis tools have become commonplace in engineering analysis and design. With the advancement of computer hardware (*i.e.,* CPU processor speed, RAM memory, and high-capacity secondary storage devices) and the development of engineering design software with graphical user interfaces (GUI) and high-performance equation solvers, large-scale finite element analyses and simulations are becoming more integrated within the design process than ever before. The geometric description of a component can be readily defined via computer-aid design tools in IGES- or STEP-format and a specific finite element spatial discretization generated via computer-aided engineering tools to generate nodal coordinates, element connectivity, and other input data for use in a finite element analysis. Many commercial finite element codes are available for linear and nonlinear stress analysis including MSC/NASTRAN[*], HKS/ABAQUS[†], ANSYS[‡], and LSTC/LS-DYNA[§]. These codes provide a vast array of modeling and analysis options for simulating the structural response of complex engineering systems.

However, there is a need to assess finite element results critically based on a solid understanding of the assumptions embedded, either explicitly or implicitly, in those numerical models. It should be clearly understood that a *large-scale* finite element model does not equate to a *high-fidelity* finite element model. The former implies a large number of finite elements and hence a large computational problem. The latter implies that the mathematical model adequately and correctly represents the physics of the systems for its intended purpose. It is true that the latter generally implies the former

---

[*] http://www.mscsoftware.com

[†] http://www.hks.com

[‡] http://www.ansys.com

[§] http://www.lstc.com

(*i.e.,* a high-fidelity model *is* often a large-scale model); however, the converse is not true (*i.e.,* a large-scale model *is not* automatically a high-fidelity model).

The idea of a high-fidelity analysis relates to the concept of *robustness* in the engineering analysis. A precise definition of robustness is very much dependent on the context. Within the engineering analysis context, robustness can be defined as a mathematical model of an engineering system that accommodates variability in design parameters and accounts for modeling assumptions and uncertainties that affect system response and performance. A robust analysis model is then able to address system response sensitivities to variability in design parameters and to modeling decisions using defined performance metrics and solution-accuracy indices. The challenge to the analyst is to define the performance metrics and accuracy indices so that quantitative measures can be used. A robust analysis can then be defined as an analysis that accurately captures the physics of the problem, can be used to assess system sensitivities to problem uncertainties, and can be used in risk mitigation.

Increased reliance on analysis for design verification and certification provides the impetus for robust analyses and increases engineering accountability of the results. This is not to say that the current approach is unable to provide accurate results, but to say that the basis and context of the analytical results must be communicated at all reporting levels. Often, preliminary results find their way to very high levels while all the associated assumptions and caveats do not. The temptation to push forth detailed three-dimensional finite element models and results from these models that “look” like the component and have “color” contour mappings of response parameters that are “animated” for visualization must be carefully assessed because of the potential far-reaching consequences of incorrect interpretation and decision making.

Finite element analyses for the ET TPS have different forms. Thermal analyses have been performed which provide the temperature distributions used for thermal stress computations. Quasi-static stress analyses have been performed for mechanical and thermal loading cases, and some analyses include both linear and nonlinear material behavior. Specialized finite element analyses have also been developed to study the divoting failure mode from quasi-static and transient-dynamics perspectives. Each of these basic analysis efforts is discussed next.

Thermal finite element analyses. Thermal analyses refer to heat transfer analyses of the structural system including conduction, aerodynamic heating and convection, cryogenic temperature, radiation, and other thermal effects. Historically, computational heat-transfer models have used lumped-capacitance finite difference formulations to solve the heat-transfer equations (*e.g.,* SINDA[*]). However, most commercial finite element codes offer heat-transfer analysis options. Effects such as conduction, convection and radiative heat transfer are included in a transient thermal response prediction. Typical results include the temperature distribution through the structure for use in a thermal stress analysis and to define the thermal loading or to define the material properties at a given

---

[*] http://www.sinda.com

temperature. The thermal stress prediction requires the mapping of a spatial, and perhaps temporal, distribution of temperatures from the discrete points in the heat transfer model to the discrete points in the stress analysis model. Spatial discretizations for a stress analysis that include through-the-thickness effects (*i.e.,* three-dimensional analysis) are perhaps also sufficient for a heat conduction analysis and provide easy access to nodal temperature values. However, when convection and radiation effects cannot be ignored, the two computational models may differ significantly. The thermal stress simulations may also be time independent or time dependent. Blosser [17] discusses the influence of thermal-structural boundary conditions from a testing perspective with a view towards modeling and analysis issues to be addressed by the analyst. The reliability and robustness of the thermal stress prediction depends on the heat transfer simulation for establishing accurate and correct temperature distributions, on the mapping or interpolation of nodal temperatures between the heat transfer model and the thermal stress model, on temperature-dependent material modeling, and on thermal-structural boundary condition specification.

Quasi-static finite element analyses. Quasi-static finite element analysis refers to a static stress analysis wherein the loads are assumed to be time-independent (*i.e.,* "snapshot" of the flight load conditions) and/or inertial effects are included as effective static loads. Often, only mechanical loading is given; however, combined thermal and mechanical loadings are frequently required and become of increasing importance as risk mitigation requirements and vehicle life-extension requirements are defined.

The analyst must make many decisions in the modeling process that reach beyond spatial discretization of the geometric definition of the component. Boundary conditions including structural boundary conditions, external loading, thermal constraints, and evolving boundary definitions due to contact must be addressed and defined for the mathematical model. Insight into how stress and strain gradients attenuate around local structural features is typically required (St. Venant effects). Material modeling is another key decision for the analyst in terms of available material models in the finite element code, availability of material data to support the material model, and the appropriateness of the material model and its anticipated response. For example, an analyst may elect to model a laminated quasi-isotropic composite structure as a linear elastic isotropic material (*i.e.,* the "black aluminum" approach) rather than defining a laminated composite based on classical lamination theory. Another example could be posed by modeling a material with different elastic moduli in tension and compression (*i.e.,* a bimodulus material) as a material with the same modulus in tension and compression.

Global finite element models, such as the ET loads model shown in figure 10, are common. Detailed local finite element models, such as the ET bipod region shown in figure 11, are also becoming commonplace to examine local design details. Finite element models of the superlightweight external tank also exploited this variable refinement approach [18-22] in order to capture local response detail. Global analysis of large shell structures pose further modeling challenges to capture a short-wavelength buckling response in local regions and local generalized imperfections resulting from fabrication (*e.g.,* welding, bolted joints) as described by Nemeth and Starnes [23] and

Starnes, Hilburger and Nemeth [24]. The necessity to anticipate such local phenomena places increased responsibility on the analyst that is not embodied by knowing how to use the modeling and analysis tools themselves. The analyst must possess the knowledge and the understanding of the underlying mechanics of the system being designed. Unfortunately, knowing *how to use* an engineering analysis tool is too often perceived as knowing the mechanics.

Large-scale finite element models are often developed for component design verification. Additional fidelity is often needed for design certification because of the influence of local design details, stiffness changes, attachment points, and load introduction details. Global-local modeling strategies strive to extract from global analysis models boundary conditions for more refined local analysis models and can be performed in a cascading manner involving multiple modeling levels. To capture the global structure behavior in the local model, boundary conditions along the edges of a local region are extracted from this global model and imposed on the local detailed model. Careful attention to this process is needed to insure compatibility of primary field variables (displacements and rotations) as well as continuity of secondary field variables (strains and stresses). Again, insight into how stress and strain gradients attenuate around local structural features must be known, especially when anisotropic materials are involved, to ensure proper modeling. Nodal compatibility of the primary field variables does not automatically insure gradient continuity across the global-local interface, as it exists in the component. Nevertheless, global-local modeling is an excellent tool for examining local details.

The engineering community at large commonly uses two global-local modeling approaches: embedded global-local modeling and independent global-local modeling. Embedded global-local modeling is defined as embedding a more refined local analysis model within a less refined global analysis model, as depicted in figure 12 for the bipod region. Exploiting the two-dimensional (2D) shell finite element model of the ET component and embedding a detailed three-dimensional (3D) finite element of the bipod region allows the direct integration of global response characteristics with the local detailed modeling. Independent global-local modeling is defined through the explicit assumptions on edge or boundary conditions for the local model, based on extracted global model results. Verifying the global-local modeling process is a necessary step in the analysis and should consider both primary (*e.g.,* displacements) and secondary variables (*e.g.,* stresses). For approaches that combine 2D and 3D finite element modeling, the analyst needs to insure that their coupling does not introduce implicit constraints as a result of the kinematic assumptions for the different structural idealizations (*e.g.,* plate elements assume the out-of-plane deflection is independent of the through-the-thickness coordinate while solid elements do not).

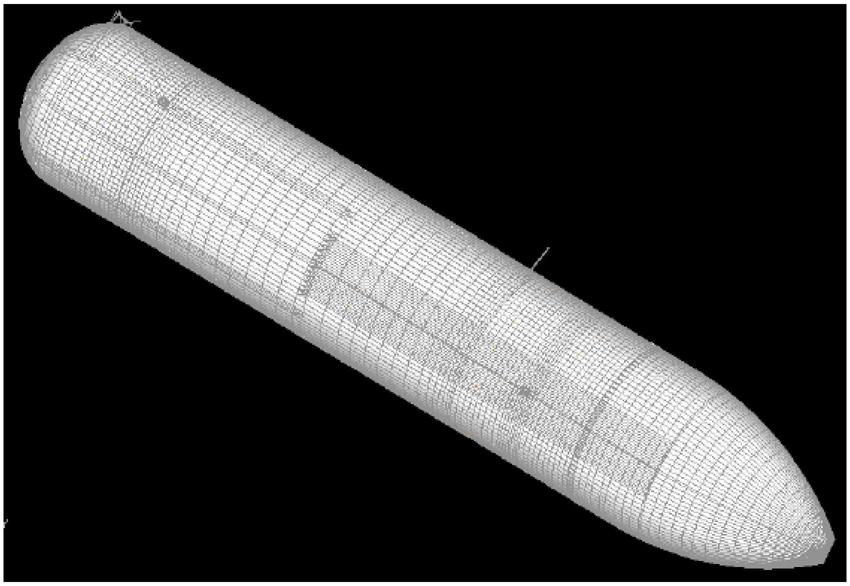


Figure 10. Example of global shell finite element model of the ET.[*]

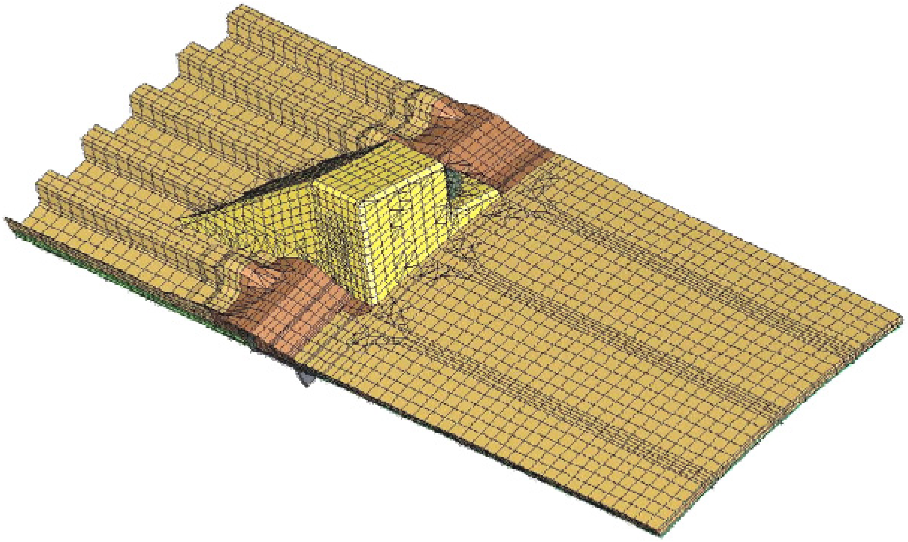
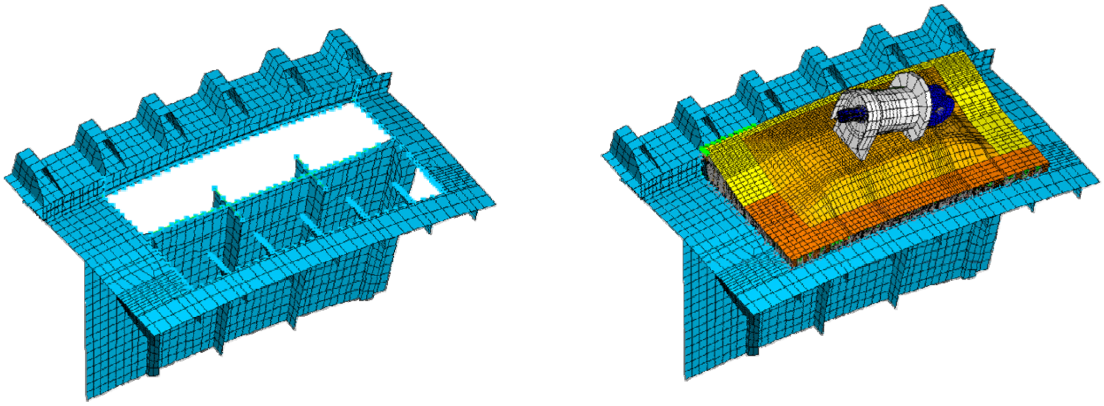


Figure 11. Independent local 3D finite element model of the STS 107 ET bipod region.[**]

---

[*] From presentation by Jeff Pilet, NASA MAF, "STS-107 Bipod FEM Analysis," ET RTF TPS Analysis Summit, NASA Michoud Assembly Facility, November 20-21, 2003.



