Fiber-Reinforced Composite Materials

Using FEA to Optimize Composite Layer Thickness to Ensure a Specific Safety Margin at Minimal Material Costs
The constant effort to improve product quality while at the same time reducing production and material costs has led to more complex designs and to the pushing of their limits. Due to the diverse properties of composite materials, such as being lightweight, durable and resistant to corrosion, they have been employed for a long time in several sectors, for example in the automotive and aerospace industries.

Composite materials are, however, more difficult to employ, as their behavior is more complex than those of homogenous materials. In this White Paper, we present a brief explanation of composite materials, their failure modes, and the steps required to model them using the finite element method.

In a Case Study about a rotor blade section we show that, after careful analysis, a reduction of the material is possible while maintaining the requirements of the product. The simulation is performed with Dassault Systèmes SIMULIA Abaqus and the Composites Modeler Add-on for Abaqus/CAE.

What are composite materials?

Composite materials are made of two or more constituent materials with significantly different properties. These constituents are joined together to achieve a specific behavior of the final structure. In the context of fiber-reinforced materials, fiber and matrix materials are used. The fibers are usually made of glass or carbon while the matrix often consists of epoxy material.

Due to the different components and processes available, composite materials can be extremely versatile and efficient. Several material combinations can be employed, allowing design flexibility. Furthermore, they are easily formed into complicated shapes. Continuous fiber-reinforced plastics offer the possibility to adjust the behavior needed for a specific application through the stacking of different layers in various orientations.

Additionally, composite materials are extremely lightweight: Their weight is often around 25 percent the weight of steel and they are approximately 30 percent lighter than aluminum. They are also stronger than many materials: Reinforced carbon fiber can be up to five times stronger than grade steel. Besides, composites can resist damage from weather and chemicals, have a long service life and require little maintenance.
Failure causes safety risks and additional costs

Failure of composite materials leads to safety risks along with additional costs for repairs. In the context of continuous fiber-reinforced plastics, prevention requires the study of the different types of failure, which can be classified into interlaminar failure and intralaminar failure, and their complex interaction.

<table>
<thead>
<tr>
<th>Interlaminar failure</th>
<th>Intralaminar failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Separation of the adjacent lamina, often referred to as “delamination”</td>
<td>Failure in the micro-mechanical components of the lamina, e.g.</td>
</tr>
<tr>
<td>Fiber fracture</td>
<td>Matrix cracking</td>
</tr>
<tr>
<td>Separation of fiber-matrix interface</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Failure processes of composite materials

Preventing failure by means of simulation

Simulation can be used to identify weak points at an early stage and to investigate design alternatives. In this way, quality, product performance and safety are not only guaranteed, but in many cases even improved. Additionally, production and material costs can often be minimized. For the analysis of a structure with composite materials the following specifications are required:

A. Geometry and discretization: Depending on the type of application, it is determined whether solid or shell elements are suitable for the simulation. Subsequently, an appropriate geometry and mesh can be created.

B. Material definition: For transversely isotropic laminates, ten different parameters are required to characterize the material behavior. These parameters must be provided by the client.

C. Creation of composite layups: The layout of the plies layup must be carefully planned in order to obtain the correct section and orientation of the materials. With the help of the SIMULIA Abaqus Composites Modeler, additional draping simulations can be performed. Furthermore, the manufacturing process can be considered.

For unidirectional materials (UD), five fracture modes typically exist:

<table>
<thead>
<tr>
<th>Fiber failure</th>
<th>Inter-fiber failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fiber parallel tensile fracture</td>
<td>Transverse tensile fracture</td>
</tr>
<tr>
<td>Fiber parallel compressive fracture</td>
<td>Transverse compressive fracture</td>
</tr>
<tr>
<td>In-plane shear fracture</td>
<td></td>
</tr>
</tbody>
</table>

Table 2: Fracture modes of unidirectional materials
Case Study: Optimization of a rotor blade section

A section of a rotor blade is investigated in order to obtain information regarding the stress levels of the material and estimate if the material thickness and therewith the weight can be reduced. The failure modes should be determined. The section is composed of FOAM, UD and BIAx materials and therefore of a continuous glass fiber-reinforced plastic (GFRP). A dynamic overload (covered with a static load case) is estimated and a Margin of Safety (MoS) is calculated. The thickness of the glass fiber-reinforced plastic layer and the stacking are idealized.

Step 1 – Geometry and discretization

- The section of interest is taken from a CAD geometry representing the full rotor blade. The geometry is composed of the mid-surfaces of the structure.

- The model is discretized using shell elements with composite sections. Since transverse shear effects are not of interest here, composite solids are not necessary in this case.
**Step 2 – Material definition**

- The domain is divided into different sections in order to apply the plies. As an example, figure 3 shows one section with the corresponding normalized material-thicknesses.

