

Dialog boxes in the ESAComp ply specification tool enable users to readily enter various input data.

Plying the Composite Trade

Coupled with technology from ANSYS, Inc., ESAComp software overcomes challenges in designing with composites, enabling engineers to evaluate part designs and better use these versatile materials to their full advantage.

By Harri Katajisto
Componeeing Inc.
Helsinki, Finland

Carbon-fiber reinforced plastics and other composite materials are used in a wide range of applications because of their high strength-to-weight ratios. High-performance composites made of continuous fibers bound with thermoset resins can be used in making extremely efficient structures, and laminated composites are well suited for lightweight parts with complex surface contours.

Composites present some complex challenges in utilizing these materials to their best advantage, however. Material properties are anisotropic — that is, they are directionally dependent on the orientation of the reinforcing fibers. Differences in thermal expansion of the matrix and reinforcing materials cause residual stresses, and asymmetric structures especially can yield unexpected responses to temperature variations. Moreover, sandwich structures exhibit complex behavior because of large differences in strength and stiffness between layers.

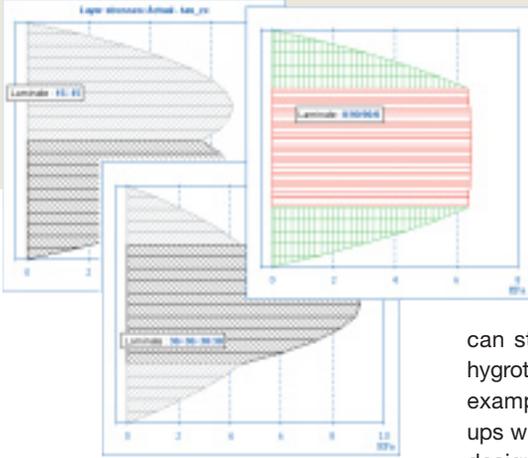
When designing with laminated composites, engineers must take into account these and many other considerations in establishing important design variables, including selection of material types, layer orientation and thickness, number of layers, and stacking sequence. Compounding the difficulty, complete material property data cannot always be found from the supplier data sheets.

Composite Analysis and Design

High-performance composites are used extensively in the aerospace industry, where engineers rely on in-house tools developed specifically for composite analysis. These programs require considerable resources to develop and maintain, however. Engineers need extensive training to understand the specialized command-based interfaces and numerical outputs. In addition, users often have to transfer data manually between multiple programs for modeling and analyzing components.

Concerned about the inefficiency and lack of consistency between the wide range of in-house codes used in the aerospace industry, the European Space Agency (ESA) initiated a project in the early 1990s to standardize the analysis approach with a single software platform combining various tools under a unified user interface. ESA, with headquarters in France and consisting of 17 member states, is in charge of shaping the development of Europe's space capability and ensuring that investment in space continues to deliver benefits to the citizens of Europe. By coordinating member resources, the agency can undertake programs and activities far beyond the scope of any single European country. ESA also works closely with space organizations outside Europe.

Development work for the composite project was conducted by Helsinki University of Technology in Finland, and the first version of ESAComp software was released in 1998. Development responsibility later



Layer charts indicate the effect of layer orientations on interlaminar shear stress distribution in a short beam test sample.

was transferred to the spin-off Finland-based company Compoengineering Inc., which now distributes and supports the software. Although the software originated in the aerospace industry, it has been developed as a general tool for engineers in other applications designing with high-performance composites, including automotive, marine, construction, machinery, rail transportation, sports and wind energy.

The software has analysis and design capabilities for solid-sandwich laminates and micromechanical analyses. It further includes analysis tools for structural elements: plates, stiffened panels, beams and columns, and bonded and mechanical joints. ESAComp focuses on the conceptual and preliminary design of composite structures as well as detailed product evaluation using ANSYS Mechanical and other analysis software. Engineers

can study constitutive relations and hydrothermal behavior of laminates, for example, and compare laminate lay-ups with respect to strength and other design requirements. Input checks help guarantee that analyses are not performed with inadequate data.

Users can run preliminary design checks to ensure that columns do not buckle, plates withstand applied loads without deflecting excessively, pressure vessels carry specified internal pressures, joint configurations are efficient for load transfer, holes in plates do not cause severe stress concentrations, and scatter in material properties does not cause unexpected problems. The initial solution obtained gives a starting point and benchmark when going to finite element analysis (FEA) of the full structure, after which post-processing of the results helps survey the numerous failure mechanism possibilities.

The software includes a material database of fibers and matrix materials, adhesives, sandwich core materials, and reinforced material systems from commercial suppliers.