(a) Local detailed model removed. (b) Local detailed model embedded.

Figure 12. Example of an embedded detailed local model using the global shell finite element model.[*]

Specialized finite element analyses. Specialized finite element analyses refer to specific event-related analyses (*e.g.,* divot simulation), typically requiring only local geometry definition. This approach also includes analyses performed to simulate coupon tests and material characterization studies. Specialized models have their purpose in exploratory studies to understand basic response characteristics and to develop "engineering" models of structural and/or material behavior complexity. A fundamental risk of these specialized analyses is the tendency to over extrapolate the basic results to actual components without full accounting of the inherent analysis assumptions. Specialized analysis models are very useful and needed; however, their range of validity also needs to be understood and clearly defined. A few examples for the ET TPS are described to illustrate these analyses.

The first example is the analysis performed to support the evaluation of the TPSMOM program.[†] Refined plane strain and 3D solid finite element analyses were performed to calibrate the TPSMOM results and to account for certain anomalies not addressed in the TPSMOM program. These finite element analyses have the potential to provide understanding of the influence of substrate curvature, substrate pre-strain, and localized imperfections (*e.g.,* voids, debonds, material variations) on the foam response.

The second example is the analysis performed to understand the divot formation mechanism using an axisymmetric solid-of-revolution finite element model.[‡] Using the

---

[*] From presentation by Eric Poole, NASA MSFC, "TPS Closeout and Coupon Specimen Analysis," ET RTF TPS Analysis Summit, NASA Michoud Assembly Facility, November 20-21, 2003.

[†] Presentation by Stan Oliver, NASA MSFC, "Analysis of TPS Verification Test Configurations," ET RTF TPS Analysis Summit, NASA Michoud Assembly Facility, November 20-21, 2003.

[‡] Presentation by Steve Scotti and Lynn Bowman, NASA Langley, "A Review of Foam Fracture Analyses," ET RTF TPS Analysis Summit, NASA Michoud Assembly Facility, November 20-21, 2003.

MSC.MARC[*] nonlinear finite element code, an unstructured mesh of triangular elements was used to model the divot simulation testing of the foam (see figure 13). A nonlinear "concrete-like" material model was used because of its similarity to the material behavior of the foam. A strain-cutoff value was specified that determines when an element has failed completely and no longer contributes to the physics of the simulation. Material modeling and failure detection are key assumptions in the analysis. Simulation results indicated that a local material failure initiated at one corner of the configuration, and then propagated along an initial 45-degree path to the outer free surface, as shown in figure 13. Implicit in the analysis are the material response in terms of the concrete model and the structural response in terms of axisymmetric behavior. Taking these exploratory results and extrapolating to test conditions or flight conditions should be discouraged. Moreover, parametric studies from such models are instructive and can be used to guide analysts as they eliminate or verify various analysis assumptions.

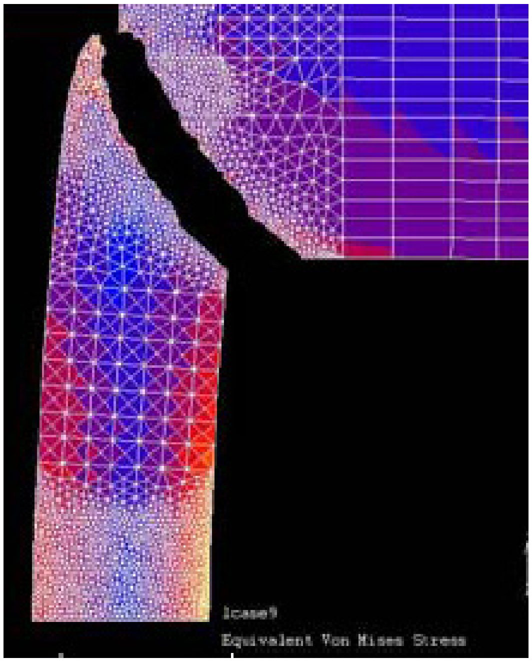


Figure 13. Failure simulation for the foam divot fracture tests.[*]

The third example of a specialized finite element analysis is the analysis that was performed to examine the transient dynamic behavior of the foam in the divoting process.[†] LS-DYNA[‡] simulations were performed for various test configurations to correlate finite element predictions with experimental results. Different finite element modeling approaches (full model versus doubly symmetric quarter model) and different material models were considered. Simulation results indicated significant mesh

---

[*] http://www.mscsoftware.com

[†] Presentation by Andy Brown and Steve Medley, "Foam Void/Divot Modeling using LS-DYNA," ET RTF TPS Analysis Summit, NASA Michoud Assembly Facility, November 20-21, 2003.

[‡] http://www.lstc.com

sensitivity to failure-stress predictions and a dependency on the material modeling was evident. Again, parametric studies using such models are instructive and can be used to guide analysts as they eliminate or verify various analysis assumptions. However, the tendency is to extrapolate the results interpretation too far.

The fourth example is the analysis performed to simulate basic material characterization tests. Finite element modeling and analysis of mechanical testing of coupon-level specimens and other subcomponents can provide significant insight for subsequent modeling and analysis and test-specimen design. The simplicity of coupon-level specimen geometry and response may contribute to complacency about such tasks. Again, material modeling and failure detection are key assumptions in the analysis. These analyses provide basic response prediction assessments for uniaxial and biaxial stress states; however, the various assumptions (*e.g.,* geometry symmetry, material modeling) and their influence on the anticipated model response (*i.e.,* too much intuition imposed on the models that then forces a specific response) must be understood.

ET TPS Constitutive Modeling Review

Foam constitutive modeling continues to be a subject of research for a wide range of applications. Most of the research effort is related to the mechanical compressive behavior of foams in support of impact-energy management system designs for improved crashworthiness, passive re-entry landing systems, and lightweight hybrid composite structural systems (*e.g.,* see Refs. [25-29] for selected foam material models in LS-DYNA[*]). A typical room-temperature tensile response exhibits an initial linear elastic response followed by a nonlinear plastic response that exhibits hysteresis. A typical compressive stress-strain response for closed-cell foam is shown in figure 14.

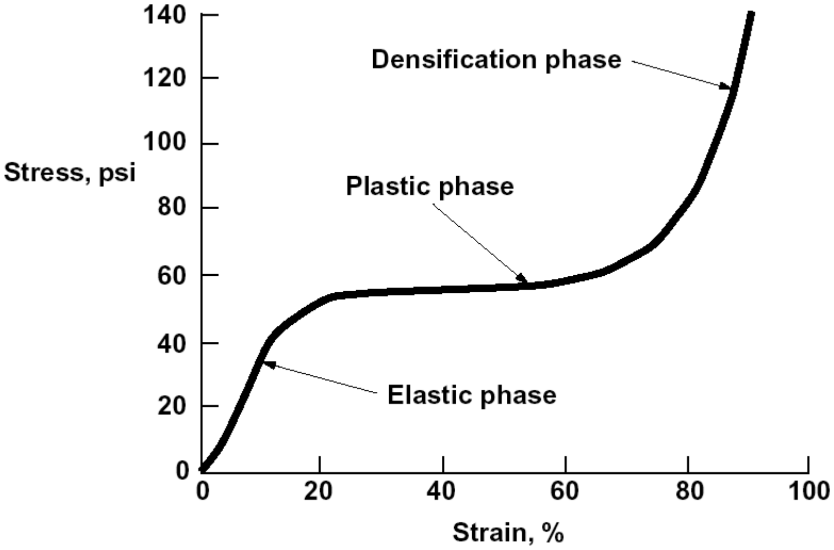


Figure 14. Typical uniaxial compressive stress-strain curve for rigid-cell foam.

---

[*] http://www.lstc.com

Three major assumptions are generally involved in modeling the uniaxial constitutive response of structural foams. First, the foam is assumed to respond elastically. While this assumption is true for very lightly loaded structures, the assumption is easily violated. Second, the foam is assumed to respond linearly. Again this assumption is true for only a very small segment of the stress-strain response curve. Third, the foam is assumed to respond similarly in tension and compression. The tensile response of a foam material is typically quite limited due to local failures of the cellular structure that, once failed, has limited ability to carry additional load. However, in compression, the foam, as shown in figure 14, compresses, plastically deforms with large permanent volume changes and then consolidates or densifies as the loading increases (*i.e.,* forcing the entrapped gasses out and collapsing the cell walls) until the stiffness approaches that of a solid. Typically the compressive stress-strain curve exhibits a small initial linear elastic region (under 10% strain) followed by a large plastic phase (up to 60-70% strain) and then a sharp stiffening or densification phase (up to 100% strain) as all the cells are collapsed. The plastic phase of the foam stress-strain response is often engineered to dissipate or "absorb" energy during an impact event. These foams also exhibit a hysteresis response when unloaded.

If the foam material is subjected to uniaxial loading, these models can characterize the stress-strain response. However, issues arise related to multi-axial stress loading, combined stress states, and bending behavior that may be cyclic. A tensile load could potentially induce local tensile failures on one surface of a plate-like specimen, while compressive failures occur on the opposite surface, thereby creating a non-symmetric material response locally about its midplane. Such complexities are typically not encountered in the analysis of crash-energy management systems.

Because foam materials are more often used to dissipate impact energy, foam constitutive models are incorporated into explicit transient-dynamic finite element codes long before they are implemented for general structural-response simulations. Constitutive models of these foams are generally well characterized for compression loading. The foam material structure is usually uniform and repeatable. These types of foams can be described as single function materials; that is, they are for impact energy dissipation. As a result, the response to loading rates and "crush" behaviors are of prime importance. The stress-strain response is also dependent on the foam density. Constitutive models for the compressive behavior of polyurethane foams have been developed (*e.g.,* Refs. 25-37) and implemented into explicit transient-dynamics codes. These constitutive models typically employ some form of a plasticity model within a continuum damage formulation that is suitable for large-strain applications.

Applications of foam materials in the aerospace industry include core material in sandwich structures, cryogenic insulation, part of a crash-energy management system for general aviation aircraft, and part of a passive re-entry landing system for planetary exploration or planetary sample return. Weiser *et al.* [38] describes potential needs for polyimide foams and the impact of their chemistry and fabrication process on their mechanical and thermal properties. In the past, only the thermal insulation properties of the ET TPS were of concern; however, understanding the structural response characteristics of these foams has become essential as engineers are asked to assess extreme loading environments and attempt to quantify operational risk associated with debris shedding. In most cases, structural ground tests are not fully representative of flight conditions and ET TPS installation process variability results in significant scatter in component test results. These aspects push the state of the art in computational material modeling and experimental testing for verification of material data and component design.

The constitutive model selected for a foam material in a computational simulation has a significant effect on the validity of the analysis prediction. It is common for an analyst, having only limited material data (*e.g.,* secant modulus and ultimate strain), to assume a linear-elastic-to-failure response for tension and compression. Such models are readily available in all finite element codes, require minimal material data as input, and are amendable to linear stress analysis methods. However, these models may be inadequate for failure-mode predictions and accurate representation of the structural response; however, often these are the only applicable models available *for the given data*. If stress-strain data are provided, the analyst can define piecewise linear or curve-fit material data to the analysis code and utilize an elasto-plastic material model. The limitation is the assumption of equal response in tension and compression and potentially the lack of hysteretic behavior in the analysis model. In addition, the strain level of the foam response has an overarching influence on the analysis model (*i.e.*, the response may involve large strains).

Neilsen, Krieg, and Schreyer [31] developed a nonlinear constitutive model for polyurethane foam by decomposing the foam response into three regimes (similar to those identified on figure 14): an initial elastic regime, a plateau regime, and a "lock-up" regime. They assumed that the foam response could be decomposed into the response of the cell-wall skeleton and the response of a diffuse continuum of air and polyurethane particles that does not resist any shear deformation. The diffuse continuum model assumes only volumetric changes. The stress state in the foam was assumed to be the sum of a skeleton stress from the cell walls that accounts for the elastic and plastic phases of the foam, and of a nonlinear elastic continuum stress that accounts for lock-up of the foam caused by internal gas pressure and cell-wall interactions (diffuse continuum contribution). The foam stress tensor can then be written as:

$$\sigma_{ij}^{foam} = \sigma_{ij}^{skeleton} + p\delta_{ij}$$

where are σ_{ij}^{foam} and $\sigma_{ij}^{skeleton}$ the stress tensors for the foam and the cell-wall skeleton, respectively, p is the diffuse continuum pressure and is δ_{ij} the Kronecker delta tensor.

Neilsen *et al.* [31] derived an expression for the diffuse continuum pressure when air is allowed to escape. In this case, they found that diffuse continuum pressure does not contribute to the foam response until the engineering volumetric strain is larger than the original volume fraction of air in the undeformed foam. To complete the model, they developed relationships for the skeleton stress by using an additive decomposition approach for the elastic, plastic and damage responses.