- The composite materials (UD, BIAx) are characterized by ten different material parameters (transverse isotropic laminate, including strength parameters).

<table>
<thead>
<tr>
<th>Material</th>
<th>Thickness [normalized]</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIAx</td>
<td>0.05</td>
</tr>
<tr>
<td>UD</td>
<td>0.90</td>
</tr>
<tr>
<td>BIAx</td>
<td>0.05</td>
</tr>
</tbody>
</table>

Figure 3: Rotor blade section with corresponding normalized material-thicknesses

**Step 3 – Creation of composite layups**

- The composite layups are defined by defining the thickness and orientation of the plies. These can be modeled using the SIMULIA Abaqus Composites Modeler or classical SIMULIA Abaqus/CAE.

- Several local coordinate systems are introduced.

- After the sections have been defined, the orientation of the plies is checked with help of the ply stack plot.

Figure 4: Composite sections and example ply stack plot
Step 4 – Modeling the problem: Boundary conditions

- Couplings are introduced to idealize the model and provide points for boundary conditions and loads.
- A specific point-load (force) is used as a load estimate for the whole model.

Failure analysis

- All individual failure modes’ efforts are calculated. An additional interaction mode determines if the material will fail. Failure is predicted when the total effort reaches unity.
- In this analysis, the MoS value obtained for the most critical location is greater than 11. This leads to the conclusion that layup optimization is possible, in order to reduce both material and weight.

Figure 5: Section with individual coordinate systems and couplings

Figure 6: Maximum logarithmic strain over all layers (normalized)

Figure 7: Margin of Safety – parallel fiber direction
An optimization analysis is performed in order to reduce the expensive GFRP material:

- A material reduction of 33 percent compared to the original material mass is achieved by optimizing the layer thickness of the GFRP material.
- This significantly reduces material costs.
- The total weight of the rotor blade section is reduced by around 25 percent compared to the initial weight.
- Safety-requirements are still satisfied by the MoS with a new value greater than 5, compared to 11 with the initial thickness.

**Additional: Draping simulation to enhance the accuracy of the results**

The manufacturing process of the components also plays an important role in the material behavior. As an abstract example, draping simulation is performed using the rotor blade geometry.

The manner in which the plies are applied to the mold can be simulated by employing a different seed point (application point) which results in different initial shear strains in the ply.

**Figure 8: Draping simulation – pattern 1**

**Figure 9: Draping simulation – pattern 2**
In order to enhance the accuracy of the results, draping simulation is critical for any geometry with double curvature. In this case, the soft double curvature of the geometry creates significant differences in the predicted stresses.

Figure 10: S11 stress (normalized) – pattern 1

Figure 11: S11 stress (normalized) – pattern 2

**Conclusion: Significant cost saving and weight reduction possible**

A rotor blade section was analyzed using the finite element method with the SIMULIA Abaqus software. Materials and composite layups were defined in order to capture the behavior of the structure. A failure analysis, which takes into account the different modes of fracture of the present composite materials, was carried out.

While considering the Margin of Safety, the analysis showed that weight and cost reduction was possible via optimization of the layer thicknesses. An additional draping simulation using different application points showed significant variation for the stresses and helped to sharpen the results.
We help you save material, weight and money

Wölfel provides comprehensive services for the analysis of structures where composite materials are present. Our support ensures that your model will be optimally defined to obtain reliable results. We can assist you with the generation of an optimal geometry, selection of appropriate element types, meshing, as well as with the creation of the composite layups. When requested, we also take charge of the whole product optimization according to your specific requirements.

We perform failure and Margin of Safety studies with state of the art criteria. Specific criteria can be implemented per client request. Furthermore, we perform draping simulations for complex geometry that contains double curvature, in order to take into account the initial shear strain from the manufacturing process.

You can also acquire the Composites Modeler Add-on for SIMULIA Abaqus/CAE through us. This Add-on complements and extends the ply modeling features present in Abaqus/CAE, directly transferring accurate fiber angles and ply thickness to the simulation for improved accuracy.

Your personal contact

Dr.-Ing. Manuel Eckstein
+49 931 49708-290
eckstein@woelfel.de
Vibrations, structural mechanics and acoustics – that is the world of Wölfel. In this world, we are the experts. It is our home. More than 100 employees are doing their best to the satisfaction of our customers every day. For over four decades, we have been supporting you worldwide with engineering services and products for analysis, prognosis and the solution of vibration and sound-induced tasks.

Are vibrations really everywhere? Yes! That’s why we need such a wide variety of solutions! Whether it is an engineering service, a product or software – there is a special Wölfel solution for every vibration or noise problem.

For example the

- simulation-supported design of industrial plants and power plants against earthquakes
- measurement of the sound emission of wind turbines
- universal measuring systems for noise and vibrations
- noise protection assessments and pollutant forecasts
- dynamic passenger simulations in automobiles and aircraft
- and many other industry-specific Wölfel solutions…