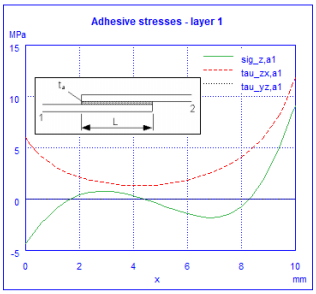
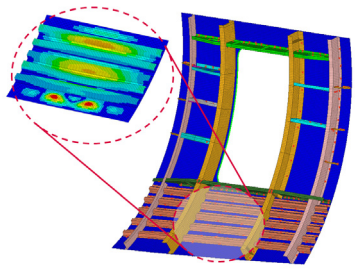
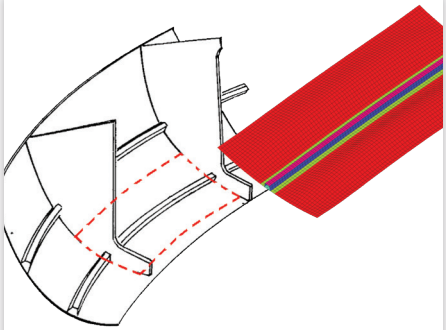
Altair ESAComp™

Analysis and Design of Composites

Altair | HyperWorks







Altair ESAComp is software for analysis and design of composites. Its scope ranges from preliminary design of layered composite structures to advanced analyses that are applicable to final design verification. ESAComp is a standalone software tool with integrations to other Altair HyperWorks™ products dealing with composites. ESAComp has a vast set of analysis capabilities for solid/sandwich laminates and structural elements.

Product Highlights

- ESAComp materials database, with properties for 1000+ commercial material systems
- Covers layered composite structures from preliminary design to analysis of details
- Standalone tool that interfaces with general finite element packages
- ESAComp integration with HyperWorks enhances composites pre- and post-processing
- Used worldwide in all industries utilizing high-performance composites

Learn more:
altair.com/esacomp

Benefits

Explore Possibilities

The combinations of composite material systems and structural concepts are limitless. The ESAComp materials database gives a good basis for trying potential materials for a design. The analysis capabilities of Altair ESAComp enable quick and easy trade-off studies, such as between solid, sandwich or stiffened designs.

Be Efficient

Using the right tool at different phases of a project ensures efficiency. The FEA environment is not ideal for laminate level studies or lay-up design. In the early phases of a project, the structural elements of ESAComp provide fast analysis without a full geometric model.

Avoid Pitfalls

Designing with composites is a challenge. Without careful assessment of the structure, a potential failure mode can be easily missed. ESAComp complements the capabilities of FE tools in doing this. Advanced ESAComp features, such as probabilistic analysis, become very useful when verifying the real performance of the design.

Optimize Your Design

Besides the user environment that allows practical hands-on optimization of designs, ESAComp integrates as part of more complex optimization systems.

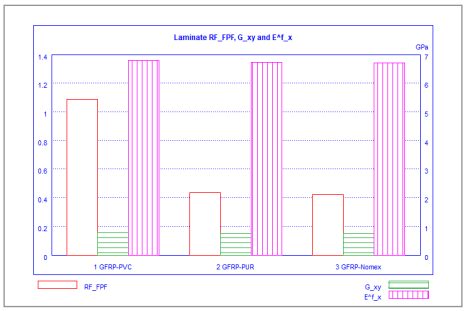
Learn About Composites

To help users get started, ESAComp's easy-to-follow documentation includes tutorials and reference materials, along with first-class technical support.

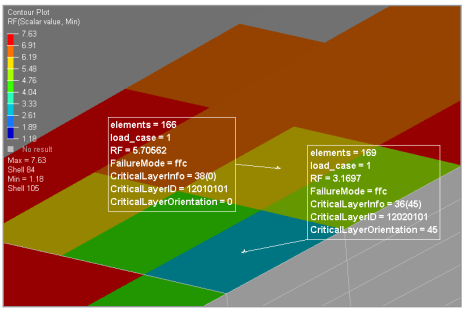
Capabilities

Data Bank and Multi-level Database

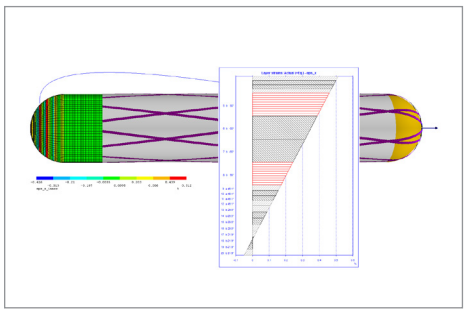
The ESAComp Data Bank includes data for a wide selection of composite materials



Comparison of laminate properties



In-depth failure information for Altair HyperView™



Composite pressure vessel analysis

and material systems. User and company specific material libraries can be stored in the database, as well as data related to design studies. Besides materials, the ESAComp database includes ESAComp objects: fibers/matrix materials, plies, laminates, beams, panels, cylinders, bonded and mechanical joints, loads, and boundary conditions.

Design Environment

ESAComp forms an efficient platform for performing design studies of composite structures. The display of graphical results helps in performing trade-off studies between materials and structural alternatives.

Users have almost limitless options for selecting and combining the resulting data and to display it as numeric tables, layer, bar, line, and polar charts, carpet plots, failure envelopes and 3D contour plots.

Comprehensive Documentation

ESAComp documentation not only helps structural engineers familiarize themselves with composite engineering, but also provides extensive theoretical reference documents needed by composites experts.

Analysis Capabilities

Analysis tools are available for various aspects of composite structural design. From micromechanics, through laminate stiffness and hygrothermal expansion using Classical Lamination Theory (CLT), laminate failure with selected failure criterion (including advanced criteria like Puck 2D/3D, LaRC03) to analyses of flat and curved rectangular panels, including stiffened ones (failure under transverse and in-plane loads, buckling, natural frequencies, geometrically nonlinear analysis/post-buckling). Beams, cylinders/cones (with and without stiffeners; geometrically nonlinear analysis/post-buckling) and pressure vessels (shell and solid element) can be easily modeled. Furthermore, bonded and mechanical joint simulation, probabilistic analyses, simulation of moisture diffusion, prediction of delamination/debonding and much more are possible.

SI and British/U.S. Units

Units and output formats can be changed at any time during the session.

Python Scripting

Enabling extension of analysis capabilities with user scripts, execution of batch runs, and integration to user's design workflows.

Altair HyperWorks Integration

Pre-processing

Materials and lay-ups can be transferred to Altair HyperMesh™ while taking advantage of ESAComp's capabilities, such as multiple strength sets, environment-dependent material data (e.g. temperature or moisture), the practical user interface for lay-up definition, and its laminate design capabilities.

Material and laminate data can be exported in the following solver formats: Altair OptiStruct™, Nastran, Abaqus, ANSYS, and LS-DYNA.

Post-processing

FE results, material, and lay-up data of related elements can be transferred to ESAComp for additional failure analysis. Failure analysis includes the application of advanced failure criteria such as Puck or LaRC03, which are often not available in solvers. Reserve factors are computed for all layers as well as on laminate level, covering, for example, wrinkling failure and interlaminar shear. This information can be passed to Altair HyperMesh™ Desktop for visualization along with information on failure modes and critical layers.

Further, ESAComp offers through-the-thickness plots, which illustrate the stresses, strains or reserve factors (margins of safety and inverse reserve factors, respectively), Loads and laminates can be derived from the imported load case to benefit from ESAComp's whole set of tools for design improvement.

Currently supported solver profiles are Altair OptiStruct™ and Nastran.