Theocaris [37] proposed an elliptic paraboloid failure surface for closed-cell polyurethane foams. The failure mechanisms are associated with the cellular structure. The bending behavior of the cell walls is related to the elastic response, the buckling behavior is associated with the elastic collapse of the cell walls, and the plastic-hinge formation at cell-wall junctures describes the plastic collapse. The availability of this foam constitutive model in finite element codes is unknown.

Other researchers have followed a similar approach that incorporates pore pressure in the constitutive model. Similarities are also noted with soil constitutive models from geomechanics [39, 40].

In nearly all cases examined, the given foam material is assumed to have a regular structure and its fabrication process is assumed to result in consistent, repeatable material characterization. The SOFI used in the ET TPS exhibits neither attribute. Mechanical properties of the SOFI tend to exhibit a strong dependency on the installation process and do not exhibit any consistent pattern. Analysis needs include having adequate material data (stress-strain data, failure strains, failure modes) that can be used, with caution, in existing finite element codes for mechanical and thermal response predictions and validation of finite element models for appropriate stress state and material characterization. In the long term, the development of constitutive models with different behaviors in tension and compression is needed for many of the finite element codes in use today. Also, constitutive models specifically developed for foam materials are needed for use in multi-functional applications (i.e., account for thermal and strain-rate effects as well as structural response including damage and failure modeling). Finally, sufficient material characterization studies need to be performed to establish the statistical basis of the material properties and the sensitivity of these properties to variations. At present, the SOFI material is not adequately characterized for use in analysis-based verification and certification efforts.

In terms of a path forward, new SOFI constitutive models should be developed to account for voids, pore pressure, anisotropy, temperature dependence, and strain-rate sensitivity. Such constitutive models could then be implemented and refined in an expeditious manner by researchers as user-defined material models within some of the commercial finite element codes (*e.g.,* UMAT routines with ABAQUS). This approach allows researchers to focus on the material response modeling rather than the entire finite element software system and leverages the existing finite element technology and infrastructure. This approach is uncoupled from the commercial software vendors' business-driven priorities and permits material model development in concert with material characterization testing. Then, once the material model is matured and verified, the commercial software vendor could be approached to integrate the new material model into their code as a fully functional option, including appropriate documentation and testing.

Trends and Future Directions

Structural design and analysis depend heavily on finite element analysis models and their predictive capabilities. Finite element structural analyses have been used successfully to design many engineering systems wherein the “correctness” of the results is judged based on test/analysis correlation for a series of static load cases and/or correlation with modal survey tests. However, the robustness and reliability of that finite element model when subjected to general loading conditions (other than those of the test) are neither established nor quantifiable. Formal procedures for estimating structural modeling uncertainties have been used in structural dynamics through model correlation and model updating techniques. Even in these cases, model uncertainty factors (MUFs) are often very large (*e.g.,* heuristic MUFs on the order of 10, 20, or higher are common) until test-analysis correlation has been achieved. As confidence in the analysis model increases through test-analysis correlation, the value of the corresponding MUF is reduced.

Structural modeling and analysis of aerospace systems and subsystems pose challenges to engineers working to provide accurate and robust computational structural mechanics simulations for use in design, verification, and certification. Analytical and computational models of structural systems are often based on rule-of-thumb procedures and/or user familiarity with a particular structural analysis tool. Tasks associated with geometry modeling based on computer-aided design (CAD) tools and spatial discretization based on finite element technology are frequently performed by those trained on a specific software tool. While being well trained on how to use the modeling tool is necessary, exposure to actual design hardware or similar design projects and perhaps even additional coursework in structural mechanics principles and computational procedures is desirable. Prior experience with a related aerospace system and knowledge of its anticipated response would contribute significantly to reducing the mathematical modeling cycle time. Analysis problems may be averted or design problems detected through previous experience on related designs. However, as analysts become experienced, they are frequently moved away from the hands-on structural modeling and analysis tasks to serve as supervisors or mid-level managers. Developing modeling and analysis expertise generally is not supported as a long-term corporate career path within the current engineering culture.

Capturing and preserving modeling and analysis "best-practices" associated with a given design system and with a given set of engineering tools is often overshadowed by the design schedule and resource limitations. As a result, such knowledge capture of experienced analysts has not been very successful, and corporate memory of relevant structural modeling and analysis procedures is frequently lost when they leave the organization (*i.e.,* take another job, retire, or die). Future rapid modeling and analysis system should address structural modeling and analysis, reliability and robustness, and computational infrastructure and include features for knowledge capture and automated electronic search of internet-accessible resources (*e.g.,* see Ref. 41). Developing such a modeling and analysis framework would require a considerable investment over an extended period of time.

In any large-scale structural modeling and analysis effort, a common question asked is "How much confidence do we have in the results?" Correlation between test and analysis is one indicator for a specific test configuration but does not validate the structural model and analysis approach in general. Test-analysis correlation certainly is a necessary condition, but not necessarily a sufficient condition to accept the mathematical model as correct and accurate for other design and off-design configurations. A simple example is a thin material coupon specimen loaded by in-plane tension and analyzed using plane-stress analysis. Correlation between test and analysis can be remarkable; however, if the loading direction is reversed (compression), the plane-stress model alone is inadequate to verify the design because structural stability aspects of the new configuration are ignored in the analysis model of the original problem.

The concept of validation and verification of modeling and analysis tools is not new. The computational fluid dynamics or CFD community has well-established guidelines for qualifying their numerical predictions [42-44], while the structures community is, for the most part, lagging behind. Some application areas have established sets of checks and balances for finite element analysis efforts. One industry may require a complete and thorough independent re-analysis of the entire system, which provides a measure of assurance and quality, but for a price. Often such a practice is employed for critical mission, one-of-kind systems that once deployed are without any possibly of adjustment or repair. Another industry may employ a checklist approach to finite element analysis, which causes the analyst to perform a self-assessment in a systematic manner. In 1995, the Ship Structures Committee published such a process that included a 30-page analysis assessment [45]. A formal verification and validation process for the modeling and analysis of aerospace systems is still needed.

Model verification is defined in Ref. 44 as the process of determining that a model implementation accurately represents the developer's conceptual description of the model and the solution to the model. *Model validation* is defined in Ref. 44 as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model. Developers of structural analysis tools often define tool validation as providing the right equations and correctly solving the numerical problem, while tool verification is demonstrated through the solution of a suite of test cases that exercise various, but not all, parts of the software.

Analysts, in contrast, generally understand model verification as applying the appropriate tools to predict the anticipated physical behavior and correctly solving the resulting equations in the mathematical model. Model validation implies that a verified mathematical model "morphs" into a validated model through model tuning and model updating so that accurate, test-correlated predictions are obtained and the range of applicability of the model is well defined. A validated model is used to establish confidence in the predictions for a particular application and hence used for risk mitigation. The structural dynamics community is perhaps further along than the structural mechanics community in the model validation and verification process because of their efforts to correlate finite element model results against modal test data by using model updating or model tuning procedures (*e.g.,* see Refs. 46-51). Within the structures community, this process has not been formalized except through the use of rules of thumb and repeated solutions of similar problems (*e.g.,* see Refs. 52, 53). Recent efforts by the American Society of Mechanical Engineers (ASME) include the formation of a new standards committee (PTC 60) on verification and validation of computational solid mechanics. Their charter is to provide procedures for assessing the correctness and credibility of modeling and simulation in computational solid mechanics. However, the role of nonlinear structural behavior is increasing, along with the integration of new material systems for innovative aerospace vehicle design, leading to a potential increase in the uncertainty of results obtained from structural mechanics modeling and analysis tools. These changes will provide further impetus to the structures community to develop formal procedures for the verification and validation of structural modeling and analysis.

Oberkampf [54] and his staff at Sandia National Laboratories have studied this issue of validation and verification of a computational mechanics model and have offered short courses on this topic.[*] Two classes of uncertainty for the computational simulation process are defined. *Aleatory uncertainties* are those inherent variations associated with the physical system or environment (*e.g.,* damping, joint stiffness, material properties). *Epistemic uncertainties* are potential deficiencies in any phase of the modeling and analysis process that are due to lack of knowledge (*e.g.,* poor understanding of system response, insufficient test data). In addition to these types of uncertainties, Oberkampf [54] defines error as a recognizable deficiency in any phase of the modeling and analysis process that is not due to lack of knowledge. Such errors can be acknowledged errors (*e.g.,* idealization and discretization) or unacknowledged errors (*e.g.,* blunders and mistakes). The role of nondeterministic analysis on verification and validation of structural mechanics models has been discussed by Thacker [55], including some of the foundation concepts that will be needed.

Another aspect that contributes as much to model verification as it does to model rejection is the application of available constitutive models to represent new material systems or new fabrication forms. Examples include modeling of textile composites using lamination theory, modeling structures with embedded sensors, modeling of progressive damage response using ply discounting for material degradation, and modeling of shape-memory alloys. These aspects lead to aleatory uncertainties due to unknown variability associated with temperature, moisture, and strain rate, to epistemic uncertainties due to limited material characterization, to acknowledged errors resulting from inappropriate use of a constitutive model and the lack of a statistical basis for the material data, and to unacknowledged errors resulting from limited knowledge and understanding of new material systems. Given these factors, test/analysis correlation is still often achieved within a research setting and extension of these modeling practices to the general design process contributes to the overall risk. A process to quantify risk associated with structural modeling and analysis uncertainties would potentially identify critical research areas for future structural analysis tools and identify technology readiness levels (TRLs).

---

[*] "Verification & Validation for Computational Simulation" a 2-day short course offered at NASA Langley.

Probabilistic risk assessment (PRA) tools are used frequently in systems engineering studies to quantify the known risk to achieve a certain mission objective and to identify critical areas (*i.e., tall poles*) that can be addressed to mitigate overall risk to the mission. PRA models can incorporate uncertainities associated with statistical variations of the design parameters, off-nominal conditions, accumulation of tolerances, and local failures, and also the severity of the consequences can be quantified. PRA models are commonly developed using event sequence diagrams that have an associated probability of occurrence. Examples of PRA models are given in the NASA workshop proceedings on nondeterministic methods [56]. A PRA model is typically based on a series of event sequences, their various outcomes and their probabilities, and supported by deterministic analyses, nondeterministic analyses, expert opinion, or a combination of these factors. However, PRA approaches have been applied only to engineering systems with mission requirements (*e.g.,* Fragola [57] and Knight, Glaessgen, and Sleight [58]). Reducing modeling and analysis uncertainty and errors through the use of a PRA approach appears to be possible by considering the structural modeling and analysis process as a sequence of events with different possible outcomes. Examples include modeling decisions required for finite element type, level of refinement, and degree of idealization (*i.e.,* uncertainy in the spatial modeling fidelity as well as in the structural idealization assumptions). Computational results obtained using various fidelity models would then be developed and compared with results obtained from a high-fidelity model in order to quantify the modeling uncertainty. The difference between the solutions provides an estimate of the uncertainty associated with a given modeling decision, for a given structural design. As an example, a blade-stiffened panel subjected to uniaxial compression and modeled using a smeared-stiffener approach may agree exactly with a branched-plate model (shell elements used to model both the panel skin and the blade stiffeners) for a panel with a thick skin because it exhibits an overall global buckling behavior. However, the smeared-stiffener approach may lead to significant errors for a panel with a thin skin that exhibits a local buckling behavior. Once the various factors that contribute to structural modeling and analysis uncertainties are identified, a PRA approach has the potential of identifying the modeling features and analysis factors that significantly affect the prediction accuracy.

Accurate and robust finite element models are needed to provide high-fidelity analysis results. Issues of structural and material idealization, geometry representation and spatial discretization, and the overall solution process continue to arise partly due to the terminology and nomenclature. Thus, the terminology associated with finite element modeling and analysis as well as a delineation of options and approaches is described in Appendix A and references 59-63 provide background information on the finite element method. Aerospace system complexity continues to increase due to higher performance requirements, extended life-cycle requirements, limited on-ground testing ability, and newer material systems. Existing computer system technology has enabled significant breakthroughs in analysis-based designs by using realistic visual models and immersive simulation environments. These breakthroughs are accompanied by the challenge to stay focused on the engineering analysis results and the underlying engineering mechanics. Risk-based design methods are evolving wherein statistical variations associated with material properties, applied loads, environment effects, and geometrical and assembly tolerances are treated in a probabilistic manner.

Definition of the design problem within a probabilistic setting often requires a new paradigm for defining performance metrics. These metrics may be approximated in some cases through a *metamodeling* approach using response surfaces, but such an approach is limited. Input into PRA models for system performance is frequently structural modeling and analysis results (deterministic and/or nondeterministic) without a quantifiable metric on the accuracy and appropriateness of the computed results. That is, the results could be correct for a given model but the model is incorrect (*e.g.,* symmetric model used instead of full model), the results could be correct for the time allotted for the simulation (*e.g.,* only a few days allocated for solving a specific problem), or the results could be accurate with minimal uncertainty (*e.g.,* model predicts test results and is consistent with physics). However, accepted, formal procedures to assess and quantify structural modeling and analysis uncertainties and errors have not been developed.

