INTEGRATED SIMULATION-SUPPORTED COMPOSITE PRODUCT DEVELOPMENT – DREAM OR REALITY?

Thomas Burkart, Monika Kreutzmann, Petra Schilling

P+Z Engineering GmbH, Frankfurter Ring 160, Munich, Germany
Email: t.burkart@puz.de, Web Page: http://www.puz.de

Keywords: Simulation process, structural components, automotive, composite materials, development process

Abstract

Given the widespread use of structural components made from composite materials in the automotive industry, the challenge for simulation is to adapt the development process to suit the requirements of composite structural components and provide efficient ways of facilitating the design and development of such components under the constraints associated with fiber-optimized structures.

To meet this challenge, P+Z Engineering applies an integrated simulation-supported approach to product development. This approach allows the advantages that fiber-reinforced materials bring to the development process to be exploited without ignoring the "industrialized engineering" aspect. It requires all parties (design, simulation, production and testing) to work in close cooperation from the beginning of the development process so as to generate an optimum solution which takes account of all disciplines, prevailing constraints and the development objective. The aim is to present the entire operation with the aid of an exemplary development process and to depict the remaining difficulties in the simulation of composites and the approaches used to overcome them.

The field of virtual development of composite components still has some catching up to do. In the pre- and post-processing phases too, standards need to be defined that leave no scope for philosophical interpretations of simulation results.

1. Motivation

For continuous fiber reinforced products to be used in structural applications in the automotive field, it is first necessary not only to overcome the challenges associated with the series production of composite components, but also to predict with reasonable precision the loads to which these components will be exposed. If innovative components are to be used that offer optimal load paths, are designed using new construction methods, and are manufactured using advanced production processes, changes will need to be made at all stages of the virtual product development process chain. Given the widespread use of structural components made from composite materials in the automotive industry, the challenge for simulation is to adapt the development process to suit the requirements of composite structural components and provide efficient ways of facilitating the design and development of such components under the constraints associated with fiber-optimized structures.

In order for similar development times to be achieved using composite structural components as are achieved with conventional components, changes need to be made, and know-how from previous projects applied, at every single stage of the development process. In the past, development processes involving structures of this kind have been based largely on trial-and-error. This was due to a lack of experience and the limitations of the available engineering tools, which meant that almost entirely computer-based, sustainable development processes were not possible. Driven by the demands of the automotive industry, a number of new tools have recently appeared in the system landscape that offer very great potential, but that can only be exploited fully and achieve the highest possible quality of results if the development engineer using them is suitably experienced. In products made from

Excerpt from ISBN 978-3-00-053387-7

composites, the material does not exist until the moment of production, and does not represent a "homogeneous", uniform starting material. This adds a further dimension to the design of structural components, and hence – depending on the nature of the production process and the degree of maturity of the product – a further variable.

As a development company specializing in CAE and simulation, we are interested in offering the best possible industrialized engineering solutions, in which as little time and money as possible is wasted on trial-and-error. To achieve this, it is important to be familiar with the available tools and their potential and limitations, and know how to use them, but also to allow the aforementioned know-how to be incorporated usefully into the process where necessary. Simulation needs to provide assistance in turning ideas into products, an approach that will be looked at in greater depth below, in relation to the concept, development and detailing phases. The potential and weaknesses of this approach will be outlined.

2. The simulation-supported development process

This section will look at the virtual, simulation-supported product development process, the possibilities it offers at each development stage, and its limitations.

2.1 The concept phase

Automotive structural components are exposed to a wide range of load scenarios, and also need to meet additional requirements arising from situations that may only occur a few times in the entire life of the component, for example "misuse loads", resulting from e.g. driving over an obstruction too fast, or crashing. Often the material of a component is swapped for a different one while keeping the same shape. Substitutions of this kind do not, however, allow the advantages of composite materials to be exploited sufficiently, if at all, since in these cases components made from fiber composite materials generally need to be increased in size, due to less-than-ideal geometries and layer/fiber orientations. To avoid these problems and derive genuine added value from fiber composites, the best approach is to start with a "blank sheet", and factor in the type of material right from the design stage.

2.1.1 Topology analysis

The first step of simulation-supported product development is "topology analysis", in which the main load paths are identified and an initial approximate draft design is generated on the basis of the key constraints (requirements).

FEM-supported topology analysis is a method for developing structures in which the load distribution is ideally uniform, and where as little material is used as possible. Here it is vital to be aware of the requirements profile as it currently stands, in order to specify the correct framework conditions for the analysis (which are small in number, but crucial to the success of the simulation). Among the most important variables to enter at this point are the available design space (i.e. the installation space in the vehicle), the bearing/connection points, and the dimensioning-relevant mechanical load data.

