This FEA (Finite Element Analysis) Guide for Aerospace provides design engineers, analyst and managers with solution summaries, articles, case studies, and white papers on the application of Finite Element Analysis (FEA) for the development of aircraft, spacecraft, and deployable space structures such as satellites. The guide was assembled by engineers with over 20 years of experience of providing FEA software solutions, consulting, training and support to leading aerospace companies.
AIRCRAFT
SECTION 1
5 Aircraft Modeling
6 Aircraft Structures
8 Case Study: Cessna Business Jets
9 Aircraft Engines
10 Aircraft Mechanisms
11 Aircraft Interiors
12 Bird Strike Simulations
13 Unmanned Vehicles
14 Article: Cirrus Aircraft Composite Fuselage
15 Case Study: Composites Concentric Layups

SPACECRAFT
SECTION 2
19 Spacecraft Modeling
21 Case Study: WhiteKnightTwo and SpaceShipTwo
22 Article: Orion Re-entry System
25 Resource: Design of Fiber Reinforced Foam Sandwich Panels for Large Ares V Structural Applications

SPACE STRUCTURES
SECTION 3
28 Large Deployable Space Structures
29 Case Study: L’Garde, Inc. - NASA Star Antenna
31 Resource: Finite Element Analysis and Test Correlation of a 10-Meter Inflation-deployed Solar Sail
33 Resource: CoNNeCT Project Antenna

COMPOSITES
SECTION 4
36 Composite Structures
37 Composite Guide
42 Resource: Realistic Simulation of Thick Composite Bolted Joints
43 Resource: Cohesive Zone Modeling of Composite Bonded Joints
### NEi Nastran Basic

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NE-L1</td>
<td>Linear Statics</td>
</tr>
<tr>
<td>NE-L2</td>
<td>Normal Modes</td>
</tr>
<tr>
<td>NE-L3</td>
<td>Transient Response</td>
</tr>
<tr>
<td>NE-L4</td>
<td>Nonlinear Static and Transient Response</td>
</tr>
<tr>
<td>NE-L5</td>
<td>Nonlinear Steady-State Heat Transfer</td>
</tr>
</tbody>
</table>

### NEi Nastran Expert*

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NE-L1</td>
<td>Linear Steady-State Heat Transfer</td>
</tr>
<tr>
<td>NE-L2</td>
<td>Buckling</td>
</tr>
<tr>
<td>NE-L3</td>
<td>Frequency Response</td>
</tr>
<tr>
<td>NE-L4</td>
<td>Automated Impact Analysis (AAI™) and Drop Test</td>
</tr>
<tr>
<td>NE-L5</td>
<td>Nonlinear Transient Heat Transfer</td>
</tr>
</tbody>
</table>

*NEi Nastran Expert also contains the analysis features found in the NEi Nastran Basic package.

---

### SOLUTIONS

#### SECTION 5

44  Product Overview
45  Special Topics: Durability, Fatigue and Fracture Optimization
46  Additional Resources

NEi Software is a leading innovator in CAE developing NEi Nastran Finite Element Analysis (FEA), engineering simulation, and virtual test software solutions for over 20 years. Engineers gain insight with digital prototypes, images, contour plots, graphs, and animations of linear and nonlinear structural stress, deformation, dynamics, vibration, impact, heat transfer, and fluid dynamic (CFD) simulations. Aerospace, automotive, maritime, oil and gas, medical, and consumer industry case studies along with white papers, online training, and demonstration videos can be found at http://www.nenastran.com/.
For over 20 years, NEi Software has provided a portfolio of modeling, analysis and simulation software to aerospace engineers involved in the design of a wide spectrum of military, commercial, and private aircraft products. Engineers use these tools for full aircraft analysis covering structures, engines, flight controls, landing gear, stores, and interiors. These solutions include linear and nonlinear analysis, implicit and explicit code, structural, dynamic, thermal, fatigue, motion, and impact simulations.

The case studies and articles presented in this section provides the reader with examples of the capabilities and benefits that simulation injects into all product categories and to every tier of the supply chain for this industry.
Aircraft Modeling

• Structures
• Engines
• Mechanisms
• Interiors
• Bird Strike Simulations

To meet the business challenges of today, aircraft manufacturers look to expand the performance limits of their structures to enable lighter and more capable aircraft which can result in considerable cost savings for the manufacturer and the end-operator.

Simulating how real-life events will impact these designs has proven to be a key asset in these efforts. This technology enables engineers to explore different designs and materials to arrive at an optimal design by reliably predicting when these designs will fail to meet the required safety margins.

Each of the following section summarizes how this technology is used to optimize the design of structures, engines, mechanisms, and interiors. This section also explores how FEA can be used to predict impact events such as bird strikes.
Aircraft Structures

- Fuselage
- Wing and Skin Panels
- Critical Joints
- Materials

Analysis challenges for the aircraft structure not only include the fuselage, wing and skin panels but also the challenges related to the critical joints of the structures and the materials used to optimize the weight of the aircraft.

NEi Nastran solutions, combined with Femap’s pre- and post-processing capabilities, enable efficient and accurate modeling and analysis of aircraft structures, from global FEMs (finite element models) of the full aircraft to local FEMs of critical components. Load interpolation tools facilitate global-local modeling and reliable transfer of loads data to the stress analysts.

Engineers can explore the results through visualization and interpretation tools that include: contour plots, X-Y plotting, shear flow, free body plots, loads comparisons, streamlines, vector plots, and cross section plots.
As the use of composite materials increases for aircraft structures, engineers look to FEA to explore the progressive damage to these composite structures including ply failures and debond failures. Engineers can predict these failures through Progressive Ply Failure Analysis (PPFA) and MCT.

The strength and damage tolerance of all types of critical joints are examined by the engineer through the analysis of inter-rivet buckling, bolted joints, adhesively bonded joints, welds, and pins. The analysis of metallic and composite panel linear buckling is routinely performed for all aerospace programs but nonlinear buckling with post-buckled residual strength needs to be also considered for extreme ultimate loads.

The next generation of capabilities improve the productivity of engineers designing and analyzing aircraft structures. For example, consider the product design time-saving benefits that are achieved with features such as:

- Tension-only Quad (TOQ) shell elements to model load redistribution resulting from post-buckled panels
- The NEi Editor which enables the quick editing and customization of Nastran decks,
- Composite failure initiation and progression methodologies
- Parameter Design Optimization

Read the case studies and article in this section to discover how Cessna and Cirrus Aircraft used FEA in their product design to improve structure design and the analysis process in areas like the fuselage, wing and skin panels, and critical joints.
CASE STUDY
Aerospace: Cessna Aircraft Company - Citation Mustang

Cessna Aircraft Company (www.cessna.com) of Wichita, Kansas used NEi Nastran to perform finite element analysis on the new Citation CJ3 and Citation Mustang business jets. Customer driven enhancements like the Tension-Only Quad Element allowed Cessna to avoid redundant models and repetitive work function resulting in reduced analysis cycle time. NEi Nastran delivered advanced finite element analysis capabilities, complete access to legacy data, and superior technical support to Cessna’s line engineers, while offering a significant reduction in life-cycle cost for the Citation Mustang and CJ3.
Aircraft Engines

Engineers evaluate combinations of severe structural and thermal loads, coupling thermal to structural analysis and temperature dependent material properties to ensure the structural integrity of the engine design.