Recent concepts such as the Grid Convergence Index [42, 43] for CFD analyses are being extended to finite element structural-dynamics analyses [64]. Such concepts are being taught through short courses by Los Alamos Dynamics.[*] Having such a procedure would contribute to the reduction of model uncertainty factors (lower MUFs), increasing the confidence level for the structural model and analysis, and increasing safety for the mission.

Acceptance Criteria for Modeling and Analysis Procedures (MAP)

Finite element analyses of structures are used in several different ways. Each analysis may have a different objective that, as a result, places different requirements on the fidelity of the mathematical modeling. For example, analyses that are used in the preliminary design phase of product development are required only to capture the bulk or global behavior of a structure and are relatively unsophisticated. Thus, the requirements of the finite element modeling procedure are the least stringent. In contrast, analyses that are used in the detailed design phase of product development, and that deal with margins of safety or analyses that are used as part of a structural certification program, are required to predict stresses and strains accurately near local discontinuities or other stress risers, in addition to the global behavior. This class of analyses must yield highly accurate results and, as a result, the finite element modeling procedures are very stringent. Moreover, as analysis methods continue to mature and are used to reduce the dependence on empiricism in the certification processes for aerospace systems, a means of quantifying the pedigree of the finite element results will be needed in the decision-making process. Some in the engineering community have suggested that this problem be best avoided by requiring engineers that conduct finite element analyses to be licensed or registered. However, this approach may not be sufficient because a licensed user may only know how to run the code, push the buttons, and generate color plots without an understanding of the structure, its function, or its response.

---

[*] "Finite Element Model Validation, Updating, and Uncertainty Quantification", a 2-day short course held at Stanford University, Stanford, CA, September 18-19, 2003.

A different strategy is taken herein, one that is not intended to over burden either the engineers or the engineering infrastructure. This strategy is concerned with quantifying the pedigree of the finite element model and its results in a way that follows standard practices in other areas (*i.e.,* development of acceptance criteria). In particular, the goals of this effort are to identify key modeling issues that should be considered in developing acceptance criteria and to bring these issues to the attention of those that are often asked to accept computed results without any measure of their pedigree. The acceptance criteria are embodied in a modeling and analysis-planning document that parallels the formal test plan documents commonly used in testing. Test plan documents are used to ensure best practices are used, to ensure that test objectives and requirements are known, defined, understood and met, to mitigate uncertainties in the test procedure and process, to maximize the benefit of the test, and to provide robust results that will need to stand the test of time.

Every analyst, or team of analysts, develops modeling and analysis plans informally. Embedded within this informal process are the unique characteristics of individual analysts. In many cases, the engineering analyst may be the worst critic of the analysis model and the results it produces. Formalizing the modeling and analysis plan (MAP) establishes a process that provides a modeling and analysis product with a pedigree. It is intended as a set of guidelines to be used to assess a specific analysis effort rather than to constrain or impose a specific modeling and analysis process. At first blush, this process may seem to be an added burden and a delay in generating needed results; however, in the long term it should prevent incorrect results from being forwarded to review boards that act on (or react to) them. Additional benefits are the mentoring of less experienced analysts and the capture of corporate memory related to mathematical modeling and analysis of specific engineering systems.

The modeling and analysis plan considered in the present study defines best-practices guidelines that contribute to an acceptance criteria for computed results in the form of closed-form solutions, finite element solutions, or other analytical solutions. The MAP should address the following major areas:

- Analysis objectives and scope
- Terminology and nomenclature
- Sources of error and uncertainty
- Modeling approach and rationale
- Mathematical model verification
- Mathematical model validation
- System response and sensitivities
- Configuration management
- Documentation and approvals

Each area is defined in subsequent sections and specific examples of topics to be addressed are given. The collective response in each area forms the MAP.

Analysis Objectives and Scope

Defining the objectives and scope of the analysis within the MAP are necessary ingredients of acceptance criteria for analytical studies. The objectives of the modeling and analysis effort, both from a short-term and a long-term perspective, need to be stated clearly and the limitations or restrictions noted. As an example, an analysis that is used as part of a certification process will most likely out live the analyst. Accident or failure investigations often discover that the "certified" analysis model is decades old. The objective of the analysis should be defined in terms of the intended purpose. For example, a basic loads model to establish load paths, a detailed stress analysis model to examine local gradients and failures, or a structural dynamics model for predicting mode shapes and frequencies. The focus of the modeling and analysis should be centered on a predictive capability for structural response characteristics. Drivers of the structural response should be clearly identified, as they are understood at that time. Some drivers are easily recognized (*e.g.,* applied loads or operating environment), while others are not as obvious (*e.g.,* friction in joint connections, structural-thermal boundary conditions, contact between mating parts, damping, or inertial loading). In addition, the scope of the modeling and analysis activity should be stated to provide guidelines for the extent to which the models and results are applicable. Using a loads-developed finite element model for detailed stress analysis is clearly an example of exceeding the scope of the original model. For example:

- Objectives – To model and analyze the structural response of the ET TPS acreage SOFI in order to assess the risk associated with SOFI disbond from ET substrate. To determine the local stress state from the substrate through the foam, including transverse stresses associated with peeling and shearing.

- Scope – Analyses will be performed based on room-temperature, linear-elastic material behavior using time-independent, uniformly distributed loading. Solutions for the longitudinal and circumferential directions will be developed.

Terminology and Nomenclature

Common (standard) terminology and nomenclature should be defined and formally imposed. Often it becomes clear that the real issues are related to semantics or the use of jargon. Clarity in terminology becomes even more important as analysis models are distributed for multidisciplinary use or by different organizations or companies. One example is the word "model". It one setting it may refer to the geometric definition of surface boundaries that form a solid geometry model, while in another setting, it may refer to a finite element model. Hence, ambiguity arises when it is stated, "the model is available".

In addition, each engineering analysis tool typically has its own nomenclature or "keywords" that describe certain input definitions or analysis directives. The intent here is not to duplicate a user's manual but rather to provide sufficient information that the meaning of keyword or command is accurately conveyed without requiring the reader to resort to a user's manual.

Sources of Uncertainty and Assumptions

Sources of uncertainty and assumptions are also identified in the MAP and contribute to understanding the limitations and risk associated with the structural modeling and analysis results. Common sources of uncertainty include geometry definition, material constitutive forms, homogeneity of material, consistency in material process, statistical basis of material properties, material state dependencies (*e.g.,* temperature dependency or strain-rate dependency), distribution and magnitude of mechanical and/or thermal loading, and sensitivities to environmental conditions (*i.e.,* temperature, humidity, barometric pressure, exposure, and aging).

Assumptions are made for various reasons. Many of these reasons are justifiable, many are necessary, but all need to be understood. The need to proceed to collect preliminary data and basic characterization is often a key reason to make assumptions. Assumptions are not always detrimental; however, not having a clear definition of what they are is often detrimental. Jones [65] has noted the tendency of many engineers to oversimplify a problem so a convenient simplistic mathematical model could be used even when sufficient fundamental (experimental) information or physical evidence is lacking. He advocates the use of the term "presumption" (something based on evidence that can be examined and evaluated) rather than the term "assumption" (something taken for granted or supposed to be a fact, arbitrary). In mechanics, most so-called assumptions are generally presumptions that are based on probable evidence in their favor and the lack of proof to the contrary. Having sufficient experimental observations and data are crucial to making the correct presumptions in developing accurate and useful mathematical models. Jones [65] concludes with this message: "*Seek and rely on experimental evidence to motivate, develop, verify, and extend your theories.*" This message embodies the high-fidelity modeling philosophy needed for analyzing any engineering system.

Modeling a structure using a representative geometry definition may provide basic insight to structural behavior but lacks sufficient specificity to be used to certify a component. Geometric definitions based on computer-aided engineering and design tools (*i.e.,* CATIA[*], ProEngineer[†]) provide direct links to electronic drawings and fabrication specifications and provide a traceability aspect as well. However, the most critical assumptions appear to occur related to the material data and its robustness (*i.e.,* sufficient statistical sampling to define limits of material properties). That is, are the material data statistically defined? Is a continuum assumption for the material microstructure correct? Is the fabrication process repeatable and consistent? Is the material homogeneous or heterogeneous, isotropic or anisotropic, linear elastic or nonlinear? Are the failure mechanisms and modes characterized and understood? The analyst should anticipate the structural response, including the identification of local discontinuities (*e.g.,* cutouts, re-entrant corners, thickness changes, and stiffness changes). While many assumptions may be necessary at the beginning of the effort, and in the MAP itself, the final modeling and analysis report (*i.e.,* parallel to the final test report) should address these assumptions and their influence on the results. For example:

---

[*] http://www.catia.com

[†] http://www.ptc.com

- Sources of uncertainties – The quality of the SOFI application process is susceptible to significant variability in material chemistry, material cell formation, and the resultant material properties. The generation of voids and pockets should be minimized through process control. However, note that the SOFI is inherently susceptible to defects.

- Assumptions –The SOFI material is assumed to respond as a linear elastic, homogenous, isotropic continuum. Loading is assumed to be uniform over the exterior surface and constant in time, and substrate deformations are ignored. Known failure modes include substrate disbond, divots, “popcorning,” cracks, and delaminations.

Modeling Approach and Rationale

The modeling approach and rationale step of the MAP refers to the engineering analysis models to be brought to bear on the mechanics problem described in the objectives and scope. The modeling approach defines a process for defining the mathematical model required to meet the stated objectives, within the scope of the effort. A description of the mathematical nature and solution of the resulting simulation models, and associated engineering tools to be used, is included in this step as well as a rationale for their selection. A key ingredient of this step is the *structural idealization* of the mechanics problem. Structural members may be idealized as one-dimensional rods or beams, two-dimensional membranes, plates or shells, or three-dimensional solid members. Kinematic approximations associated with different idealizations should be defined and the idealization decisions justified. Idealization of connections and contact need to be made, which vary from penalty-approach contact formulations to nonlinear springs and multi-point constraints.

Similar modeling definitions are required for material response idealization (*i.e.,* linear elastic, nonlinear elastic, elasto-plastic, classical lamination theory, and continuum damage models). Existing stress-strain response data should be exploited as opposed to only modulus data. Constitutive models with state dependencies (*e.g.,* mass-loss-dependent, strain-rate-dependent or temperature-dependent properties) are often required and may impose restrictions on analysis tool selection.

Idealization of external loads refers to assumptions related to their spatial distribution, magnitude, and temporal variation. Often flight loads are idealized as a sequence of “snapshot” load cases derived from flight conditions and a quasi-static analysis is performed rather than a complete transient-response prediction. Thermal loading is often necessary as well and poses different challenges in terms of the mapping of nodal temperatures from the thermal model to the structural model.

Engineering tools would include those based on closed-form solutions, either derived or from handbooks (*e.g.,* Roarke and Young [66] or Blevins [67]), analytical solutions to partial differential equations obtained by using the method of weighted residuals, finite element methods, boundary element methods, or some hybrid solution procedure. Spatial discretization modeling tools (*e.g.,* PATRAN, I-DEAS, and FEMAP) and associated solvers (*e.g.,* NASTRAN, ABAQUS, LS-DYNA) also need to be identified. Use of multiple mathematical models is recommended so that a hierarchical solution process can be established using a building block approach. While such an approach can be time consuming from a short-term perspective, the approach provides robustness, credibility, and engineering understanding in the long term. This long-term perspective must be examined and its potential future significance considered.

Depending on the nature of the engineering tools selected, the mathematical model definition will take different forms. Handbook solutions are very helpful for preliminary assessments but their range of applicability is often limited because of simple geometry and boundary conditions. Analytical solutions are the next level and increase the fidelity of the predictions over handbook solutions in many cases. Identifying the nature of these solutions in terms of their functional form (*i.e.,* trigonometric series, Chebychev polynomials, and Legendre polynomials) and the convergence characteristics of these solutions is needed in the MAP. Tools such as the finite element and boundary element methods require the domain to be spatially discretized. Spatial discretization introduces approximations on two levels: geometry approximations for curved edges and surfaces and response approximations using piecewise continuous functions. The MAP should include a process that defines the *initial* spatial discretization approach (perhaps based on rules of thumb and prior experience) as well as a path leading to mathematical model verification. For example:

- The ET TPS acreage SOFI will be modeled as a generalized plane strain problem with uniform pressure loading applied to the foam external surface. The rationale is that the ET has a large radius, is locally considered as flat, and the foam width extends the length of the ET. The TPSMOM analysis tool will be used for preliminary analysis. Detailed generalized plane strain finite element solutions will be developed to predict a detailed through-the-thickness stress state. Three-dimensional finite element models of the foam that are attached to a two-dimensional shell finite element model of the ET substrate will be developed to verify the plane-strain assumptions.