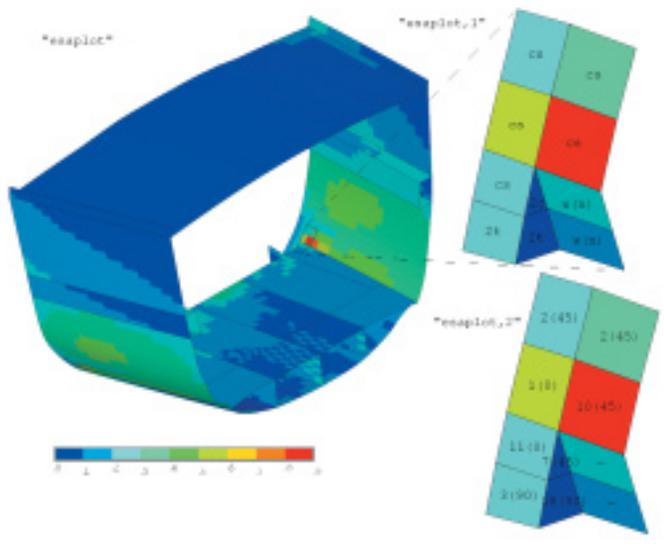
A ply specification tool in ESA-Comp is particularly valuable in setting up the input data for various material configurations, such as a cured fiber-matrix system or a honeycomb core material. Since ply behavior is typically between isotropic and fully anisotropic, the ply specification tool utilizes material symmetry rules to help in defining the data. Ply data also can be derived from fiber and matrix data with micromechanics analyses.

With the laminate lay-up tool, laminates can be created and edited efficiently. The user has a wide range of options for performing analyses as well as selecting and combining result data. For example, several laminates, laminate orientations or failure criteria



ANSYS, Inc. software and ESAComp were instrumental in the design of the 75-foot Wally-class racing yacht.

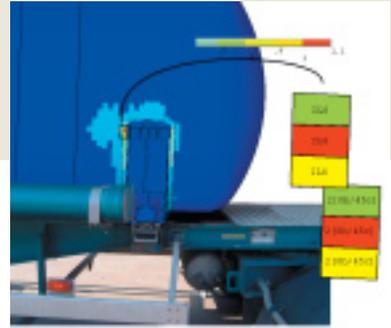
Image courtesy Johannes Schlieben, University of Applied Sciences of Northwestern Switzerland.



The esaplot viewing utility gives quick insight into the overall safety performance of the yacht design. For each composite element, a safety factor determined by the most critical layer of the section is given. Options provide detailed data on the laminate failure mode and the most critical layer, with this information overlaid on each element. Here, for example, cs indicates core shear failure, and w(n) denotes wrinkling of the face sheet. The second option characterizes the most critical layer: the stacking number and the orientation.



A liquid-hauling tank and associated structures of the truck were analyzed. The truck's tank was made of filament-wound composites and sandwich structures.



Analysis determined criticality of the interlaminar shear strength for the trailer tank structure during deceleration. Margin to safety is indicated with contours, and results of the failure analysis are overlaid on the elements.

can be selected for different types of analyses. The results display options include numeric tables, line and bar charts, failure envelopes, and contour plots.

Integration with Software from ANSYS, Inc.

ESAComp is fully integrated with ANSYS Mechanical software. ESAComp FE export supports ANSYS pre-processing. Also, the program can be launched from the ANSYS interface to perform detailed stress analysis and post-processing. ANSYS Mechanical software allows defining FE model input files in text format using specific commands, which is, in many cases, the best way to set up models; the ESAComp FE export capability fits in this scheme. The ANSYS Workbench platform supports these text format laminate definitions as well. For each part in the model tree, the user can give ANSYS commands through ANSYS Workbench command objects. Laminates can be defined with ESAComp FE export, and the definitions override the default material definitions.

Currently, the best way to simulate complex composites structures is to import computer-aided design (CAD) geometry in ANSYS Workbench as surface bodies and use enhanced contact features, automated meshing and environment commands. Then, open the simulation model in ANSYS Mechanical software and read in all

laminate definitions from an ESAComp FE export file. When the geometry is imported as surfaces, ANSYS Workbench automatically uses shell 181 elements. After elements have been updated to correspond to the correct laminate definitions, the model is solved and post-processed.

Integration of ESAComp post-processing with ANSYS has been realized with the versatile ANSYS Parametric Design Language (APDL) and is used through two commands: esapost and esaplot. The most relevant data can be combined in a single ANSYS contour plot showing safety margins for the most critical failure mode, including layer failure, interlaminar shear, or sandwich core shear and wrinkling. Text labels on elements provide additional information on the failure modes or critical layers. Through this procedure, the user quickly identifies design-driving areas, since all relevant failure modes are considered automatically and clearly displayed.

ANSYS, Inc. technology and ESAComp are complementary tools used routinely in developing products made of high-performance composites. The technologies were instrumental in the design of the 75-foot Wally-class racing yacht, for example, which features a unique canting keel for balancing the moments produced by the sails. In this application, the weight of the boat was a dominant design

driver; simulation tools were critical in optimizing the lay-ups and certifying the laminates of heavily loaded components, such as the chain plates and the junction between the keel box floor and the hull.

Another application involved a design project for a truck with a liquid-hauling tank made of filament-wound composites and sandwich structures. Advanced contact features and automatic meshing capabilities in the ANSYS Workbench environment were used to transform the CAD geometry of the tank support structure to the FE model. ANSYS parametric modeling features and interfacing capabilities with ESAComp were further used to optimize the design. Finally, ANSYS Mechanical software was used for validating the design against the certification authority's requirements. Processing indicated how interlaminar shear (ILS) strength of the laminate structure is a dominant design factor in the discontinuity location while the truck is decelerating.

The ANSYS ESAComp post-processing utility indicates to designers the weakest point of the structure, the weakest ply in that location and the most likely mechanism of ply failure. This information gives users valuable insight for making informed decisions on refining the design of the structure. ■