Aeroengine components involve numerous analysis types and features which include:

- Rotor dynamics
- Vibration
- Creep
- High temperature
- Wear
- Composite fatigue and damage tolerance
- Cyclic symmetry
- Complex contact conditions

NEi Software’s interpolation and extrapolation capabilities transfers environmental loads and conditions to the simulation model from a previous analysis or external test data.

The types of environmental loads and conditions include 3D maps of:

- Temperatures
- Pressures
- Forces
- Moments
- Displacements
Aircraft Mechanisms

• Landing Gears
• Wing Flaps
• Cargo Doors

NEi Software solutions enables engineers to explore complex system such as landing gears, wing flaps and cargo doors through a single model. Engineers can explore complex events in both extremely short durations and extremely long durations.

Engineers can simulate realistic motion and mechanism simulations through:
• Coupled kinematic-stress analysis
• Durability analysis
• Transient response
• Coupled thermal-structural analysis
• Steady-state response in frequency domain
• Heat transfer with thermal contact resistance

Available results for flexible bodies and rigid bodies include:
• Motion
• Velocities
• Accelerations
• Joint forces
• System eigenfrequencies/eigenmodes
• Animations
• Stresses, strains, and fatigue life/damage plots
Aircraft Interiors

Passenger safety, comfort, and convenience are the concerns of designers and engineers creating seating, lighting, storage, entertainment, and the dozens of other elements that go into next generation aircraft environments. As with other aircraft modeling, they must do this while meeting a series of regulations, as well as design objectives for parameters like weight, strength, durability, noise, and energy efficiency.

The ability to model a range of raw materials, titanium, aluminum skin and castings, laser-sintered parts, fiber glass and carbon fiber composites, honeycomb cores, and adhesives is a basic requirement.

Similarly, comprehensive FEA tools prove essential to the exploration of design alternatives, trade-off studies and optimization exercises, and generating data for meeting certification requirements.
Bird Strike Simulations

Bird strike analysis requires specific simulation technologies to ensure safety and survivability under this type of severe event which include:

- Seamless transition from nonlinear implicit to nonlinear explicit analysis
- Eroding contact surfaces
- Material deletion and element deletion under material-specific failure criteria
- Test correlated bird models
- Damage propagation and perforation

NEi Software’s explicit solution offers best-in-class composite modeling capabilities which enable the engineer to examine the constituent-based failure progression for unidirectional and fabric/weave composite materials. Highly optimized parallel explicit solvers guarantee fast solve times for large models under impact and crash scenarios.
Unmanned Aerial Vehicles

The design and development of Unmanned Aerial Vehicles (UAVs) is a vibrant and challenging sector of aerospace. Absence of a pilot and all the safety, ergonomic and physiological requirements associated with that means a long list of parameters are subject to new considerations such as size, structure, configuration, launch methods, propulsion, mission objectives, and flight time. Design objectives for payload mission capability and flight time reinforces the need for simulation tools that can explore material options and design optimization strategies.

Activity in the UAV sector has already led to a breadth of developments including launch methods and vehicle scale ranges from large structures to micro vehicles. Flight time of many months to years is an area of investigation and development as well.

The degree to which simulation is employed can be inferred by looking at a sample of some of the design components that have emerged as solutions in various UAVs. For example, consider the analysis requirements for constant chord rectangular wings, propulsive wing where fan suction prevents flow separation on the trailing edge providing 10x payload increase, pusher propellers, flying wing, solar-electric power, inverted-vee tail, modular disassembly for quick worldwide transportation, “flat-spin” or “falling leaf” landing capability, and break-apart as opposed to breakable structures for survivability offering quick re-assembly and return to service.

UAVs are coming into their own because of the confluence of several key technologies including engineering analysis and simulation software. These technologies are critical to the success of these programs since they enable engineers to quickly find the right balance across many conflicting requirements.
Cirrus Aircraft’s SR22 second-generation design improves functionality and enables faster processing.

Quick, name the best-selling piston-engine airplane for the past seven years running. If you’re an aeronautics aficionado, you likely already know it’s the all-composite-airframed Cirrus SR22. With 3,483 units sold to date, the SR22’s tremendous popularity and market success have allowed the company to reinvest in and improve its basic design and tooling through the years, with the dual goals of improving performance and manufacturability.

Cirrus Aircraft (Duluth, Minn.) was founded in 1984 by Dale and Alan Klapmeier and several associates as a means to market their composite kit plane. While that first kit aircraft, the VK-30, never gained wide acceptance (only a few still exist), the Klapmeiers built on its fundamentals and manufactured a more conventional all-composite, four-passenger, single-engine aircraft, the SR20. Introduced in 1994, it was certified by the U.S. Federal Aviation Admin. (FAA) in 1998. In 2000, the company expanded the line during a period of general aviation growth, fielding the FAA-certified SR22, with a larger engine and greater flying range.

Thanks to robust sales, the company has been able to continuously improve part design, materials and processing, resulting in an updated, second-generation SR22 over the the past several years, says Cirrus airframe engineer Patrick Bergen. “Our roots are home-built aircraft, and at the beginning that meant wet layup on low-temperature tooling. What we have really done, over time, is design a better combination of material and process.”

The second generation

The fuselage skins were a target for the company’s redesign decisions. In their earliest form, all Cirrus fuselages were produced in two halves, left and right, via wet layup. The fuselages included the vertical tail fin. According to Bergen, the left/right split simplified both tool and part construction: “The lengthwise split puts the bond joint at the middle of the windshield, which is not great, but integrates the vertical stabilizer into the skin molds, which simplified tooling.” A top/bottom split would have put the bond joint through the door openings, significantly complicating door details.

For the SR22, each half is 240 inches/6m long, from firewall to tail, and 50 inches/1.27m tall, roof to floor. The original design of the skin parts was accomplished with ProEngineer CAD software from Parametric Technology Corp. (PTC, Needham, Mass.) and finite element analysis (FEA) modeling was accomplished with FEMAP and Nastran software supplied by NEi Software Inc. (Westminster, Calif.). The FEA model determined the part thickness and laminate structure necessary to resist the various load cases, including landing loads and towing scenarios. The SR22 is certified for a 3 G vertical acceleration and a 3 G vertical inverted impact case, as well as typical FAA crash case requirements for seat attachments. In addition, the aircraft had to meet a unique load case in which the passenger cabin must remain intact in the event of deployment of, and landing suspended from, the plane’s emergency parachute. (Cirrus is the only aircraft OEM that builds to FAR Part 23 requirements that also, like many builders of smaller planes, offers a built-in safety parachute, which it calls the Cirrus Airframe Parachute System. The SR-22 is, in fact, the heaviest plane so equipped.)

Early on, however, the company made the decision to abandon wet layup and fabricate the SR20 and SR22 with prepreg for the sake of greater part consistency, notes Bergen. But without an autoclave, the material had to be low-temperature-curable at ambient pressure and still deliver a good surface finish. The vast majority of commercially available prepregs at that time were fully infused through the thickness, and optimized for autoclave cure. Cirrus engineers realized that a prepreg with resin on only one side could enable high-quality laminates with oven cure and vacuum pressure, because air could escape through the dry side, resulting in much lower void content. Bergen says the company initially sourced 3M (St. Paul, Minn.) for the one-sided fiberglass/epoxy prepreg, part of a product line that was later sold to Cytec Engineered Materials, Inc. (Tempe, Ariz.). Subsequently, this semipreg concept was adopted by many material suppliers and fabricators (see “Learn More,” p. 48). Cirrus qualified a second material from TenCate Advanced Composites USA (Morgan Hill, Calif.) as an alternative prepreg supply. A new automated ply cutter supplied by Autometrix Precision Cutting Systems, Inc. (Grass Valley, Calif.) was installed at the same time to cut and kit the prepreg, which
significant reduced material waste and helped optimize material throughput.