- The foam will be modeled as a linear elastic, isotropic material and the influence of material property variable will be assessed. The geometry of the foam model in the longitudinal direction will extend a distance approximately equal to five times the local foam thickness to ensure attenuation of unwanted edge effects. The generalized plane-strain analyses will use a nearly uniform mesh of square elements, having an edge dimension roughly equal to half the substrate thickness. The three-dimensional finite element models will include a square region in both the longitudinal and circumferential directions with several solid elements through-the-thickness of the foam – 8-node brick elements are acceptable, 20-node brick elements are preferred in order to better represent the local through-the-thickness bending response. The spatial discretization should result in well-formed, nearly uniform hexahedral elements or an assessment of the influence of the element distortion on the results should be performed.

Mathematical Model Verification

The mathematical model verification process of the MAP is used to determine the adequacy and accuracy of the mathematical model. Justification for the mathematical model definition should be documented in this step. Documentation to support the geometric definition of the structure, the material properties, and the loading conditions, including test fixtures and/or support structure attachment stiffness coefficients, need to be provided and documented. This information must define the problem sufficiently such that an independent analysis effort could replicate the analysis results.

The analysis tool or procedure should be clearly identified and described as well as any specific feature or requirement particular to a given tool. For example, analytical solutions based on series solutions should provide convergence results and solution sensitivities to analysis parameters. Discrete solutions such as those obtained using the finite element method need additional attention. Specifically, factors affecting solution accuracy include the selection of element type and associated options for that element; distribution of elements and nodes (*i.e.,* finite element meshing); definition of external mechanical loads, thermal loads, and boundary constraints; definition of local coordinate systems; and definition of solution procedure parameters.

Verification of material orientation relative to a computational coordinate system is also needed. Modeling materials with embedded reinforcement that include through-the-thickness stitching or Z-pins, structurally tailored designs with variable fiber orientation, and structures with complex geometric shapes and curvatures require special attention. Proper orientation and transformations need to be treated to ensure material directionality and structural-element orientation is maintained.

Most finite element pre-processing tools such as PATRAN can quickly and routinely perform basic finite element modeling checks. Element quality checks can be performed on the mesh to assess element distortion, element aspect ratio, element warping, element surface normal direction, and so on. Many finite element analysis tools provide optional model checks and/or model computations. Examples of these checks include mesh quality checks, a rigid-body-motion check, weight computation, and model inertia characteristics. The mathematical model verification step should document the basic checks performed and their outcome converted into performance metrics that provide a quantitative measure to judge model spatial discretization fidelity.

Loosely coupled multidisciplinary analyses include extracting temperatures from a thermal analysis or pressure loads from an aerodynamic analysis and imposing them on a stand-alone structural analysis model. Typically, the spatial discretization requirements for the individual disciplines are different because of differences in the response characteristics (*e.g.,* elliptic versus hyperbolic response characteristics); and hence, mapping functions or interpolation procedures for interfacing these results are needed.

These mapping functions must accommodate the result variable (*e.g.,* nodal temperatures) and any local gradients.

A systematic mesh-convergence study should be presented to verify: (1) the adequacy of the geometry modeling; (2) the adequacy of stiffness and mass modeling; (3) the adequacy of boundary condition restraints; and (4) the adequacy of local-gradient modeling in primary and secondary solutions (*i.e.,* displacements and stresses or strains, respectively). Mesh dependencies can result from boundary condition changes, changes in local-detail dimensions, and changes in material systems (*i.e.,* different structural configurations). For example, a converged finite element mesh for a simply supported plate is most likely not a converged mesh for a clamped plate because of the local bending gradients induced by the boundary restraints. Short-wavelength response phenomena require a refined finite element mesh. Knowing to anticipate the short-wavelength response requires experience and expertise on the part of the analyst. Mesh convergence studies are often *ad hoc* (*e.g.,* double the mesh or selectively re-mesh in one or more local regions). However, error estimators should be incorporated as part of mesh-convergence study. The *p*-version of the finite element method offers an inherent error estimation process, but unfortunately the p-version finite element tools are rarely used even though available (*e.g.,* MSC/NASTRAN offers a *p*-version elements, StressCheck* offers a *p*-version-based finite element stress analysis). Alternatively, multi-level finite element modeling techniques have been developed for embedding locally refined meshes into a larger, less-refined global finite element model to independent local finite element models with boundary conditions derived from a global model. The process for verifying the interface conditions between these modeling levels must be defined and demonstrated.

At this point, the concern is mainly verifying that the mathematical model of the problem is predicting a response consistent with the assumptions and the approach – not validating the correctness of the mathematical model. The outcome of this verification step is that the mathematical model accurately predicts an intended response and any response sensitivity to modeling assumptions is understood and mitigated.

Mathematical Model Validation

The mathematical model validation step of the MAP is a more tedious task. Model validation refers to the correctness of the mathematical model for the problem. This step involves making assessments of different modeling idealizations and approximations and determining if the proper physics is represented. By necessity, this step commonly results in the development of multiple mathematical models. Assuming that the model verification step has been performed, the model validation step now critically examines underlying assumptions and uncertainties of the mathematical model definition.

Common modeling issues to examine include: correctness of idealization assumptions; material overlap as a result of dimensional reduction in the idealization step; interface conditions between independently modeled subcomponents and global/local domains;

---

* http://www.esrd.com

joint and connection modeling, material constitutive models and response, linearity assumptions, contact and frictional interface modeling; and the role of generalized imperfections (see Refs. 23-24, 68). This step is neither an automated process nor one to be performed by a neophyte to the problem or the analysis tool. Questions addressed at this step often include:

- Do the results make physical sense?
- Do they relate to previous documented experiences?
- Is there any sensitivity to the idealization assumptions?
- What are the key response parameters?
- Are the results consistent with intuition?
- What is the confidence level in the results?
- Is the model applicable to other loading or boundary conditions definitions?
- Is the response truly quasi-static or transient?
- Would an eigenvalue analysis reveal response prediction limits?
- How far can the use of the model or results be extrapolated?

System Response and Sensitivities

System response and sensitivities are the products of the MAP. Key system response parameters are defined and trends presented to quantify the effects of known uncertainties. Graphs of key response parameters (*e.g.,* load vs. end shortening, contact force vs. time, stress intensity factor vs. crack length) are developed to support the objectives and goals of the analysis effort. Color contour plots provide a qualitative perspective view of the system response and general trends. These contour plots may identify the need for additional graphs. Most finite element post-processing tools offer a variety of response-parameter plotting options that should be used with care (*e.g.,* does it use element centroidal values, element nodal values, or smoothed nodal values?). Translation of results from a finite element tool, say NASTRAN, to a post-processing graphics tools, say PATRAN, typically introduces another set of approximations in terms of contouring response parameters or solution animation.

Using the verified and validated mathematical model, systems sensitivities can be assessed and incorporated into a nondeterministic analysis tool such as NESSUS[*]. This process then establishes bounds for response parameters to given problem uncertainties. At this point, the mathematical model should be capturing all the relevant physics for this problem definition and response sensitivities are defined for known uncertainties.

Configuration Management

Configuration management ensures that the distribution of analysis models is controlled and managed so that users of the analysis models can establish traceability for component geometry, material properties, load cases, and analysis tool version. Changes or modifications are frequently necessary in the geometry model definition as the design evolves and in the finite element model as mesh convergence studies and modeling details are added. For large complex systems involving multiple teams or groups or

---

[*] http://www.swri.org

contractors, model traceability becomes critical to ensure everyone has the same reference condition. Configuration management could be as simple as establishing a common location for accessing systems definitions (geometry, material data, loads, requirements, etc.). Easy access to read or copy data files from this central location would be provided to those requiring access, while controlling who has permission to "write" or update the data files.

Documentation and Approvals

The documentation and approval step ensures that the stake holders of the engineering models and analyses concur with the MAP and that sufficient resources and time are allocated for the effort. The stake holders may include independent government and industry teams, contractors and customer teams, or management and engineering staff. Resource estimates need to include effort for briefings and other reviews. The MAP is then reviewed, revised as necessary, and approved. Once the MAP is executed, formal presentation and documentation of the modeling and analysis effort should be accomplished. These documents (formal documents or presentations) would undergo a peer review prior to release and should include clear and definitive documentation that establishes the robustness and correctness of the approach and results. For example, distributed presentation charts may be required to be in a slide/facing-page narrative format so that the presenter's thoughts accompany the charts. This entire process for modeling and analysis parallels the test plan and test-report process, including preliminary "quick look" reports to summarize results and disseminate findings to the MAP stake holders. The result is a process that guides the engineering analysis team, establishes a process for model verification and validation, provides credibility and pedigree of results for managers, and provides confidence bounds for review boards. In addition, it provides an archival product for historical purposes and subsequent future needs.

Recommendations

This assessment of the structural analysis technology for the ET TPS has generated a series of recommendations. An overall fundamental recommendation is to pursue a careful and thorough review of the design-process philosophies proposed by Ryan *et al.* [69]. These philosophies and practices were developed based on experience and lessons learned from numerous NASA spacecraft-development programs. The summary and recommendations presented by Ryan *et al.* [69] should be integrated into every design process as a set of systems engineering requirements. While design specifics vary from vehicle to vehicle (or product to product) the fundamental principles and guidance should be incorporated in an effort to understand the system, to mitigate risk, and to improve safety and reliability.

The remaining recommendations are more specific and are divided into a short-term list (needed before return to flight) and a long-term list (achieved after return to flight to improve safety). These recommendations are focused on structural-analysis-related efforts for the ET TPS foam system.

The short-term recommendations include:

- Continue material characterization studies needed to establish the statistical basis of the material properties and the sensitivity of these properties to variations.

- Develop a mathematical constitutive model of the SOFI material suitable for implementation into a finite element code.

- Implement a SOFI material model in a commercial finite element analysis code (*e.g.,* using the UMAT feature of ABAQUS) and evaluate its performance using the data from coupon- and element-level specimen tests.

- Continue the specialized finite element modeling and analysis effort to support coupon-level and element-level SOFI specimen testing (monostrain test, Poisson's ratio test, lap shear test, compressive strength test, flatwise tension/bond tension test, torsion shear test, divot testing, and cyroflex tests) and to help identify failure modes and mechanisms.

- Verify the global-to-local finite element modeling methodologies for 2D-to-2D locally refined models as well as 2D-to-3D locally refined models. Testing of the kinematics along the global-local interface and continuity of both displacement and strain fields needs to be demonstrated. Extensions of the global-to-local methodologies to nonlinear-response applications are also needed.

- Draft a Modeling and Analysis Plan (MAP) for key analysis models that are influencing decision-making.

- Examine the use of *p*-version finite element technology such as StressCheck[*] for selected SOFI regions as a potential analysis approach for the acreage TPS.

The long-term recommendations include:

- Develop and verify an advanced TPSMOM-like analysis tool that accounts for curvature, material anisotropy, cellular material structure, pore-pressure effects, and temperature dependence. Leveraging of related research in structural adhesives should be pursued.

- Integrate the SOFI user-defined material models directly into the commercial finite element code (requires code vendor to perform this effort).

- Adopt the MAP concept as part of the acceptance criteria for analysis results.

---

[*] http://www.esrd.com

Summary

An assessment of the structural analysis technology applied to the Space Shuttle External Tank (ET) thermal protection system (TPS) has been performed that includes a review of the modeling and analysis tools used for the ET TPS. Strengths and weakness of the past and present approaches for these analyses have been identified and discussed. Short-term and long-term efforts have been identified that are needed to enhance the ET TPS structural analysis technology base.

The analytical ability to simulate the structural response of complex systems reliably is tied directly to the understanding and characterization of the constitutive model of the material system and its failure modes. Lack of a verified material model for the SOFI appears to be a significant weakness in the past and present structural analysis work. In the past, during the early design years of the ET system, computing power was limited and finite element technology was in its formative years. Now high-performance computing systems are readily available and pre- and post-processing graphical user interfaces (GUI) make solving large-scale computational models and interpreting the results relatively straightforward. However, large-scale finite element models are not necessarily high-fidelity analysis models that capture adequately the physics of the response. Therefore, as the analysis tools increase in capabilities, and as the computing environments enable larger and larger computational models, analysts need to exercise an increasing amount of due diligence in scrutinizing their finite element models and results prior to formal presentation.

Because of the practical limitations on structural verification testing, reliance on analysis as part of verification and certification processes is necessary, and hence a responsibility is placed on the analysts to verify and validate analytical predictions. An accountability process or plan is needed. To this end, it is proposed that a formal modeling and analysis plan or MAP be developed as part of an acceptance criteria for any analysis effort (*i.e.,* simplified analysis, finite element analysis, and boundary element analysis). The generation of a MAP could be viewed as a burden on the analyst in the short term; however, in the long term, it offers many advantages. The development of a MAP provides a mechanism to document the modeling and analysis effort, to define rationale for approach and assumptions, to provide a solid basis for acceptance of the results, to mentor other analysts, and to capture corporate memory of the current analysts. As a result, analysts, managers, and reviewers would have a document that can be used to judge the "pedigree" of the analysis effort and its results, and a well-defined level of confidence as to how much weight the analysis should be given in a decision process. This MAP process would then contribute to preventing analytical results from being used out of context or from being extrapolated outside their range of validity. In addition, time wasted re-executing and re-examining analyses that were conducted during early stages of product development would be eliminated. Overall, the robustness of the design (*i.e.,* understanding of system sensitivities) would be increased and time and resources would be saved.