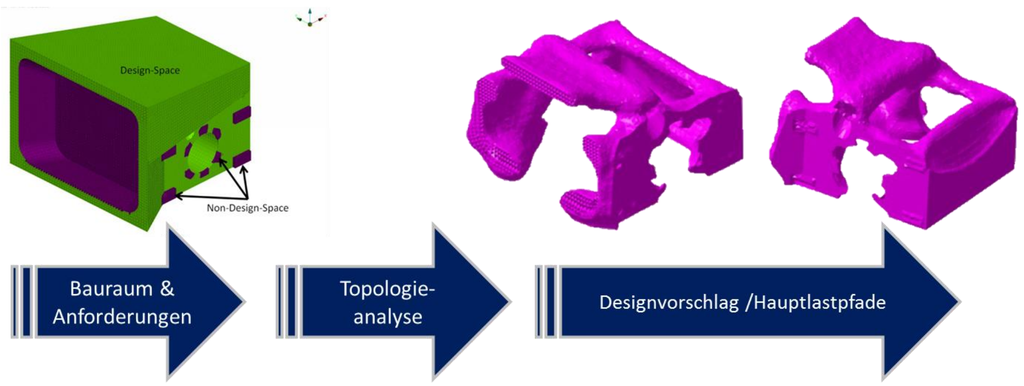


Figure 1: Topology analysis for an electric motor mount

Excerpt from ISBN 978-3-00-053387-7

Today this method is still rarely used in the development of products made from composite materials, because the load distribution in the component needs to be uniform (i.e. an isotropic material). In the process described in this article, this analysis is used mainly for the initial determination of the geometry, and for identifying the main load paths. These are not subsequently accounted for by thicker walls and/or ribs, as they are in the case of isotropic materials, but through selection of the optimum fiber material and orientation.

2.1.2 Design

The results of the analysis can now be used to obtain a first “realistic” geometry, select an initial material, and decide on possible construction methods.

However, input should be obtained even now from the design, testing and production departments, to enable meaningful decisions to be made even at this early stage. Although the development process is only in its very early stages at this point, the know-how of the other disciplines is already essential if a well-balanced concept is to be achieved. There are not currently any tools available to replace this know-how in industrial-scale development processes.

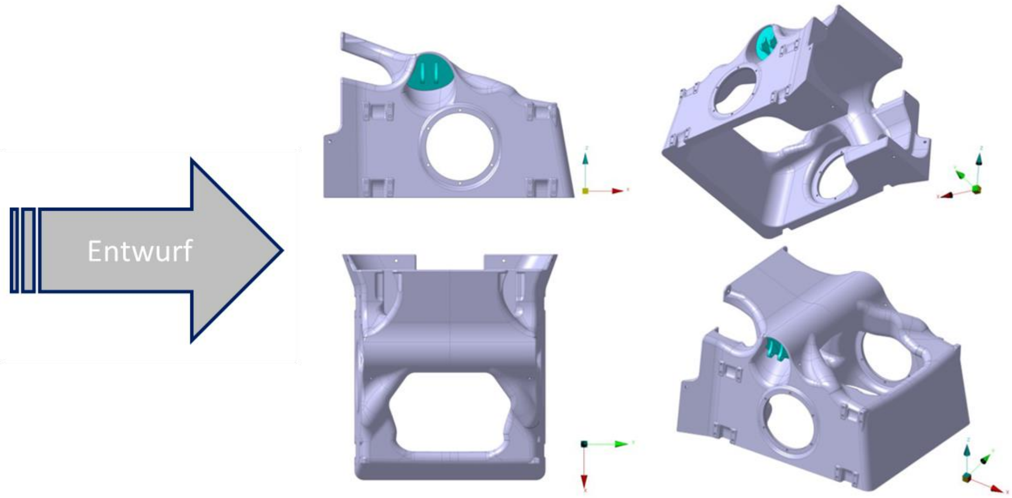


Figure 2: Initial design concept based on the topology analysis

Choosing a material:

The first step here is to choose not just the type of fiber, but also a suitable matrix. Mechanical loads need to be taken into account, as do other aspects, including the construction method, number of units to be produced, environmental influences (chemical resistance), recycling, etc. As regards structural components, the emphasis is generally on mechanical performance. The material is of course also chosen on the basis of the development objective.