Modeling results showed that a sandwich of plain-weave fiberglass/epoxy prepreg facesheets with polyvinyl chloride (PVC) thermoplastic core could meet the design requirements, says Bergen. The skin thickness ranged from 2/core/1 in thinner areas to 25 plies thick in areas where the fuselage bears high point loads. From the beginning, Cirrus preferred foam core to honeycomb because PVC foam was available at lower cost, offered easier handling properties and required no edge treatments. Further, it could be thermo-formed to shape to fit the tool.

During layup, unidirectional fiberglass tapes were added to structurally reinforce the skins around the seating area, essentially forming a stiffer “roll cage” in that part of the skins, a requirement of the parachute landing load case. An aluminum mesh supplied by Dexmet Corp. (Naugatuck, Conn.) was placed in the outer skin for lightning protection.

As sales of the SR22 soared, capital became available for a retooling ef-
Sandwich-to-solid transition

This view of the layup of the right-side fuselage skin shows the inner skin and core inserts (orange color). Edges of the core are beveled, to allow a transition to solid laminate in uncored areas, where the loads are expected to be greater.

Designing-in assembly simplicity

In the layup on the left-side fuselage skin tool, the multistepped edge design around the baggage door opening (about two-thirds of the way back) allowed the use of a simpler door hinge and hardware.

Integrating safety

Visible in this cured but unpainted fuselage skin is the parachute strap, which is cocured with the skin laminate.

fort and, Bergen explains, “Eventually, we understood that because we were already committed to spending money on new tools, the incremental cost of making part design changes was small. In the end, we changed every composite part and almost every metal part in the fuselage.” Design engineers and manufacturing engineers worked together to revisit all of the fuselage skins’ details, to effectively create new parts along with new tools.

Finding an optimal design “balance”

“Most of the changes we made would never be noticed by the customer, but they improved assembly time and quality of the airplane,” points out Bergen. The redesign eliminated one ply in thinner areas of the laminate, and simplified the interfaces between the core inserts and the monolithic laminate. For instance, the first generation layup had foam core butted against some of the offsets in the mold, particularly around the baggage compartment door. When everything fit perfectly, “it was a good structural detail and laminate quality was high,” he says. However, any mismatch due to slippage of the core led to laminate wrinkles and rework. Thus, in the second-generation or what Cirrus calls the “G2” fuselage, the new layup was designed with the core inserts tapered down to solid laminate around openings in the fuselage parts, which caused a slight weight penalty because of less efficient load paths, but allowed faster layup and resulted in far fewer laminate issues.

Another aspect of the redesign, also involving the baggage door opening, was a new, multi-stepped offset in the fuselage skin. Although the new offset was more complicated, it allowed Cirrus to attach the door with a much simpler and lighter-weight hinge, and eliminate the first-generation, heavy gooseneck hinges and numerous metallic fasteners. “The deeper pocket created by the design change allowed the forward edge of the baggage door to dive under, and not hit, the fuselage when opened,” says Bergen.

Part consolidation was another benefit. Obviously a key component of the aircraft, the CAPS parachute straps run from the firewall along the side of the fuselage up to the parachute cover near the vertical tail. Prior to the redesign, the flat, braided aramid fiber straps with metal fittings — 1.25 inches/31.75 mm wide, 0.125 inch/3.18 mm thick, and approximately 200 inches/5m in length — were secondarily bonded to the fuselage skins after cure and assembly. In the G2 fuselage, the straps have been incorporated into the fuselage skins as a cocured detail, says Bergen. “The straps are the first down into the mold during fabrication, and elimination of the bonding step has saved a lot of time.” Bergen is quick to point out that the company didn’t want too much part consolidation, so opted not to cocure parts like the cabin floors or longerons into the fuselage skins: “I believe it’s a false economy to consolidate too many parts into one component. Too much three-dimensional structure can cause wrinkling of the laminate under consolidation,” he explains. “We did what made sense and gave us confidence, without making the part too complicated.”

In the end, the redesigned, second-generation SR22 skin saved a total of approximately 23 lb/10.4 kg over the previous version, says Bergen, including metal hardware and fasteners. Fabrication efficiency was greatly improved: While 24 technicians on three shifts could produce 10 first-generation skin halves per week, six technicians working two shifts can produce 16 G2 skin halves per week, with fewer subassembly steps. At full production, the company now can produce about four planes per day. Bergen concludes that the second-generation SR22 was ultimately a compromise: “If you reduce material cost, you add weight, if you want to reduce weight, you add cost — those are realities that had to be balanced. Our structural engineers worked closely with our manufacturing engineers to produce a coordinated redesign effort — the result has been better parts at an optimized cost.”
Christos Kassapoglou, Composites Analysis, Greece, performed studies to improve the performance of rectangular composite panels under compression.

The use of composites has led to significant decreases in weight in various aerospace applications. In particular, optimum design of rectangular panels where the layup and material(s) are selected so that the weight is minimized under a given set of loads has led to very efficient designs for fuselage and wing skin applications. As the performance requirements become more stringent, additional weight savings beyond these optimized designs are necessary. This requires the use of innovative concepts that open up more possibilities. One such concept is the use of multiple concentric layups in the same rectangular panel. This added tailoring leads to further improvements in performance.

In order to further reduce the weight of such skin panels it is necessary to have reliable and accurate analysis methods especially when the panel is under compression, which, for most applications, is more critical than other loadings. A study was undertaken in order to develop such an analysis method. NEi Nastran was used to complement other analytical methods in order to examine their accuracy and to help explain the meaning of some of the results, in particular the buckling mode shape.

The first step was to understand the performance of panels with two concentric layups under compression. Emphasis was placed on determining the buckling load and mode as well as the stress distribution prior to buckling throughout the panel. The design of such panels requires that no stress exceed the allowable stress for the material prior to buckling anywhere in the panel and that the buckling load is no lower than the applied load. Using NEi Nastran composites capabilities and the buckling analysis option led to the determination of the stacking sequences of the two concentric layups and the dimensions of the center layup so that the weight of the entire panel equals the weight of a panel with a single layup but the strength and buckling capability are now improved. Improvements of at least 15% were demonstrated using simple optimizing concepts. This means a 15% increase in load capacity for the same weight or the potential for decreasing the weight by up to 15% for the same applied load. Further improvements are expected when the approach is coupled with a powerful optimization algorithm.
NEi Software provides a portfolio of modeling, analysis and simulation software to aerospace engineers involved in the design of spacecraft products. Engineers use these tools to evaluate strength and stiffness, vibration, as well thermal distribution and junction temperatures for mission-critical electronic components.

The case studies and articles presented in this section provides the reader with examples of the capabilities and benefits that simulation injects into all product categories and to every tier of the supply chain for this industry.

<table>
<thead>
<tr>
<th>Page</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>Spacecraft Modeling</td>
</tr>
<tr>
<td>21</td>
<td>Case Study: WhiteKnightTwo and SpaceShipTwo</td>
</tr>
<tr>
<td>22</td>
<td>Article: Orion Re-entry System</td>
</tr>
<tr>
<td>25</td>
<td>Resource: Design of Fiber Reinforced Foam Sandwich Panels for Large Ares V Structural Applications</td>
</tr>
</tbody>
</table>
NASA’s Commercial Orbital Transportation Services (COTS) program has enabled companies like SpaceX, Sierra Nevada, Blue Origin, Boeing, and Orbital Sciences to deliver crew and cargo to the International Space Station. Virgin Galactic with the development company Scaled Composites, XCOR Aerospace, Sage Cheshire and SpaceDev are all working on versions of spacecraft that could become the foundation of a space tourism industry.