References

1. Anon., *Space Shuttle External Tank System Definition Handbook SLWT, Volume 1 – Configuration and Operation*, Lockheed Martin Michoud Space Systems, Report No. LMC-ET-SE61-1, December 1997.
2. Columbia Accident Investigation Board (CAIB) Report, Volume 1, August 2003.
3. Columbia Accident Investigation Board Report, Volume 4, October 2003, Appendix F.1 Water Absorption by Foam, pp. 7-18.
4. Columbia Accident Investigation Board Report, Volume 4, October 2003, Appendix F.2 Follow the TPS, pp. 19-52.
5. Columbia Accident Investigation Board Report, Volume 4, October 2003, Appendix F.4 ET Cryoinsulation, pp. 179-238.
6. Columbia Accident Investigation Board Report, Volume 4, October 2003, Appendix F.5 Space Shuttle STS-107 Columbia Accident Investigation, External Tank Working Group Final Report – Volume 1, pp. 239-395.
7. del Casal, E. P., "Debris from Spallation of Foam Insulation of Cryogenic Fuel Tanks in Space Launch Systems," AIAA Paper No. 83-1457, June 1983.
8. Gibson, Lorna J. and Ashby Michael, *Cellular Solids: Structure and Properties*, Second Edition, Chapter 5, Cambridge University Press, 1997.
9. Weiser, Erik and St. Clair, Terry, *Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement – Part I: Material Characterization and Analysis*, NASA TM-2004-213238, 2004
10. Anon., ET Stress Analysis Report, Section E – Thermal Protection System, EAS No. 3521-826-2188.
11. Oliver, Stan, "Analysis of TPS Verification Test Configurations," presentation at TPS Verification Team Analysis Technical Interchange Meeting, NASA Michoud Assembly Facilities, New Orleans, LA, November 20-21, 2003 (24 charts total including backup charts).
12. Sun, Z., Wan, K.-T., and Dillard, D. A., "A Theoretical and Numerical Study of Thin Film Delamination Using the Pull-Off Test," *International Journal of Solids and Structures*, Vol. 41, No. 3-4, February 2004, pp. 717-730.
13. Ferracin, T., Landis, C. M., Delannay, F., and Pardoen, T., "On the Determination of the Cohesive Zone Properties of an Adhesive Layer from the Analysis of the Wedge-Peel Test," *International Journal of Solids and Structures*, Vol. 40, No. 11, June 2003, pp. 2889-2904.

14. Cui, J., Wang, R., Sinclair, A. N., and Spelt, J. K., "A Calibrated Finite Element Model of Adhesive Peeling," *International Journal of Adhesion and Adhesives*, Vol. 23, No. 3, 2003, pp. 199-206.

15. Sun, D. and Tong, L., "Modeling and Analysis of Curved Beams with Debonded Piezoelectric Sensor/Actuator Patches," *International Journal of Mechanical Sciences*, Vol. 44, No. 8, August 2002, pp. 1755-1777.

16. Olia, M. and Rossettos, J. N., "Analysis of Adhesively Bonded Joints with Gaps Subjected to Bending," *International Journal of Solids and Structures*, Vol. 33, No. 18, 1996, pp. 2681-2693.

17. Blosser, Max L., "Boundary Conditions for Aerospace Thermal-Structural Tests," in *Aerospace Thermal Structures and Materials for a New Era*, Earl A. Thornton (editor), Progress in Astronautics and Aeronautics, Vol. 165, AIAA, 1995, pp. 119-144.

18. Nemeth, M. P., Britt, V. O., Young, R. D., Collins, T. J., and Starnes, J. H., Jr., "Nonlinear Behavior of the Space Shuttle Superlightweight Liquid-Oxygen Tank Under Prelaunch Loads," *Journal of Spacecraft and Rockets*, Vol. 36, No. 6, November-December 1999, pp. 788-803.

19. Young, R. D. and Rankin, C. C., "Modeling and Nonlinear Structural Analysis of a Large-Scale Launch Vehicle," *Journal of Spacecraft and Rockets*, Vol. 36, No. 6, November-December 1999, pp. 804-811.

20. Nemeth, M. P., Young, R. D., Collins, T. J., and Starnes, J. H., Jr., "Effects of Welding-Induced Imperfections on Behavior of Space Shuttle Superlightweight Tank," *Journal of Spacecraft and Rockets*, Vol. 36, No. 6, November-December 1999, pp. 812-819.

21. Young, R. D., Nemeth, M. P., Collins, T. J., and Starnes, J. H., Jr., "Nonlinear Behavior of Space Shuttle Superlightweight Tank Under Booster Ascent Loads," *Journal of Spacecraft and Rockets*, Vol. 36, No. 6, November-December 1999, pp. 820-827.

22. Nemeth, M. P., Young, R. D., Collins, T. J., and Starnes, J. H., Jr., "Nonlinear Behavior of Space Shuttle Superlightweight Tank Under End-of-Flight Loads," *Journal of Spacecraft and Rockets*, Vol. 36, No. 6, November-December 1999, pp. 828-835.

23. Nemeth, M. P. and Starnes, J. H., Jr., *The NASA Monographs on Shell Stability Design Recommendations – A Review and Suggested Improvements*, NASA TP-1998-206290, January 1998.

24. Starnes, J. H., Jr., Hilburger, M. W., and Nemeth, M. P., "The Effects of Initial Imperfections on the Buckling of Composite Cylindrical Shells," in *Composite Structures: Theory and Practice*, ASTM STP 1383, P. Grant and C. Q. Rousseau

(editors), American Society for Testing and Materials, West Conshohocken, PA, 2000, pp. 529-550.

25. Neilsen, M. K., Morgan, H. S., and Krieg, R. D., *A Phenomenological Constitutive Model for Low Density Polyurethane Foams*, Sandia National Laboratories, Report SAND86-2927, April 1987.

26. Lu, W. Y., Korellis, J. S., Lee, K. L., and Grishaber, R., *Hydrostatic and Uniaxial Behavior of a High-Density Polyurethane Foam (FR-3720) at Various Temperatures*, Sandia National Laboratories, Report SAND93-8227, March 1993.

27. Chang, Fu S., *Constitutive Equation Development of Foam Materials*, Ph. D. Dissertation, Department of Mechanical Engineering, Wayne State University, Detroit, MI, April 1995.

28. Hirth, A., "A Material Model for Transversely Anisotropic Crushable Foams in LS-DYNA," Proceedings of the Seventh International LS-DYNA Users Conference, 2002, pp. 16-23 to 16-34.

29. Serifi, E., Hirth, A., Matthaei. S., and Mullerschon, H., "Modeling of Foams using MAT83 – Preparation and Evaluation of Experimental Data," Proceedings of the Fourth European LS-DYNA Users Conference, 2003, pp. D-II-59 to D-II-71.

30. Endebrock Elton G., *Load-Rate Characteristics of Certain Polyurethane Foams*, Los Alamos National Laboratory, Report LA-10567-MS, November 1985.

31. Neilsen, M. K., Krieg, R. D., and Schreyer, H. L., "A Constitutive Theory for Rigid Polyurethane Foam," in *Use of Plastics and Plastic Composites: Materials and Mechanics Issues*, ASME, Vol. 46, 1993, New Orleans, LA, pp. 181-194. Also as Sandia Report No. SAND92-2487C.

32. Chou, C. C., Zhao, Y., Chai, L., Co, J., Lim, G. G., and Lin, T. L., "Development of Foam Models as Applications to Vehicle Interior," SAE Paper No. 952733, Proceedings of the 39th STAPP Car Crash Conference, San Diego, CA, 1996.

33. Chang, F. S., Hallquist, J. O., Lu, D. X., Shahidi, B. K., Kudelko, C. M., and Tekelly, J. P., "Finite Element Analysis of Low-Density High-Hysteresis Foam Materials and the Application in the Automotive Industry," SAE Paper No. 940908, 1994.

34. Chou, C. C., Zhao, Y., Lim, G. G., Patel, R. N., Shahab, S. A., and Patel, R. J., "Comparative Analysis of Different Energy Absorbing Materials for Interior Head Impact," SAE Paper No. 950332, 1995.

35. Chou, C. C., Liu, N., Zhao, Y., Wu, C.-C., Lim, G. G., Patel, R. N., and Walker, L., "Testing, Analysis and Finite Element Modeling of Polyurethane Foams for Upper Vehicle Interior Applications," Proceedings of International Body Engineering Conference, Automotive Body Interior and Safety Systems, Volume 23, S. Kelkar (editor), October 1-3, 1996, Detroit, MI, pp. 52-62.

36. Chou, C. C., Zhao, Y., Lim, G.G., and Song, G. S., "A Constitutive Model for Polyurethane Foams with Strain-Rate and Temperature Effects," SAE Paper No. 980967, 1998.

37. Theocaris, P. S., "Failure Modes of Closed-Cell Polyurethane Foams," *International Journal of Fracture*, Vol. 56, 1992, pp. 353-375.

38. Weiser, E. S., Johnson, T. F., St. Clair, T. L., Echigo, Y., Kaneshiro, H., and Grimley, B. W., "Polyimide Foams for Aerospace Vehicles," *High Performance Polymers*, Vol. 12, 2000, pp. 1-12.

39. Chen, W. F., "Constitutive Modelling in Soil Mechanics," in *Mechanics of Engineering Materials*, C. S. Desai and R. H. Gallagher (editors), John Wiley & Sons, New York, 1984, pp. 91-120.

40. Chen, W. F. and Mizuno, E., *Nonlinear Analysis in Soil Mechanics: Theory and Implementation*, Elsevier, Amsterdam, 1990.

41. Knight, Norman F., Jr. and Stone, Thomas J., *Rapid Modeling and Analysis Tools: Evolution, Status, Needs and Direction*, NASA CR-2002-211751, July 2002.

42. Roache, Patrick J., "Verification of Codes and Calculations," *AIAA Journal*, Vol. 36, No. 5, May 1998.

43. Roache, Patrick J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, NM, 1998.

44. Anon., *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*, AIAA G-077-1998 Guide, May 1998.

45. Basu, R. I., Kirkhope, K. J., and Srinivasan, J., *Guidelines for Evaluation of Ship Structural Finite Element Analysis*, Ship Structures Committee, U. S. Coast Guard, Report No. SSC-387, December 1995.

46. Cook, R. D. and Avrashi, J., "Error Estimation and Adaptive Meshing for Vibration Problems," Computers and Structures, Vol. 44. No. 3, pp. 619-626, 1992.

47. Campbell, Mark E., "Uncertainty Effects in Model-Data Correlation," AIAA Paper No. 97-1030, 1997.

48. Hemez, Francois, M. and Brown, Gregory W., "Improving Structural Dynamics Models by Correlating Simulated to Measured Frequency Response Functions," AIAA Paper No. 98-1789, 1998.

49. Hasselman, T. K., Zimmerman, D. C., and Herendeen, D. L., "An Integrated FEA Software Capability for Dynamic Model Validation and Verification," AIAA Paper No. 99-1595, April 1999.

50. Sarkar, A. and Ghanem, R., "Mid-Frequency Structural Dynamics with Parameter Uncertainty," AIAA Paper No. 2001-1454, April 2001.

51. Horta, Lucas G., Reaves, Mercedes C., and Voracek, David F., *A Probabilistic Approach to Model Update*, NASA TM-2001-18097, June 2001.

52. Kacprzynski, Jerzy, *Reliability Studies of Finite Element Methods in North America*, AGARD Report No. 748, 1987.

53. Stockwell, Alan E., *A Verification Procedure for MSC/NASTRAN Finite Element Models*, NASA CR-4675, June 1995.

54. Oberkampf, William L., "Methodology for the Estimation of Uncertainty and Error in Computational Simulation," in *Nondeterministic Approaches and Their Potential for Future Aerospace Systems*, A. K. Noor (compiler), NASA CP-2001-211050, September 2001, pp. 67-80.

55. Thacker, Ben H., "The Role of Nondeterminism in Verification and Validation of Computational Solid Mechanics Models," SAE Paper 2003-01-1353, in *Reliability and Robust Design in Automotive Engineering*, SP-1736, SAE International, SAE 2003 World Congress and Exposition, Detroit, MI, March 2003.

56. Noor, A. K. (compiler), *Nondeterministic Approaches and Their Potential for Future Aerospace Systems*, NASA CP-2001-211050, September 2001.

57. Fragola, Joseph R., *Probabilistic Risk Assessment of the Space Shuttle: Phase 3 – A Study of the Potential Losing the Vehicle During Nominal Operations, Volume 5: Auxiliary Shuttle Risk Analyses*, NASA CR-197812, 1995.

58. Knight, N. F., Jr., Glaessgen, E. H., and Sleight, D. W., "Probabilistic Risk Assessment for Damage-Tolerant Composite Spacecraft Component Structural Design," *Journal of Spacecraft and Rockets*, Vol. 40, No. 1, January-February 2003, pp. 72-82.

59. Kardestuncer, H. and Norrie, D. H., *Finite Element Handbook*, McGraw-Hill Book Company, New York, 1987.

60. Cook, R. D., Malkus, D. S., and Plesha, M. E., *Concepts and Applications of Finite Element Analysis*, Third Edition, John Wiley & Sons, New York, 1989.