In terms of the fiber, there are two main decisions to make: the type of fiber (carbon, glass, etc.), and what kind of configuration is needed (UD, roving, weave, mesh, etc.).

During the design phase a number of different possible approaches are discussed, but UD fiber is generally used initially for purposes of the simulation. A major challenge in this respect is knowing what material data to use. As mentioned above, the material – and hence its properties – only come into being at the time of production of the component. At the component design phase, the production process has not yet been clearly defined, and it is not yet possible to produce samples that can be used to determine the material’s characteristic values. This is not due to challenges resulting from the use of simulation, but is due to the development process for the composite product itself. In order to proceed with the simulation-supported development process, material data are needed at this stage that are similar to what can later be expected. Like other variables, these material data, and any factors used to account for influences during production, are currently derived from know-how gained from past projects. The current simulation landscape does not offer ways of generating the data themselves, or rules for taking account of production influences.

Excerpt from ISBN 978-3-00-053387-7

Construction method:

The decision over whether to use a sandwich, shell or plate construction is generally taken early in the design phase. If additional functions may subsequently be integrated into the product, thereby changing its system boundaries, this will also affect the choice of construction method and decisions regarding connection types.

Based on consideration of requirements relating to lightweight construction, suitability for implementation using fiber composites, and maximum integration, a sandwich construction was found to be particularly suitable for the concept shown here. This type of construction naturally has its own special implications for the component simulation process, which are described in section 2.2.

Connection methods:

Two different types of connection need to be considered: Firstly, connections joining together individual elements to form a single component, and secondly, connections between the component and its neighboring structures. With regard to the first type, it is necessary to look back to the results of the topology analysis, because the joints should ideally be positioned at points on the component that are exposed to only very small loads. Once again, engineering know-how is needed here, because the positions of the joints must be chosen on the basis not only of the load paths, but of course also the aspect of manufacturability. Adhesive joints are generally used for the joints within the component. The engine mount shown above is produced in one piece, i.e. without any joints, using a suitable tool concept. This is possible in this case thanks to the construction method selected.

For joining components to neighboring structures, bolt connections are generally used, in a very wide range of different incarnations. Peel and bending stresses should be avoided as far as possible in joints of this kind; the design concept and all subsequent steps should take account of this. Possible settling should be borne in mind when selecting the positions of bolts.

Once concepts have been drawn up for all factors relevant to dimensioning and design, a concept then has to be chosen based on criteria such as manufacturability, TRL (Technology Readiness Level), fulfilment of the development objective, etc. Ideally this will produce an initial overall design concept that is a synthesis of all the best individual solutions, which should then serve as the starting point for the development phase.

2.2 Development phase

The development phase begins with preparation of a first complete CAD dataset, and specification of the materials that will actually be used, based on the overall design concept. These two types of "data" serve as the input variables for all subsequent simulation work.

In each of the process steps shown, to obtain meaningful, realistic results, appropriate adjustments and settings must be made to reflect the fact that fiber composites are being used.

2.2.1 Pre-processing

Inputting the CAD model & simplifying the geometry:

Depending on the maturity of the CAD data, it may be possible – or indeed necessary – to simplify the geometry of the component for purposes of the simulation. It is possible to make simplifications that have no relevance, or barely any relevance, to the "problem" at hand. In initial simulation runs, small drill holes can be closed, radii and chamfers can be removed, and connecting elements can be "imitated" using suitable elements, or the connections idealized. The simplifications made here should however be kept in mind until the later detailing phase, because no standards exist with regard to what simplifications are acceptable in terms of their influence on the quality of the results.

Meshing/modelling:

Simplifications can be made not just to the component geometry, but also in terms of setting elements to a specific size. In an internal study, we investigated the influence of element size on the "correctness" of results in the form of the Tsai-Wu failure index.

Excerpt from ISBN 978-3-00-053387-7

We determined that the Tsai-Wu failure index increases as the mesh density increases, up to the point of convergence, and this increase is not linear. When conducting a component analysis in FEM, the precision of the results should therefore always be questioned.

There are other special factors to consider when modelling sandwich components if meaningful results are to be obtained: sandwich structures with a core thickness of less than 10 mm are meshed with just one shell element plane. In the laminate definition, the core then corresponds to one or several layers (depending on the point in the cross-section at which stresses and strains are to be output). Where the core thickness is 10 mm or more (or if the core geometry is too complex), the core is meshed with first-order solid elements, and the cover layers with shell elements. Special attention should be given in this case to the element orientation and layer sequence (see below). The shell elements should ideally be joined homogeneously to the volume elements, i.e. with coincident nodes. It must also be decided on what planes the shell elements should be meshed. Normally the plane on which the monolithic area is meshed is used on the tooling side as well. However, the center plane in the sandwich naturally changes. On the bag side, it is possible to mesh on the middle surface, or directly on the plane of the adhesive bond between the core and cover layers. In general it is thus necessary to adjust the height of the core (solid), and to assign appropriate offsets for the cover layers.