This transition from government to commercial manned space flight and cargo transportation will affect the businesses and entities supplying parts, software, services, and support. The expected result is lower costs, more innovation, and continuing growth in these sectors.

NEi Software has provided software, training and support for a number of these space initiatives and is well positioned to understand the demands of the many sectors of this industry that include the launch vehicle structures, the payload separation and mechanisms, manned and unmanned spacecraft, rocket engines as well as ground support equipment.
Engineers can explore the design of launch vehicle and spacecraft modeling with strength and stiffness analysis covering a wide range of structures including:
- Cylindrical and conical thin-wall shells
- Ring frames
- Bending beams
- Tension and compression members
- Sandwich and composite structures
- Pressure vessels
- Tanks

Dynamic analysis can also be performed for vibration loads to ensure successful missions of commercial spacecraft. A sample of applications includes:
- Sine vibe
- Random vibration jitter
- Random vibration fatigue
- Relative displacement/rotation calculations
- Shock analysis (SRS modal summation and time history)
- Superelements

The thermal distributions and junction temperatures of mission-critical electronic components can also be examined through:
- Steady-state and transient heat transfer
- Conduction
- Convection
- Radiation
- Phase change
- Thermal contact resistance
- Thermal stress
- Temperature-dependent material properties
- Temperature-dependent heat transfer

Engineers can also explore the structural and dynamics analysis of composite structures including honeycomb panels, complex laminations, 3D micro models of joints and failure zones.

NEi Software continues to develop innovative and cutting-edge capabilities for composite implicit and explicit analysis. Users can scrutinize the following failure modes through industry standards, such as Tsai-Wu, and new micromechanics-based failure theories, such as the MultiContinuum Theory:
- Matrix microcracking
- Fiber-matrix splitting
- Fiber failures
- Wrinkling
- Crimping
- Dimpling
- Interlaminar shear
- Interlaminar tension

Fracture mechanics approaches, such as Cohesive Zone Modelling, simulate crack initiation and crack growth in pure mode and mixed mode fractures. The incremental growth of a crack due to environmental loading can be monitored from analysis, and the damage tolerance of the composite structure can be assessed. Both composite in-plane and out-of-plane damage progression can be taken into account simultaneously.

Those interested in learning how Virgin Galactic and NASA use FEA in their product designs are encouraged to read the case study, article and white paper in this section.
CASE STUDY

WhiteKnightTwo and SpaceShipTwo

WhiteKnightTwo (WK2) and SpaceShipTwo are being developed by Scaled Composites, LLC (www.scaled.com) for their customer Virgin Galactic (www.virgingalactic.com) for the purpose of transforming the safety, cost and environmental impact of manned space travel.

The high profile project continually breaks new ground in aerospace engineering and design. The latest example, the large mothership WK2 pictured above, boasts a wingspan of 140 ft and was constructed from the world’s longest single carbon composite aviation component ever manufactured. Driven by a demanding performance specification set by Virgin Galactic, WK2 has a unique heavy lift, high altitude capability and an open architecture inspired design which provides for maximum versatility in the weight, mass and volume of its payload potential. WK2 was designed to maintain a coast to coast range within the US while being able to deliver its space ship payload to an altitude of 50,000 ft. At this height, SpaceShipTwo disconnects from WK2 and continues its climb to the edge of space at 360,000 ft. An all carbon composite vehicle the size of WK2 represents a giant leap for a material technology. Plus, these efforts contribute to the commercial aviation sector’s urgent need for dramatically more fuel efficient aircraft.

Scaled Composites uses NEi Nastran Finite Element Analysis (FEA) software as part of its process for developing innovative and reliable structural composite components. Types of analyses include: buckling, normal modes and nonlinear solutions. Their modeling techniques are extensively validated through test results. Despite receiving offers of free software from NEi Software’s competitors, Scaled Composites specifically chose to purchase NEi Nastran to meet its needs, based on its strong composites platform and excellent technical support. Dan Kreigh, Lead Structural Analyst for Scaled Composites, underscores the value of both these needs, “We walked through many problems where I was constantly reminded of not only how powerful NEi Nastran is but how really basic and crude many of the other FEA product features are.” On technical support and a good working relationship, “I am impressed at your company’s willingness to quickly react to and address our issues. Your good work in your product is already being used to help us design the first commercial man carrying spaceship. In other words, expect us to be bugging you a lot!”
It's all about taking the heat — and the cold. Since the beginning of human space travel, space vehicles have had to insulate their occupants from the extreme heat and cold of space and protect them from the searing temperatures of re-entry into Earth's atmosphere. Prior to the Space Shuttles, with their heat-resistant tiled wings and underbodies that enable conventional winged landings, the U.S. National Aeronautics and Space Admin. (NASA) relied on capsule-type vehicles that plummeted to Earth in reverse, slowed and protected by a robust, one-piece heat shield that not only could endure extreme heat but also could absorb splashdown energy during ocean landings. The largest of these capsules carried three-man crews into Moon orbit for NASA's Apollo program in the 1960s.

In 2006, as the Space Shuttles neared retirement, NASA launched its Constellation program. A team headed by Lockheed Martin Space Systems Co. (Houston, Texas) was assembled to design and build a replacement for the Space Shuttle, dubbed Orion. The new spacecraft would be capable of...
transporting a crew of six to and from the International Space Station, the Moon and, eventually, Mars.

To maximize the interior volume in the Orion crew cabin design, Lockheed Martin and its Orion partners took a page from space history, opting for a classic “blunt body” shape, similar to but quite a bit larger than that used for the Apollo craft. As a result, Orion is equipped with a massive 5m/16.4-ft-diameter heat shield.

HPC recently had an exclusive opportunity to visit Lockheed Martin’s Colorado facility to learn about how the heat shield was designed and fabricated. Although Constellation has been scuttled in anticipation of the Obama Administration’s proposed NASA budget cuts, the Orion capsule, at press time, received a reprieve (see p. 17). The capsule is nearly complete (for related stories, see “Learn More,” p. 64), and its heat shield is an impressive engineering achievement.

Building on tradition
The Orion heat shield is the latest in a legacy of successful composite heat shields designed and built by Lockheed Martin Space Systems and is the largest ever built for manned and unmanned missions. “Orion has allowed us to take lessons learned from our long history with composite heat shields and push the envelope one step further, with new cutting-edge materials and out-of-autoclave technology,” notes Colin Sipe, Orion ground test article thermal protection system design lead. “The result is a lighter part that makes the entire capsule more mass efficient.”

The shield’s design requirements were considerable, explains Sipe. At launch, it has to survive incredible vibration loads without fatigue failure. In space, it must deliver thermal performance sufficient to protect astronauts from the extreme temperature variations of space and withstand impacts of micrometeorite orbital debris (MMOD), which could damage the shield, compromising re-entry. During re-entry, with help from added ablative thermal barriers, the shield must not melt or burn, despite 5,000°F/2,760°C heat. And finally, as the capsule makes its final descent, suspended below the re-entry parachutes, the shield must withstand a 40-ft/sec (12.2 m/sec) controlled crash into the sea.