61. MacNeal, Richard H., *Finite Elements: Their Design and Performance*, Marcel Dekker, Inc., New York, 1994.

62. Bathe, K.-J., *Finite Element Procedures*, Prentice Hall, Englewood Cliffs, New Jersey, 1996

63. Knight, N. F., Jr., "Finite Element Techniques for Nonlinear Postbuckling and Collapse of Elastic Structures," in *Structural Dynamic Systems, Computational Techniques, and Optimization: Finite Element Techniques*, Cornelius T. Leondes (editor), Gordon and Breach Science Publishers, Australia, 1998, pp. 1-95.

64. Maupin, R. D., Luscher, D. J., Hemez, F. M., and Doebling, S. W., "Verification of Calculations Using the Grid Convergence Index for Structural Finite Element Analyses," presented at the Seventh US National Congress on Computational Mechanics, July 27-31, 2003, Albuquerque, NM.

65. Jones, Robert M., "Reflections on the Importance of Experimental Results to All Mechanicists, Especially Theoreticians," in *Recent Advances in Experimental*

Mechanics: In Honor of Isaac M. Daniel, Emmanuel E. Gdoutos (editor), Kluwer Academic Publishers, Dordrecht, 2002, pp. 537-550.

66. Roarke, R. J. and Young, W. C., *Formulas for Stress and Strain, Fifth Edition*, McGraw-Hill Book Company, 1982.

67. Blevins, Robert D., *Formulas for Natural Frequency and Mode Shape*, Krieger Publishing Company, Malabar, Florida, reprint edition, 2001.

68. Nemeth, M. P., Young, R. D., Collins, T. J., and Starnes, J. H., Jr., “Effects of Initial Geometric Imperfections on the Nonlinear Response of the Space Shuttle Superlightweight Liquid-Oxygen Tank,” *International Journal of Nonlinear Mechanics*, Vol. 37, 2002, pp. 723-744.

69. Ryan, R., Blair, J, Townsend, J, and Verderaime, V., *Working on the Boundaries: Philosophies and Practices of the Design Process*, NASA TP 3642, July 1996.

Appendix A – Finite Element Modeling and Analysis Terminology

Finite element models and analysis results are deemed “acceptable” and “correct” based on a wide spectrum of approaches – some are sound and deliberate while others are based on “appearance” of the finite element model. Often an acceptance of the finite element results is because the finite element model involves a large number of elements (large-scale), the results are displayed in color and perhaps animated, and the results appear to be consistent with the perceived physics of the behavior. It is becoming increasingly necessary to identify a series of gates that finite element models need to get through in order to define “certified models.” Fundamental to this question of “certified models” is having an understanding of the overall system being analyzed and its component interaction, a clear definition of the goals and objectives of the analysis, and an understanding of the anticipated structural response.

The purpose of the finite element model and analysis needs to be clearly defined and stated so that those involved with performing the analysis and those involved with reviewing results from those analyses are on common ground as to the outcome of the finite element analyses. Some common purposes for performing finite element analyses include:

- To determine internal load paths
- To determine displacement fields and deformation patterns
- To predict detailed local stress fields
- To characterize anticipated structural response (quasi-static, transient, eigenvalues)
- To perform exploratory, developmental, preliminary or certification analyses for understanding and insight
- To develop component-level analysis models for use in a larger integrated systems model

Next, the analyst must strive to understand the structural response that is anticipated, based on the applied loading and/or type of analysis. The structural response may involve one or more sources of nonlinearities as indicated in this list:

- Elastic, elasto-plastic, progressive damage (materially linear or nonlinear)
- Small deflection, large deflection, large rotations (geometrically linear or nonlinear)
- Evolving boundary/surface conditions (contact or slip)
- Quasi-static or time dependent

Characterization of finite element model fidelity (consistent with analysis goals) also needs to be grasped by the analyst. Often, a finite element model developed for one purpose is used for a different purpose, assuming that the model fidelity transfers with it directly. This assumption is generally untrue – a high-fidelity loads model is not a high-fidelity stress model; a high-fidelity model for a quasi-static analysis is not a high-fidelity model for a transient-dynamics analysis. Basic definitions of different modeling fidelity levels include:

- Low-fidelity modeling – captures basic geometry and overall response well enough for trends and trade-studies); could include closed-form solutions, approximate analytical solutions

- High-fidelity modeling – captures all physics important to meet analysis goal; could be a coarse mesh or refined mesh depending on the analysis need, the geometric modeling requirements, and the physical response being simulated.

- Hierarchical modeling – a functional definition that refers to the use of different fidelity models to understand the structural response; while a mathematical definition refers to the use of solution techniques that include an ever increasing set (*e.g.,* more terms in a series solution)

Finite element modeling fidelity is a concept that needs careful study. Mesh refinement – the process of locally or globally refining the finite element mesh (referred to as *h*-, *p*-, *hp*- or *r*-refinement) – can be used to assess the sensitivity of solution to spatial discretization where a minimal change in results is a necessary, but not sufficient, condition for mesh convergence. The common terminology associated with mesh refinement includes:

- *h*-refinement is subdividing an element into additional elements and may require underlying geometry definition (numbers of nodes, elements and degrees of freedom increase)

- *p*-refinement is increasing the polynomial approximation within elements without any change to spatial discretization (number of elements stays the same but number of degrees of freedom per element increases)

- *hp*-refinement is a combination of *h*- and *p*-refinement

- *r*-refinement is the relocation of existing nodes within the mesh to improve the results (numbers of nodes, elements, and degrees of freedom stays the same; effect of element distortion needs to be assessed)

Finite element results based on models composed of large numbers of finite elements that take days to run on even high-performance computing systems have the potential for misinterpretation if there is no familiarity with that finite element model development or its solution. Experience is the teacher in these large-scale, complex engineering simulations. The analysis tools themselves provide easy access to mesh refinement procedures without any direct assessment of modeling or analysis error. For example, doubling the mesh is one common approach to verifying the adequacy of a given finite element mesh. However, if the main driver in the structural response is contact and it is ignored in the analysis, then no amount of mesh refinement will capture the true physics.

To develop a validated finite element model, the analyst needs to consider the following aspects and document their sources and any and all assumptions made:

- Geometry (surfaces, solids, discrete points)

- Verify the underlying geometry of the FE model
    - 2D models (IML or OML or mid-surface)
    - 3D models (solid geometry)

- Subcomponent geometry interface requirements, if needed

- Structural idealization process
    - Can parts be modeled as 1D rods or beams?
    - Can parts be modeled as 2D membranes, plates or shells?
    - Can fillets or holes be ignored?
    - Is there material overlap near 2D surface intersections?
    - Should bolted or riveted connections be treated as discrete connections or smeared?
    - Approximations of curved edges and curved surfaces
    - Stiffness representation of supporting structure or explicit model?
    - Use of symmetry assumptions (geometry, loading, material)

Spatial discretization refers to finite element meshing; it is the process of taking the idealized model of the structure and laying out the finite element nodes and elements on the idealized geometry. The finite element modeler, in conjunction with the analyst, must decide upon various choices to be made. These choices include:

- Element order (linear, quadratic, etc.)

- Element type (1D, 2D, 3D; displacement based or alternate formulation)

- Element shape (quadrilaterals, triangles, hexahedrals, wedges, tetrahedrals)

- Element geometry (curved or straight edges or faces; curved or flat surfaces)

- Distribution of elements and nodes (mesh grading, mesh transition modeling)
    - Number of elements
    - Through-the-thickness representation

- Boundary/surface condition definition

- Analytical definition of boundary conditions (clamped, hinged, simple support) compared to direct modeling of the hardware interface/boundary conditions (fixture modeling)

Material model definition (*i.e.,* the form of the constitutive relations) is one of the more critical finite element modeling decisions that needs to be made. Analysts need to understand the basis of the material model being selected, the required material data for input, and the limitations of the material model. To a certain extent, the material model selection is tied to the anticipated structural response of the system. For example, if the loading is anticipated to be monotonic loading with no unloading occurring then one material model may be selected over another; whereas, if loading and unloading cycles are to be predicted, a different material model may be desirable. Material data needed include:

- Stress-strain data available (tension and compression)
- Elastic mechanical properties
- Strength allowables
- Fracture parameters
- Failure modes and mechanisms

Loading definition also requires modeling decisions to be made and can have an impact on the predicted structural response. Direct application of point forces and moments need to be understood relative to the local effects and St. Venant's problem. Finite element formulations can have an impact on the "modeled" load as well. For example, if a user manually defines a set of work-equivalent nodal loads rather than specifying a distributed line load and the element formulation includes so-called "drilling freedoms," then the applied nodal moments associated with the drilling freedoms is often ignored. Specific questions about the loading are related to the following items:

- Discrete forces and moments

- Distributed surface pressures and running edge (or line) loads

- Proportional loading assumption

- Sequence of loading (apply one loading up to a certain level and then add an additional loading) and combined mechanical and thermal load cases for load interaction studies

- Mechanical, thermal and vibro-acoustic loads – applied independently or in combination

- Quasi-static "snapshot" loads

- Time-dependent loading

- Deformation-dependent loading (follower forces)

Solving the resulting finite element definition of the problem is perhaps the next step in the modeling and analysis process. A natural sequence of analyses can be proposed. First, a linear elastic stress analysis of the finite element model should be performed and verified. The next step would depend on the main purpose of the analysis effort. If it is more focused on structural dynamics, then a linear free vibration eigenvalue analysis is needed. If the loading is quasi-static, then a linear bifurcation buckling analysis may be in order depending on the type of loading. Nonlinear quasi-static stress analyses are often the next step. Deciding whether a nonlinear analysis is really needed or not can be a challenge. Solving the problem with the full load level applied using nonlinear solution procedures can be attempted. If the nonlinear analysis converges easily (*e.g.,* two or three iterations) then the response is probably only mildly nonlinear and a check of the stress distributions corresponding to the nonlinear solution should reveal whether the nonlinearities really had an influence. If the nonlinear analysis does not converge without reducing the load step size, or if it requires a large number of iterations, then most likely nonlinear effects are important. Specific solution controls for nonlinear solution procedures include:

- Convergence metrics (change in residuals, change in displacement increments, change in energy; relative measures or absolute measures)

- Specified convergence tolerance – too small and no solution is obtained; too large and the solution will "drift" from solution equilibrium

- Solution control procedure (load control, displacement control, arc-length control)

- Nonlinear solution algorithm (Newton-Raphson procedure, modified Newton-Raphson procedure, quasi-Newton procedures)

- Number of negative roots in the tangent stiffness matrix decomposition (more than the number of Lagrange-multiplier constraints?)

Transient-dynamics analyses may be required to capture time-dependent effects and/or loading events. Solution procedures for linear structural dynamics simulations generally use modal methods for predicting response time histories for large-scale problems. The selection of the appropriate eigenmodes and the choice of how many to include in forming the solution subspace have an effect on the solution accuracy. Alternatively, direct time integration procedures for the semi-discrete finite element equations can also be performed in the event the spatial distribution of the applied load is not readily represented by a fixed set of vibration mode shapes or Ritz vectors. Direct time integration procedures are classified as either explicit methods or implicit methods. Explicit methods usually approximate the time derivative terms in the equations of motion using a central-difference operator. The resulting algebraic equations are then linear algebraic equations at each time step – even for severe nonlinearities. If the mass matrix is diagonalized (*i.e.,* lumped-mass approach), the solution of these linear equations (regardless of the type or degree of nonlinearity) is trivial for each time step. The drawback is that explicit methods are generally only conditionally stable numerically thereby requiring a time-step size smaller than a critical value for the given spatial discretization. A typical size of a time step is often in the tenths of microseconds.

Implicit methods usually approximate the time derivative terms using the Newmark-β, the Wilson-θ, or linear multi-step operators. The resulting system of algebraic equations requires a solution using Gauss elimination or LU-decomposition procedures. Implicit procedures are generally unconditionally stable (numerically) and their accuracy is dependent on the time step size.

For structural systems exhibiting a complex nonlinear response, predicting the overall nonlinear behavior often cannot be done by using only quasi-static methods or transient dynamics methods alone. Hybrid static-dynamic solutions procedures are becoming increasingly popular and available in commercial finite element tools. Essentially, a quasi-static analysis is performed up to a point where convergence difficulties are encountered (possibly due to structural collapse, local material failures, or bifurcation of the solution response). At that point, a transient solution is started with a slight increase in load and the transient analysis is continued until the dynamic effects attenuate (*i.e.,* near zero kinetic energy, near zero inertial loading, or near zero velocity components over a range of time). Then, a static restart is performed where first equilibrium is re-established from the transient analysis by using a load-relaxation procedure and then new solutions for continued loading are determined. Procedures to automate the transitioning process between different solution methods are current research topics.