Assigning material properties and specifying additional framework conditions:
When choosing what material data to use, it is necessary to consider the component’s area of application, and possible safety strategies. If some engineering constants are known, a proprietary program (i.e. one developed for use internally at P+Z) can be used to determine the others. It is also possible to calculate engineering constants for a variable fiber volume content, based on the rule of mixtures. Solutions are certainly available today for performing these “simple” calculations – i.e. proprietary software or tools programmed by e.g. universities or institutes – but no applications have so far become established that are integrated in the standard tools.

Depicting connections:
Connections are normally depicted in almost exactly the same way as when modelling isotropic materials. For purposes of initial, approximate analysis, screws/bolts/rivets are represented using Rigid Body Elements (RBE2) positioned on the components that are to be joined, plus a beam element of perfect rigidity (CBAR/RBAR & CBUSH), to enable the bolt loads to be read off. For detailed modelling, the shape of the screw/bolt/rivet should be depicted as realistically as possible using a solid, and the conditions in the joint should be modelled using contact and pretension elements.
For these initial investigations, bonded joints are generally modelled using a combination of solid elements for the adhesive, and RBE3 elements for the joint. This allows the properties of the adhesive to be taken into account without any exaggeration of the stress in the composite components. For purposes of detailed investigation, here too the model needs to be adapted, where relevant using different element types to which a failure model can be assigned, e.g. cohesive elements.

This overview of the key issues already provides a good indication of the limitations of the pre-processors available today. Meaningful results can only be obtained if the component is modelled appropriately, but calculation times, and framework conditions specific to the individual solver, also need to be considered. Finally, pre-processing should also always include model validation, which should be based on the following criteria:

- Element quality
- Free edges/element connectivity
- Coincident nodes
- Duplicate elements
- Element orientation
- Element offsets
- Material orientation

2.2.2 Solvers

Excerpt from ISBN 978-3-00-053387-7

Developers are virtually unrestricted in their choice of solvers for use in the simulation of composite components, because virtually all standard tools can handle fiber-composite structures and the associated challenges. P+Z Engineering uses a wide range of different solvers to suit different purposes and simulation methods. For example, for linear simulations for assessing rigidity, it uses MSC Nastran; for assessing strength, or for non-linear simulations, it uses Abaqus Standard or ANSYS; and for explicit (crash) calculations, it uses Abaqus Explicit, PAMcrash and LS-Dyna, among others. For the topology optimization referred to above, it uses TOSCA or DAKOTA.

2.2.3 Post-processing

The post-processors available offer a very wide range of possible settings; this fact alone means that the results produced can vary enormously. One setting defines, for example, whether the values for an element are first determined in the nodes, and the average of the sum of these values then calculated, or whether the averages of the node values are first calculated, and then added together. As with isotropic materials, it is important to bear in mind in what coordinate system stress and strain values are being presented. Composites should always be evaluated in the material coordinate system – or in an element coordinate system aligned to that. In the case of shell elements, it is necessary to check at what point these are evaluated. The solver calculates the values at the integration points or Gauss points. The values for the element (centroid) or nodes (corners) are calculated; this interpolation is either performed by the solver, or later, by the post-processor.

Nastran only outputs centroid values for PCOMPs, so in Nastran calculations, the centroid values are also evaluated. Abaqus calculates the values directly at the Gauss points. Abaqus allows users to define the number of evaluation points across the cross-section of the individual layer. The values for the bottom, top and center can all be output. A good approach is to output the values for the bottom and top of each individual layer. Nastran only outputs the stress in the center of the layer. This is particularly important to bear in mind in calculations involving a small number of thick layers, such as e.g. in the sandwich construction in our example.

Efforts are generally preferable to failure indices, because the information they provide about the load has linear scalability (double load □ double effort). This is not the case with failure indices. In the case of the strain criterion, the failure index provides the same information as the effort. Some post-processors offer “composite tools” that can automatically read in and depict all stresses. This enables plots to be created that determine the critical layer or direction for various failure theories (e.g. Puck), or that show how stress or strain vary at different points in the cross-section.