Sipe and a team, headed by Lockheed Martin’s Orion Thermal Protection System manager Jan Thornton, started with a legacy heat design but were determined to make it better. Previous designs consisted of titanium sheet metal skins over titanium honeycomb. Pro/ENGINEER 3D CAD software from Parametric Technology Corp. (PTC, Needham, Mass.) was used to design the part, and Nastran and FEMAP finite element analysis (FEA) software from NEi Software (Westminster, Calif.) helped determine whether the part could meet the structural design loads. LS-Dyna FEA modeling software from Predictive Engineering Inc. (Portland, Ore.), originally a car crash simulation tool, was used to determine if the dish would handle water-landing loads.

Based on these analyses, a composite heat shield was created, which consists of several parts. Its one-piece, curved dish is a sandwich structure, featuring composite skins, with a titanium honeycomb core supplied by Benecor Inc. (Wichita, Kan.). A mounting ring made from the same sandwich materials is adhesively bonded to the upper edge of the dish’s circumference to form a flat surface for attachment to the Orion capsule’s outer composite “aeroshells” or protective panels, called the “backshell.” Sipe adds that because the re-entry parachutes cause the capsule to descend at an angle, one side of the heat shield hits the water first. To ensure sufficient strength but keep shield weight to a minimum, the design is asymmetrical, with additional plies and a stronger honeycomb on the leading edge of the heat shield. Further, because the capsule will be attached to the service module situated below it within the launch vehicle, six large compression pads of abla-
The heat shield is lowered onto a jig for machining and subsequent placement of the mounting ring and structural clips.

The heat shield is lowered onto a jig for machining and subsequent placement of the mounting ring and structural clips.

Although the heat shield that will fly will feature a 5-harness satin weave prepreg and a 2-inch/50-mm-thick Bene- cor titanium honeycomb core thermoformed to the correct dish curvature, the ground test article described here was made with less-expensive materials to save development costs and time. The team substituted a plain-weave prepreg, and used a more flexible aluminum honeycomb called Flexcore (Hexcel, Dublin, Calif.), with a cell pattern that conforms more easily to compound curvatures.

Streamlining production

For the heat shield’s facesheets, says Sipe, the layup was designed as a rosette, with triangular gores of prepreg placed like the petals of a flower around a circular center section. The fiber architecture and layup strategy were developed with the assistance of FiberSIM software from VISTAGY Inc. (Waltham, Mass.), which took the design data produced in Pro/ENGINEER, Nastran and LS-Dyna and optimized the laminate to avoid wrinkles while using the smallest quantity of material possible.

reports Jerry Brown, the thermal protection team’s manufacturing lead. “FiberSIM also seamlessly integrated with our automated ply cutter,” he adds. The cutter, from Eastman Machine Co. (Buffalo, N.Y.), produced layup kits “in a matter of days. It would have taken our technicians about two weeks to manually cut all the materials for the dish layup.”

The Invar layup tool was fabricated by Coast Composites Inc. (Irvine, Calif.), with a cell pattern that conforms to the correct dish curvature, the phenolic honeycomb then adhesively bonded to the dish’s underside. “About 30 percent of the Avcoat will burn away,” Sipe notes, pointing out, “The good part is that less Avcoat will be needed because of the better temperature performance of the new prepreg.”

During that time, the outer skin was oven cured first, under a vacuum bag, for about 12 hours. A second cure cycle followed placement and bonding of the core, followed by cure of the inner skin in a third oven cycle. The syntactic core fill and adhesive materials, also from TenCate, were compatible with the prepreg. Airtech International Inc. (Huntington Beach, Calif.) provided the bagging consumables.

At HPC press time, the cured 1,000-lb/455-kg dish had been machined and was ready to receive its mounting ring, structural clips and additional thermal protection. Although the ground test article will have only a foam simulator added, Sipe and Thornton report that the actual flight capsule will be treated with Avcoat ablative material from Textron Defense Systems (Wilmington, Mass.), first used on Apollo-era heat shields. Designed to ablate (erode) during re-entry but leave the underlying composite dish undamaged, Avcoat comprises a silica fiber-filled epoxy novolac resin slurry that is injected into the cells of a fiberglass/phenolic honeycomb then adhesively bonded to the dish’s underside. “About 30 percent of the Avcoat will burn away,” Sipe notes, pointing out, “The good part is that less Avcoat will be needed because of the better temperature performance of the new prepreg.”

The completed heat shield is slated to ship this summer to Lockheed Martin’s Michoud Assembly Facility (New Orleans, La.), where it will be installed on the ground-test capsule for vibration, acoustics and water-landing tests.

“We have been able to meet our performance requirements,” Sipe concludes, “with a lighter and more cost-effective composite design, saving considerable weight for the overall system.” Says Lockheed Martin VP and Orion program manager Cleon Lacefield: “We achieved a $10 million cost savings and improved the project schedule by 12 months.”
Aries V

Cutaway view of the Multi Purpose Crew Vehicle. Image credit: NASA.
ABSTRACT

The preliminary design of three major structural components within NASA’s Ares V heavy lift vehicle using a novel fiber reinforced foam composite sandwich panel concept is presented. The Ares V payload shroud, interstage, and core intertank are designed for minimum mass using this panel concept, which consists of integral composite webs separated by structural foam between two composite facesheets. The HyperSizer structural sizing software, in conjunction with NASTRAN finite element analyses, is used. However, since HyperSizer does not currently include a panel concept for fiber reinforced foam, the sizing was performed using two separate approaches. In the first, the panel core is treated as an effective (homogenized) material, whose properties are provided by the vendor. In the second approach, the panel is treated as a blade stiffened sandwich panel, with the mass of the foam added after completion of the panel sizing. Details of the sizing for each of the three Ares V components are given, and it is demonstrated that the two panel sizing approaches are in reasonable agreement for thinner panel designs, but as the panel thickness increases, the blade stiffened sandwich panel approach yields heavier panel designs. This is due to the effects of local buckling, which are not considered in the effective core property approach.

VIEW THIS REPORT

http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20100040417_2010044062.pdf

If you are unable to access the report at the above location, please contact info@neisoftware.com.
NEi Software provides a portfolio of modeling, analysis and simulation software to aerospace engineers involved in the design of space structures such as satellites. Engineers use these tools to explore materials, structural satellite frames and struts, as well as linear and nonlinear dynamics needed for vibration analysis and payload protection.

The case studies and articles presented in this section provide the reader with examples of the capabilities and benefits that simulation injects into all product categories and to every tier of the supply chain for this industry.
Large Deployable Space Structures

- Deployable and Inflatable Space Structures
- Solar Sails
- Reflector Antennas
- Space Radars
- Optical Platforms
- Satellite Bodies

The FAA 2012 Commercial Space Transportation Forecast anticipates an average of 30 commercial satellite launches per year over the next decade. Major satellites are expensive to engineer, build, launch, and operate. Examples of efforts to reduce the cost of space-based assets include in-orbit robotic repair of existing satellites such as the Robotic Refueling Mission (RRM) scheduled for trial this summer.

Other examples to reduce costs include reducing satellite size through new designs. Although much of the technology and experience gained through the launch of larger satellites can be transferred in designing these micro-, nano- and pico-satellites (under 1 kg, 10 kg, 100 kg respectively), engineers are facing new design challenges. Some of these challenges include the miniaturization of electronics as well as the design of smaller launch vehicles.

