

# FiberSIM®

## Analysis Interface™ for ANSA

### How ANSA and FiberSIM Work Together

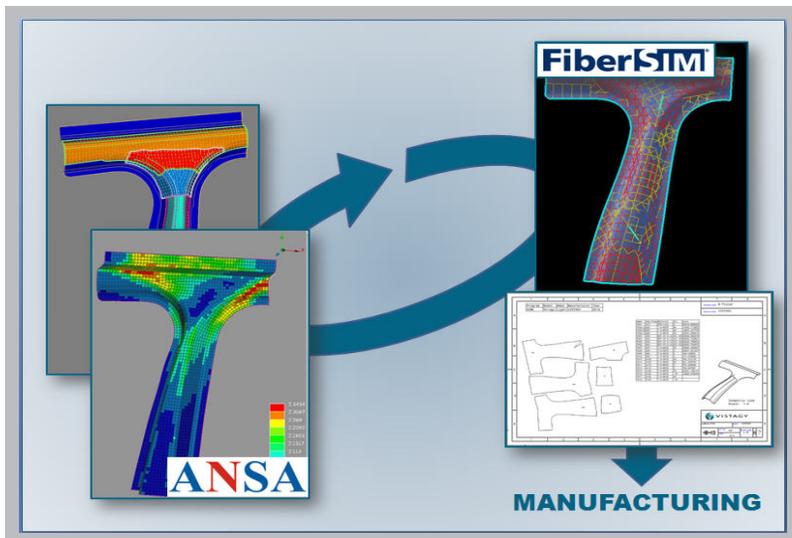
The integration of BETA CAE's ANSA and VISTAGY's FiberSIM enables the exchange of composite layup data so both design and analysis are completed from the same master model. ANSA analysis is performed using the detailed ply layup and true fiber orientations provided by FiberSIM's producibility simulation. Design changes are easily and rapidly transferred between the two applications so more iterations can be completed more quickly. This results in an optimized part that is ready for manufacturing earlier.

Composite materials offer engineers tremendous potential weight savings, increased performance, and design flexibility. However, designing products with these materials can be very complicated. A typical part is made of tens or hundreds of individual plies of various materials, each having a unique shape, orientation, and location. This complexity is compounded by the fact that in most cases the final design of a part cannot be analyzed in its to-be-manufactured state. This greatly increases both the perceived and real risk of using composite materials.

The FiberSIM® Analysis Interface™ for ANSA solves this dilemma by

enabling a concurrent design and analysis workflow that allows design and manufacturing organizations to dramatically reduce engineering and prototyping costs as well as cycle times.

The typical composite part design process starts with the analyst using ANSA software to determine the requirements for the design, including laminate thickness, material etc., based on expected loads. Engineers can then use FiberSIM software to design the actual part in the CAD system. FiberSIM has many tools to assist the designer in achieving the design specifications provided by the analyst. However, even with the best tools and most diligent efforts, the final design may differ significantly from the idealized part that was analyzed. Fibers in the resulting laminates deviate significantly from the specified ply orientations, causing unknown variations in properties. The final design will also contain many details and modifications, including additional plies that were not considered in the original analysis. All of these issues can have a considerable effect on the performance of the final part. Prior to the existence of the FiberSIM Analysis Interface, the true performance of the final part was consequently never known.



The FiberSIM® Analysis Interface™ drives analysis from the same single CAD master model that is used for design and manufacturing. This permits engineers to analyze a part in its to-be-manufactured state.

# Supporting the End-to-End Composite Engineering Process Across Multiple Industries

The FiberSIM Analysis Interface addresses this problem by enabling engineers to design a composite part in their familiar CAD system, determine the state of the manufactured part using simulation technology, and then use the complete definition of a part in its to-be-manufactured form as input into the ANSA pre-processing environment. The analyst can now re-run the analysis using the ply definitions and fiber orientations from the detailed part design, resulting in a more accurate assessment of the part. The analyst can also send back additional ply modifications to FiberSIM if required.

Closing the loop between the FiberSIM designer and the ANSA analyst allows companies to proceed to the manufacturing stage with greater confidence that parts have been properly designed. It also eliminates the practice of part over-design that so often defeats the original purpose of using composites in the first place.

**“When the leaders of two fields join forces, they can achieve great things. VISTAGY and BETA CAE Systems have integrated FiberSIM composites engineering software and ANSA finite element analysis software to deliver a high performance, cutting edge solution that drives innovative design and analysis methodologies in the composites engineering domain. It is a great opportunity to work with such an inspired development partner to meet and exceed the requirements of the engineering community.”**

-Dr. Eng. Sam Saltiel, Head of Customer Service, BETA CAE Systems SA

## Key Features

- **User friendly environment:** The FiberSIM Analysis Interface operates entirely from within the CAD / FEA user interface. Users work within a familiar environment so there is no new interface to learn. This encourages use of FiberSIM products early and often in the design process, supporting a concurrent engineering methodology.
- **Data integrity:** The FiberSIM Analysis Interface ensures data integrity and accuracy. The FiberSIM Analysis Interface operates directly on the CAD model of the part, evaluating native CAD geometry with no translation or approximation. All the information required for analysis, design, and manufacture is generated from the same CAD model. This ensures success when the separate data paths all converge at the lay-up station.
- **Accurate analysis:** The interface ensures that a detailed composite part definition is used for analysis. FiberSIM uses the part surfaces, 3D ply boundaries, and ply stack-up to compute true fiber orientations that can be used readily for accurate structural analysis.
- **Rapid iterations:** Laminate definitions are independent of the analysis mesh. The analysis mesh can be optimized without re-specifying the composite plies. Engineers are able to perform more design iterations more quickly, yielding final part designs in a shorter period of time.
- **Multiple options:** Many material and process simulation models are available. FiberSIM currently supports simulations of woven, unidirectional, and non-crimp fabrics (NCF), for dry and prepreg materials.