Eigenvalue analyses often provide insight into the state of the finite element model even when an eigenvalue analysis is not required. Extracting eigenpairs (eigenvalues and corresponding eigenvectors) gives information about the free vibration and/or bifurcation buckling response of the structure. The eigenvectors (or mode shapes) give an indication of the anticipated deformation patterns that may be expected and the adequacy of the finite element mesh to represent those patterns. Consideration should be given to these items:

- Buckling analysis for given applied load (linear vs. nonlinear prestress state)
- Vibration analysis for an unstressed or pre-stressed (linear or nonlinear) state
- Convergence criteria for the eigenvalue analysis
- Solution procedure for extracting the eigenpairs (is the subspace sufficiently spanned to represent the deformation states?)
- Influence of finite element meshing (can short-wavelength mode shapes be represented by the given finite element mesh?)

Basic finite element modeling checks are again necessary steps that the analyst needs to take as part of finite element model checkout process. These basic checks provide a sanity check on the overall modeled geometry and mass properties, as well as basic checks on finite element meshing related to element distortion, connectivity, and local coordinate systems. Finite element modeling and discretization tests include:

- Rigid-body displacement check
- Model weight calculation
- Model center of mass, overall inertia characteristics

- Mesh criteria (element aspect ratio, element skew, element taper, and element quality checks typically provided by the finite element meshing software)
- Surface normals for 2D elements
- Boundary lines (tests for *free* element edges)
- Resultant loadings

Demonstrating convergence of finite element solutions for the given assumptions is a necessary step in verifying the accuracy of the numerical solution. Because of the accessibility of high-performance computing systems and the user-friendliness of nonlinear finite element computer codes and tools, an analyst can be easily lured into accepting a finite element solution solely based on its general appearance. That is, the finite element mesh "looks" like the system, it may be three-dimensional in form and involve a large number of elements and nodes, and the results may be depicted using animation post-processing tools. These characteristics describe a large-scale computational model, but do not address the issue associated with the solution fidelity – the ability to capture all the relevant physics and underlying mechanics. Careful review of the finite element model of the structure (mesh definition, material modeling, loading definition, boundary specification, coordinate system definitions) is required to determine the sensitivity of the predicted response to changes in one or more aspects of the finite element model. Establishing this sensitivity can be achieved by using a number of different approaches, including:

- Mesh convergence studies – Typically local mesh refinement is performed rather than wholesale mesh doubling – partly due to complex geometry issues and overall computational cost. Changing the element type or element approximation order from one formulation to another can also be used to provide an indication of solution sensitivity.

- Multiple finite element tools – Solving the same basic finite element definition of the system using different finite element tools can provide added assurance that a given finite element model solved using one tool is consistent for the given set of approximations and assumptions. It does not indicate that all relevant response characteristics are included.

- Demonstrated building-block approach for model complexity – This approach starts by developing finite element models of simpler physical problems that *morph* to the actual physical problem. For example, if the real problem is a plate with a circular hole, then first model a rectangular plate with no hole and uniform mesh; then model plate with a hole except fill in the hole with triangles (now just a rectangular plate with an odd mesh to see the influence of mesh distortion on the stress state); then model plate with the hole by removing the triangles. Another example would be the use of a linear elastic material model, then a nonlinear elastic material model, and finally a material model accounting for material failures.

- Multi-level finite element modeling – This approach could involve two different starting points. One start point is the global-local approach commonly used in linear stress analyses that extracts displacements from a global solution and applies them as boundary conditions on a separate, independently modeled, local model of some detail in the structure. The other starting point involves embedding the local model within the global finite element model and subsequent re-solving of the new multi-level finite element model. Procedures such submodeling, mesh zooming, interface element modeling, and so forth are common in commercial finite element codes.

Postprocessing computed finite element results is straightforward but still requires careful attention and an understanding as to what the post-processing software does with the finite element results in order to generate contour plots, deformed geometry plots, and animations of the solutions. The analyst should be aware of how the post-processing software treats:

- Nodal results (primary results or secondary results)
- Elemental results (centroidal values or integration-point values)
- Graphing requirements
- Contour plotting requirements
- Deformed geometry plotting requirements (scaling)
- Animation of computed solutions (mode shapes, deformed shapes, nonlinear solutions, stress results)
- 3D visualization options using immersive technology
- Coordinate systems used to display results (e.g., material coordinate system, element coordinate system, or global structural coordinates)

Appendix B – Candidate MAP Format

This appendix describes the basic content and format of a candidate Modeling and Analysis Plan or MAP. This organization should be considered as a guide to be tailored to specific applications and efforts. That is, the basic contents of a MAP should include:

- Title page including signatures and approvals for modeling and separately for analysis
- Section 1 – problem statement, analysis objectives, and scope including needs and requirements estimate
- Section 2 –identify sources of error and uncertainty that potentially will affect the analysis
- Section 3 – identify modeling approach and rationale including data sources and responsibilities
- Section 4 – define the model verification process and success criteria
- Section 5 – define the model validation process and success criteria
- Section 6 – present results and discussion; perform production runs and describe system response and parameter sensitivities
- Section 7 – present conclusions and recommendations
- References (as necessary)

To illustrate the procedure, a sample MAP is given in Appendix C with suggested subsections and information. This sample is only intended as a guide and is representative of the information that should be captured in a MAP.

Appendix C – Sample MAP

Modeling and Analysis Procedure No. __________

Title: ET TPS Acreage SOFI Assessment

Modeling Effort

Prepared by: _____________________ Date: ____________
Norm Knight
Stress Analyst, GDAIS

Approved by: _____________________ Date: ____________
Mark Hilburger
Team Lead, Stress, MDB/NASA LaRC

_____________________ Date: ____________
Mike Nemeth
Manager, ET TPS Verification Team, MDB/NASA LaRC

Analysis and Results Efforts

Performed by: _____________________ Date: ____________
Norm Knight
Stress Analyst, GDAIS

Approved by: _____________________ Date: ____________
Mark Hilburger
Team Lead, Stress, MDB/NASA LaRC

_____________________ Date: ____________
Mike Nemeth
Manager, ET TPS Verification Team, MDB/NASA LaRC

Type of MAP (check one):

___ Exploratory analyses– no results will be used for certification or qualification
___ Preliminary analyses – some results may be used to satisfy certification or qualification
___ Certification/qualification analyses – results will be used for certification or qualification

1. Problem Statement
   1.1. Overview

   The ET TPS acreage SOFI is subject to a variety of loadings and environments. The basic SOFI geometry is a function of location on the ET (*i.e.,* open or acreage regions, where an automated application process is used, to close-out regions, were hand-spraying is used). The modeling and analysis effort will support coupon- and element-level testing and be extended to the acreage SOFI as appropriate.

   1.2. Analysis Objectives

   To model and analyze the structural response of the ET TPS acreage SOFI to assess the risk associated with SOFI disbond from ET substrate. To determine the local through-the-thickness stress state from the substrate through the foam, including transverse stresses associated with peeling and shearing.

   1.3. Scope

   Analyses will be performed based on room-temperature, linear-elastic material behavior using time-independent, uniformly distributed loading. Solutions in the longitudinal and circumferential directions will be developed.

   1.4. Terminology and Nomenclature

   1.5. Needs and Requirements

   Basic SOFI material data including elastic modulus, Poisson's ratio, and stress-strain curve. Basic geometry definition to support coupon-level and element-level testing. Basic loading definition, loading rates, fixture definition, and data acquisition plan.

   Access to TPSMOM code, NASTRAN or ABAQUS finite element codes, PATRAN or I-DEAS pre- and post-processing tools, and computing resources for modeling and analysis.

2. Sources of Error and Uncertainty
   2.1. Assumptions

   Treat the SOFI foam as a linear elastic homogenous isotropic continuum. Because the ET is a large diameter thin-walled shell, the effects of shell curvature are neglected. Loading is assumed to be uniform over the exterior surface and constant in time, and substrate deformations are ignored. Known failure modes to be included are substrate/SOFI disbond and delaminations.

   2.2. Sources of Input

   Basic material characterization data for mechanical properties from tests. Measured test specimen geometry and configuration.

   2.3. Sources of Error

Potential limitation due to the assumption that the SOFI is a linear-elastic, homogeneous isotropic continuum.

2.4. Sources of Uncertainty

Application of the SOFI leads to significant variability in material chemistry and resultant material properties. Minimize the generation of voids and pockets through process control; however, the nature of the SOFI itself is susceptible to defects. Statistical variation of SOFI mechanical properties and failure mechanisms is needed.

2.5. Limitations

The analyses performed under this MAP support the assessment of a flat substrate and a plane strain formulation of the SOFI. This assumption limits the failure mode representation of the mathematical model to being within the plane. Elastic material response assumptions limit the predictions to the onset of debond, delamination, or damage growth; progressive damage modeling is beyond the limits of these analyses.

3. Modeling Approach and Rationale

3.1. Mathematical Model Definition

The ET TPS acreage SOFI will be modeled as a generalized plane strain problem with uniform pressure loading applied to the foam external surface. The TPSMOM analysis tool will be used for preliminary analysis. Detailed generalized plane strain finite element solutions will be developed to predict the detailed through-the-thickness stress state. Three-dimensional finite element models of the foam attached to a shell finite element model of the ET substrate will be developed to verify the plane strain assumptions.

3.2. Assumptions

The foam will be modeled as a linear elastic isotropic material and the influence of material property variable will be assessed. The geometry of the foam model in the longitudinal direction will extend a distance approximately equal to five times the local foam thickness. The generalized plane strain analyses will use a nearly uniform mesh of square elements having an edge dimension roughly equal to half the substrate thickness. The three-dimensional finite element models will include a square region in both the longitudinal and circumferential directions with several solid elements through-the-thickness of the foam – 8-node brick elements are acceptable, 20-node brick elements are preferred to better represent the local bending response. The spatial discretization should result in well-formed, nearly uniform hexahedral elements.

3.3. Rationale

The rationale for these analyses is primarily to assess the basic assumptions embodied within the TPSMOM analysis tool, to verify underlying assumptions, and to identify limitations. Detailed finite element analyses are planned to verify boundary conditions, structural response type, and local stress state determination. These analytical models are for the acreage ET TPS foam system and are not intended for the close-out regions.

4. Procedure for Mathematical Model Verification
    4.1. Verify Geometry
    4.2. Verify Material Model
    4.3. Verify Coordinate Systems
    4.4. Verify Mechanical Loading
    4.5. Verify Thermal Loading
    4.6. Verify System of Units
    4.7. Verify Boundary Conditions and Constraints
    4.8. Verify Spatial Approximation
    4.9. Verify Solution Process and Procedures
    4.10. Benchmark Problems and Comparison with Classical Theory

5. Procedure Mathematical Model Validation
    5.1. Validate Idealization Assumptions
    5.2. Validate Material Modeling Assumptions
    5.3. Validate Interface Conditions
    5.4. Validate Connection Modeling Assumptions
    5.5. Validate Contact Modeling Assumptions
    5.6. Validate Generalized Imperfection Treatment
    5.7. Configuration Management Strategy

6. Numerical Results and Discussion
    6.1. Baseline Configuration with Different Loadings
    6.2. Parametric Studies
    6.3. Influence of Uncertainties
    6.4. Identification of Potential Risks and Limitations

7. Conclusions and Recommendations
    7.1. Conclusions
    7.2. Recommendations

8. References

REPORT DOCUMENTATION PAGE	*Form Approved* *OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)* 01- 08 - 2004	**2. REPORT TYPE** Technical Memorandum	**3. DATES COVERED** *(From - To)*

4. TITLE AND SUBTITLE

Assessment of Technologies for the Space Shuttle External Tank Thermal Protection System and Recommendations for Technology Improvement

Part 2: Structural Analysis Technologies and Modeling Practices

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Knight, Norman F., Jr.; Nemeth, Michael P.; and Hilburger, Mark W.

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER

23-794-40-4M

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

NASA Langley Research Center
Hampton, VA 23681-2199

8. PERFORMING ORGANIZATION REPORT NUMBER

L-19044

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSOR/MONITOR'S ACRONYM(S)

NASA

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

NASA/TM-2004-213256

12. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 27
Availability: NASA CASI (301) 621-0390 Distribution: Standard

13. SUPPLEMENTARY NOTES

Nemeth and Hilburger: Langley Research Center; Knight: General Dynamics.
An electronic version can be found at http://techreports.larc.nasa.gov/ltrs/ or http://ntrs.nasa.gov

14. ABSTRACT

A technology review and assessment of modeling and analysis efforts underway in support of a safe return to flight of the thermal protection system (TPS) for the Space Shuttle external tank (ET) are summarized. This review and assessment effort focuses on the structural modeling and analysis practices employed for ET TPS foam design and analysis and on identifying analysis capabilities needed in the short-term and long-term. The current understanding of the relationship between complex flight environments and ET TPS foam failure modes are reviewed as they relate to modeling and analysis. A literature review on modeling and analysis of TPS foam material systems is also presented. Finally, a review of modeling and analysis tools employed in the Space Shuttle Program is presented for the ET TPS acreage and close-out foam regions. This review includes existing simplified engineering analysis tools are well as finite element analysis procedures.

15. SUBJECT TERMS

External tank foam; Space Shuttle; Thermal protection; Foam; Structural analysis verification and validation; Acceptance criteria

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			STI Help Desk (email: help@sti.nasa.gov)
					19b. TELEPHONE NUMBER *(Include area code)*
U	U	U	UU	67	(301) 621-0390

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18