Engineers within this industry can use many of linear and nonlinear solutions, both implicit and explicit, from NEi Software.
These capabilities have been used by engineers to design deployable and inflatable space structures, solar sails, reflector antennas, space radar, optical platforms and satellite bodies. These innovative space companies are required to evaluate highly complex systems with various nonlinearities all occurring at the same time which include:

- Large displacements and rotations
- Sliding contact with friction
- Tension-only cables
- Shape memory materials
- Postbuckling
- Membrane wrinkling
- Hyperelastic/elastomeric materials
- Localized crippling
- Prestress from pressure

With NEi Software solutions, engineers can run nonlinear analysis statically or in a transient dynamic scenario to combine nonlinear effects with dynamic effects.

Read the case study and white papers in this section to discover how L’Garde and NASA are using this technology for the design and optimization of these large deployable space structures.
CASE STUDY
L’Garde, Inc. - NASA Star Antenna

NEi Nastran was used by L’Garde, Inc. (www.lgarde.com) to carry out the analysis and modeling of the NASA Langley STAR Antenna configuration. The above top right figure shows the original STAR antenna concept. A planar, triangular membrane is stretched via 3 booms. Note the circular RF components positioned accurately on the stretched membrane.

The top left figure shows the Tensioned Tripod concept (loadings shown with red arrows) used in the finite element analysis performed with NEi Nastran. NEi Nastran accurately accounted for both the compressive loads on the booms and the tension loading within the membrane.

NEi Nastran was also used to carry out a prestress modal analysis. This was necessary to determine what modes and natural frequencies the structures would undergo. The first figure below shows the finite element model used while the other three figures show the first few modes of vibration.

The finite element modeling and modal analysis made possible with NEi Nastran was instrumental in creating an accurate, efficient, and cost effective solution.
RESOURCES

Finite Element Analysis and Test Correlation of a 10-Meter Inflation-deployed Solar Sail

ABSTRACT

Under the direction of the NASA In-Space Propulsion Technology Office, the team of L’Garde, NASA Jet Propulsion Laboratory, Ball Aerospace, and NASA Langley Research Center has been developing a scalable solar sail configuration to address NASA’s future space propulsion needs. Prior to a flight experiment of a full-scale solar sail, a comprehensive phased test plan is currently being implemented to advance the technology readiness level of the solar sail design. These tests consist of solar sail component, subsystem, and sub-scale system ground tests that simulate the vacuum and thermal conditions of the space environment. Recently, two solar sail test articles, a 7.4-m beam assembly subsystem test article and a 10-m four-quadrant solar sail system test article, were tested in vacuum conditions with a gravity-offload system to mitigate the effects of gravity. This paper presents the structural analyses simulating the ground tests and the correlation of the analyses with the test results. For programmatic risk reduction, a two-prong analysis approach was undertaken in which two separate teams independently developed computational models of the solar sail test articles using the finite element analysis software packages: NEiNastran and ABAQUS. This paper compares the pre-test and post-test analysis predictions from both software packages with the test data including load-deflection curves from static load tests, and vibration frequencies and mode shapes from vibration tests. The analysis predictions were in reasonable agreement with the test data. Factors that precluded better correlation of the analyses and the tests were uncertainties in the material properties, test conditions, and modeling assumptions used in the analyses.

VIEW THIS REPORT

http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20050175895_2005173050.pdf

If you are unable to access the report at the above location, please contact info@neisoftware.com.
Independent Peer Review of Communications, Navigation, and Networking re-Configurable Testbed (CoNNeCT) Project Antenna Pointing Subsystem (APS) Integrated Gimbal Assembly (IGA) Structural Analysis

November 2010

циальн Glenn Research Center Chief Engineer’s Office requested an independent review of the structural analysis and modeling of the Communications, Navigation, and Networking re-Configurable Testbed (CoNNeCT) Project Antenna Pointing Subsystem (APS) Integrated Gimbal Assembly (IGA) to be conducted by the NASA Engineering and Safety Center (NESC). At this time, the IGA had completed its critical design review (CDR). The assessment was to be a peer review of the NEi-NASTRAN1 model of the APS Antenna, and not a peer review of the design and the analysis that had been completed by the GRC team for CDR. Thus, only a limited amount of information was provided on the structural analysis. However, the NESC team had difficulty separating analysis concerns from modeling issues. The team studied the NASTRAN model, but did not fully investigate how the model was used by the CoNNeCT Project and how the Project was interpreting the results. The team’s findings, observations, and NESC recommendations are contained in this report.

Independent Peer Review of Communications, Navigation, and Networking re-Configurable Testbed (CoNNeCT) Project Antenna Pointing Subsystem (APS) Integrated Gimbal Assembly (IGA) Structural Analysis

VIEW THIS REPORT

http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20100042358_2010045132.pdf

If you are unable to access the report at the above location, please contact info@neissoftware.com.

www.NENastran.com
Figure 6.1-1. APS NASTRAN Model

Figure 6.1-4. Mode Shapes for Modes 8 and 34 – Modes with Most Modal Mass Participation
NEi Software provides innovative and cutting-edge capabilities for composite implicit and explicit analysis. Aerospace engineers can perform structural, dynamics, honeycomb panel, complex lamination and 3D micro models of joints and failure zones analyses.

The case studies in earlier sections of this guide along with the Composite Guide and white papers in this section provide the reader with examples of the capabilities and benefits that simulation injects into products designed with this advanced material.
Composite Structures

• Structural
• Dynamics
• Honeycomb Panels
• Complex Laminations
• 3D Micro Models of Joints & Failure Zones

Composite materials are gaining popularity for space applications as reductions in product weight directly impacts production and fuel costs as well as payload. Composites have long been touted not only for weight but performance advantages in stiffness, fatigue and corrosion resistance. Nonetheless, the material has not been fully exploited to date due to several factors.

These factors include the:
• Learning curve to understand how this material behaves in real-life conditions
• Investment in developing new manufacturing equipment
• Investment in new inspection processes

Of these factors, product design teams are challenged with changing their long-established practices and proven design strategies, based on the familiar isotropic material properties of metals. NEi Software solutions enable engineers to apply a new approach, a new way of thinking about their designs, by considering how to best apply this material’s orthotropic and anisotropic properties.

Readers can learn more about the application of FEA for composite analysis by reading the Composite Guide and white papers included in this section. Case studies and articles are also presented in previous sections that explore this application.
Laminated composite materials have found their way into many military and consumer products. While composites are valued for their cost, strength and weight, some types have less desirable characteristics that come with these benefits. Shortcomings include brittleness, poor environmental resistance and poor temperature resistance. In addition, mechanical properties of composites are often dependent to a large degree on the manufacturing process.

From an analytical point of view, this variability has presented problems. For starters, composite materials vary greatly in properties, so there is no one-size-fits-all token value to use for preliminary designs. For any particular metal, Young’s modulus (stiffness) is relatively constant, regardless of the variety or alloy. This is true of most commonly used metallic materials, and assists greatly in preliminary designs. Unfortunately for the analyst, this is not true of composites, and care must be taken to ensure that an appropriate material model is used, even in preliminary design. The additional variability of strength makes analytical determination of failure difficult as well. This free composite guide is intended to help answer the five most common questions regarding the analysis of composites.
What material data is needed for Finite Element Analysis (FEA) of fiber reinforced composites and how do I obtain this material data?

Unlike material property data for isotropic metals such as steel and aluminum, material property data for composites is usually proprietary data and not readily available in sources such as the Metal Handbook or MIL-HDBK, or what is now MMPDS. Typically, a composite analyst obtains the data from either the manufacturer or proprietary company laboratory testing.

Composite materials will generally have varying stiffness and strength depending on the orientation, so they are typically modeled using orthotropic material properties. Following are some typical FE 2-D Orthotropic material data input for composite lamina:

- Longitudinal Modulus
- Lateral Modulus
- Poisson’s Ratio
- In-Plane Shear Modulus
- Longitudinal Transverse Shear Modulus
- Lateral Transverse Shear Modulus
- Longitudinal Tensile Failure Stress
- Lateral Tensile Failure Stress
- Longitudinal Compressive Failure Stress
- Lateral Compressive Failure Stress
- Shear Failure Stress

The Failure Stresses (shown above) are not required for running a composite FEA. However, they are needed to use First-Ply Failure Theories to output a Failure Index. Some of the Failure Theories incorporated into FEA are Hill, Hoffman, Tsai-Wu, Max Stress, Max Strain, NASA LaRC02, and Puck. These utilize the FE-calculated ply-by-ply stresses and the 2-D Orthotropic Failure Stresses to output a Failure Index value for every element. The convention is: a Failure Index over unity indicates First-Ply failure in that region. First-Ply Failure indicates at least one ply has failed and possibly other plies have failed as well due to a specific loading environment.

After I run a composite Finite Element Analysis, what type of information can I obtain in my results?

By looking at a contour plot of Failure Indices, critical regions (where Failure Index is over 1, predicting First-Ply Failure) can quickly be spotted. The critical plies that the high Failure Indices are coming from can be shown in visual format as well. All the different stress components, such as fiber stress, transverse fiber stress, in-plane shear stress, and the interlaminar shear stresses, can be contoured from each of the plies as well.

The in-plane Failure Indices are an indicator of fiber and matrix damage. To predict debonding or delamination of plies, the interlaminar shear stresses that are output can be compared to the allowable Interlaminar Shear Stresses (ILS) of the bond material between plies.

Certain FEA codes also have the capability to predict sandwich laminate local instability failure modes such as wrinkling, dimpling, and crimping. Usually published empirical equations are used for this.

![Figure 1: Maximum failure index for a simplified carbon fiber bike frame.](image-url)
Can I easily modify my composite layup/stackup (number of plies, fiber orientation angles, ply thickness, etc.)

This is exactly what composite Finite Element Analysis is intended for — fast and easy iterations on the composite layup to see how the composite results change.

From using a layered composite definition (NASTRAN PCOMP), there is no need to alter the original geometry. Most FEA programs utilize a “minispreadsheet” where the analyst can quickly “layup” his composite by selecting predefined materials, entering in fiber orientation angles and ply thickness, and moving/reflecting plies around. After an analyst has run an initial composite FEA, the “minispreadsheet” allows the analyst to quickly change any parameter that was previously set in the layup. Then the analysis can be readily rerun, revealing the new composite results to see if the design work is going in the right direction.

To model ply drop-off and transition regions between different composites, multiple layered composite definitions need to be defined.

What motivates the use of layered 3-D Solid Composite Elements over the more common layered 2-D Shell Composite Elements?

Interlaminar stresses are one of the failure mechanisms uniquely characteristic to composite materials. In Classical Laminate Theory (CLT) and 2-D Shell Finite Elements, plane stress assumption is made and the out-of-plane stresses are assumed to be zero. From composite testing, it has been quickly found that at the free edges of a laminate, the interlaminar shear stress is very high and would cause failure in the bond material between the plies. These interlaminar shear stresses arise to balance the moment resulting from different in-plane stresses in the different plies. Many FEA codes, such as NASTRAN, utilize a “beam shear” approximation for 2-D Composite Shells to calculate interlaminar shear stresses. Recent developments have “pushed” composite FEA into 3-D Composite Solids – with a layered composite hexahedral element, interlaminar shear stresses can now be directly computed without assuming plane stress condition to assess the failure mode of delamination. Loading actions can also be important and may motivate the need to move away from shell elements.
What if I am concerned about what happens to my composite structure beyond First-Ply Failure, to more efficiently design to Ultimate Loads?

This is where Progressive Ply Failure Analysis (PPFA™) comes into play. With nonlinear progressive ply failure, the FE analyst can assess the nonlinear damage accumulated by composite structures. The loss of stiffness in certain plies and certain directions (fiber and matrix) can be simulated, and the analyst can post-process output such as percentage of plies failed and location of plies failed.

PPFA allows us to answer engineering questions such as:

**Is the failure of the first ply associated with abrupt failure of the component?**

In some cases, the load may be such that the residual load carrying capacity of a part is insufficient to maintain integrity with a damaged laminate. In that case, a Progressive Ply Failure Analysis will show a swift failure of all remaining plies once the first ply has failed. This is useful information for determining margins of safety to incorporate into a design.

**Is the stress re-distributed over adjacent plies, maintaining the load-carrying capability of the part?**

In some cases, failure of a single ply will simply result in the load being taken by other plies in the laminate. In this case, a Progressive Ply Failure Analysis will show that equilibrium is reached after failure of some plies, but that part does not fail catastrophically. Like the abrupt failure case, this information can assist in determining design margins.

**Is load redistributed over adjacent elements?**

It is possible that the partial failure of a section will redistribute the load to other parts of the structure. A progressive failure analysis would show where the loads end up, and can provide valuable insight into the design of the other members in order for them to take the additional load properly. An option may exist to channel the redistributed load away from critical members into ones designed for the additional load.

**How far is First-Ply Failure (FPF) from Last-Ply Failure (LPF)?**

If the last ply fails relatively quickly after the first ply, it may make sense to use a FPF based design approach for the rest of the model, or for the rest of the design cycle. However, this would be hard to verify without at least one progressive failure analysis.

**What is the evolution of the part from FPF to LPF failure?**

The nonlinear nature of progressive failure analysis also makes the progression from FPF to LPF nonlinear. In fact, it may be that a few plies fail initially, but that the remaining plies remain intact for a large portion of the loading cycle, only failing catastrophically at the very limit of the load. This information might be able to provide the basis for a redesign that eliminates the early failure, thus extending the range of the intact laminate considerably. Similarly, if most of the plies fail immediately, and the last ply only fails at the very end, it implies that the initial damage takes away most of the load carrying capacity of the laminate. In this case, a FPF based approach would also work reasonably well.
How do I separately assess fiber failures and matrix failures to gain a better understanding of the failure initiation and failure progression of my composite structure?

The traditional finite element approach to fiber reinforced composites is to smear the properties of the fiber and matrix together to arrive at a homogeneous representation of each layer. The smeared properties can be found from testing or a micro-mechanical approach. This “Black Aluminum” smeared approach provides limited information about exact failure mode and is known to be severely conservative for various loading scenarios. The micromechanics-based MultiContinuum Theory (MCT) overcomes these limitations by separating the matrix and fiber properties, handling each composite constituent differently and allowing failure to progress through a multi-step damage mechanism. After the FEA is run, exact failure mode can be obtained through accessing separate failure index calculations for fibers and epoxy matrices.

MCT is developed for both unidirectional and woven composite materials. By coupling MCT with kinetic theory of fracture, accessing constituent stress/strain will lead to a direct link to composite fatigue life prediction.

How do I analyze for the effects of both delamination initiation and propagation?

Cohesive Zone Modelling (CZM) is a fracture mechanics approach for simulating crack initiation and crack growth in pure mode and mixed mode fractures. The incremental growth of a crack due to environmental loading can be predicted from the analysis, and the damage tolerance of the composite structure can be assessed. Many different delamination scenarios can be analyzed with CZM: delamination in the adhesive/adherend interface of a bonded joint or delamination in the skin/core interface of a large scale honeycomb sandwich structure, for example. CZM is a generally applicable method for nonlinear structural analyses, and can be used in conjunction with progressive ply failure analysis (PPFA) and micromechanics based failure model MCT. With CZM and PPFA used simultaneously, both composite in-plane and out-of-plane damage progression can be taken into account.

Find additional guides, white paper and videos at www.nenastran.com/kb.

NEi Software is a leading innovator and global provider of Nastran Finite Element Analysis (FEA), engineering simulation, and virtual test software. Engineers can learn about linear and nonlinear structural stress, deformation, dynamics, vibration, kinematics, impact, heat transfer and composite analysis by visiting http://www.nenastran.com/kb.
Realistic Simulation of Thick Composite Bolted Joints: A Novel NASTRAN Method

SAMPE 2013 Fall Technical Conference

Allan Hsu
NEi Software
5555 Garden Grove Boulevard
Westminster, CA 92683, USA

ABSTRACT

A computationally efficient method for modeling thick composite bolted joints is presented to aid in aircraft structures analysis. The modeling strategy incorporates the following joint parameters to accurately capture load distributions in composite joints: laminate-to-laminate friction, bolt preload, and bolt-hole clearance. Three dimensional layered solid elements maintain efficiency by using a layered definition like layered shell elements, but provide the unique advantage of directly calculating through-thickness interlaminar stresses. The MultiContinuum Theory (MCT) damage model is utilized as part of the modeling strategy to extract separate failure predictions for the fibers and epoxy matrices throughout the composite joints.

1. Introduction

Efficient and realistic simulation of composite bolted joints is absolutely necessary in fully realizing the benefits of composite materials to build lighter and more capable aircraft. Various techniques for efficient modeling of bolted joints have been investigated in recent publications – these techniques either use layered shell elements to represent the laminates or large numbers of three-dimensional solid elements by explicitly meshing each layer of the laminate. As the focal point of this particular modeling strategy, three-dimensional layered solid elements maintain efficiency by using a layered definition like layered shell elements, but provide the unique advantage of directly calculating through-thickness interlaminar stresses, both through-thickness normal stresses and through-thickness interlaminar shear stresses. As the micromechanics-based failure model, MultiContinuum Theory (MCT) is utilized as part of the modeling strategy to extract separate failure predictions for the fibers and epoxy matrices throughout the composite joints. Using examples, the investigation examines how fiber fracture and matrix damage information can be separately accessed, and how critical fastener loads can be extracted from the finite element model.

Download this white paper at http://www.nenastran.com/composite-white-paper/.
A computationally efficient method for modeling delamination initiation and propagation in composite bonded joints is presented to aid in aircraft structural analysis. The modeling strategy incorporates the following state of the art finite element technologies to accurately capture the complex damage mechanisms in bonded composite joints: cohesive zone modeling (CZM), three-dimensional layered solid elements, and micromechanics-based damage degradation. Simulation results are presented for a Double Cantilevered Beam (DCB) Mode I Fracture Test and a Mixed Mode Skin Stiffener Tension Test, and compared to experimental data from technical publications.

1. Introduction

Analytical assessment of composite bonded joints is a recurring issue in aircraft design today. Various techniques have been applied to this problem with mixed results. Some techniques involve a two-step approach where a conventional finite element stress analysis predicts delamination initiation, and the initial debond results are passed to a fracture mechanics approach for delamination propagation predictions. Other techniques are not able to fully capture all of the critical components of complex damage mechanisms: initial damage resulting from through thickness matrix cracking and branching, matrix cracks driving delaminations to occur, stiffness degradations in fiber and/or matrix directions, and complete delamination across the adhesive bondline. This particular one-step modeling strategy is designed to capture the true failure mechanism of the joint, and incorporates the following state of the art finite element technologies to accurately capture the complex damage mechanisms in bonded composite joints: cohesive zone modeling (CZM), three-dimensional layered solid elements, and micromechanics-based damage degradation. As the micromechanics-based failure model, MultiContinuum Theory (MCT) is utilized as part of the modeling strategy to extract separate failure predictions for the fibers and epoxy matrices throughout the composite joints. Using a Double Cantilevered Beam (DCB) Mode I Fracture Test and a Mixed Mode Skin Stiffener Tension Test, the paper compares critical delamination propagation failure loads predicted by NASTRAN with experimental data from previously published technical publications.

Download this white paper at http://www.nenastran.com/composite-white-paper/.
NEi Software’s software solutions enable efficient and accurate modeling and analysis of aircraft structures, from global FEMs (finite element models) of the full aircraft to local FEMs of critical components such as longerons and fuel floors. Loads interpolation tools facilitate global-local modeling and reliable transfer of loads data to the stress analysts. Results visualization and interpretation tools span the whole spectrum: contour plots, X-Y plotting, shear flow, freebody plots, loads comparisons, streamlines, vector plots, and cross section plots.
From a structural simulation, stress and strain data can also be stored in a results database for use in fatigue analyses at a later point in time.

In proposal phases and initial design phases of aerospace programs, automated design optimization accelerates product development and technical feasibility. The NEi Editor includes parameter optimization for plate thicknesses, beam cross sectional dimensions, composite layups, material properties, as well as other parameters.

The Optimization graphical user interface enables easy management and definition of optimization constraints, variables, and objectives. Using design-space searching algorithms, the Optimization tool efficiently takes monocoque fuselage structures, iterates intelligently on various model scenarios, and minimizes weight.

Special Topics

• Durability, Fatigue and Fracture
• Optimization

Durability and Damage Tolerance (DADT) analysis is crucial for the maintenance and longevity of military and commercial aircraft. NEi Software’s software solutions are ideal for High Cycle Fatigue (HCF) through Stress-Life methods, Low Cycle Fatigue (LCF) through Strain-Life methods, multiaxial fatigue, vibration fatigue, complex load histories, rainflow cycle counting, crack initiation and propagation, and calculation of stress intensity factors. NEi Software’s Fatigue Solutions directly calculates fatigue damage and fatigue life from the NASTRAN model in a single code.
ADDITIONAL RESOURCES

When Analysis Goes Nonlinear  [http://nenastran.com/nonlinear/whitepaper]
Composite Structures Subjected to Blast Loading  [http://www.nenastran.com/compositeblastloading/]
10 Reasons to Upgrade to Femap  [http://nenastran.com/femap/10reasons]
About NEi Software

NEi Software is a world leader in Finite Element Analysis (FEA), engineering simulation, and virtual test software. The core product NEi Nastran is a powerful, industry-proven FEA solver that thousands of companies routinely use to perform linear and nonlinear structural stress, dynamics, and heat transfer analysis. In addition, NEi Software’s portfolio includes products for impact, kinematics, fatigue, acoustics, optimization, aeroelasticity, and Computational Fluid Dynamics (CFD) with support for a full range of materials from composites to hyperelastic rubber. NEi Software covers the different needs of each stage of the product development process, from designers looking for affordable, easy-to-use, CAD-based simulation for validation and trade-off studies to dedicated FE analysts looking for high accuracy, productivity, and real world fidelity. The website features case studies in aerospace, automotive, maritime, military, civil, petroleum, medical, and consumer products with videos, webinars, tutorials, and options for evaluation.

Contact Information

5555 Garden Grove Blvd. Ste 300
Westminster, CA 92683-1886, USA
E-mail: info@neissoftware.com
Phone: +1 (714) 899-1220
Fax: +1 (714) 899-1369
Website: www.NEiSoftware.com