2.3 Detailing phase

The detailing phase consists of several iterative loops aimed at achieving an optimal layer configuration, inserting appropriate local reinforcements, deciding on a final design for joints, finding the right level of drapability, etc. In addition to considering the most relevant load cases, at this point it is also necessary to provide evidence of the component’s performance in relation to all other requirements, including derivative requirements. The kind of challenges faced during simulation will depend on the development objective (surface, rigidity, weight, crash performance, cost, etc.).

2.3.1 Optimization

The optimization tools currently available are capable of adjusting the orientation and position of the fiber reinforcement within the laminate in such a way as to optimize their resistance to loads. However, the resulting design is generally not ideal from the point of view of component manufacture, so a large amount of know-how and “manual work” is still needed.
One possible approach is to apply a three-phase composite optimization method using OptiStruct. This method is described below.

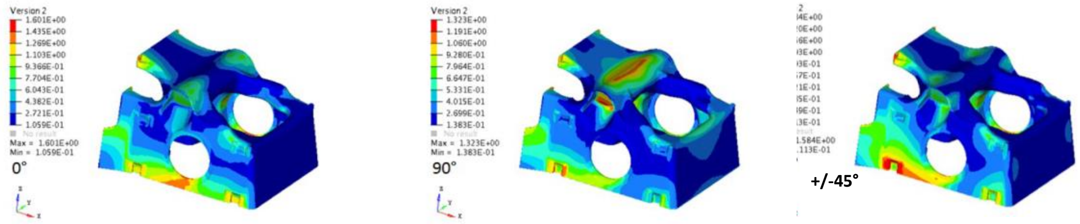
OptiStruct works well with linear problems where composite components with a layer-based structure need to be optimized in relation to specific target values (e.g. weight, rigidity or strength). The specified fiber orientations (e.g. 0°/45°/-45° and 90°) and layer thicknesses and dimensions should be

Excerpt from ISBN 978-3-00-053387-7

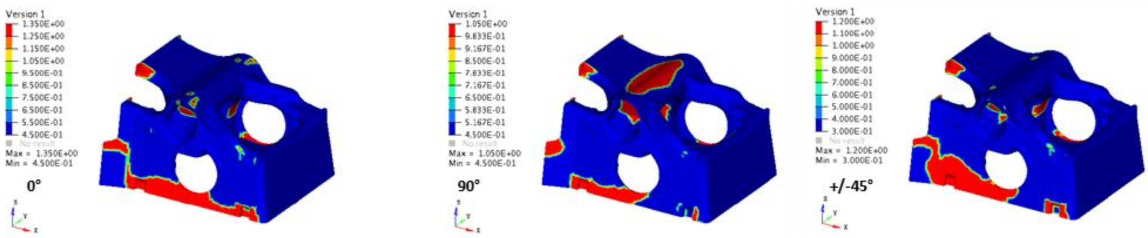
used and arranged optimally from the point of view of load paths. The aim of the 3-phase optimization process is to minimize the amount of material used while adhering to the existing framework conditions, e.g. target rigidity parameters or failure criteria.

The 3 phases are briefly outlined below, once again based on the example of a mount for an electric motor:

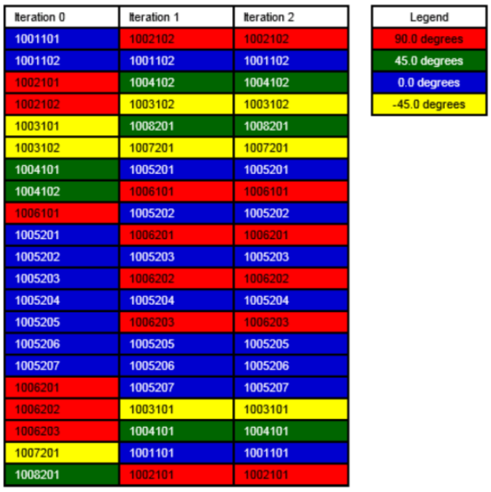
Phase A – FreeSizing:
Generates a design concept for the layer thickness for each of the various fiber orientations



Phase B – Sizing:
Layer-based optimization adhering to specifications regarding the thickness of each individual layer bundle



Phase C – Shuffling:
Determining the optimum arrangement of the individual layers



Iteration 0	Iteration 1	Iteration 2
🟦 1001101	🟥 1002102	🟥 1002102
🟦 1001102	🟦 1001102	🟦 1001102
🟥 1002101	🟩 1004102	🟩 1004102
🟥 1002102	🟨 1003102	🟨 1003102
🟨 1003101	🟩 1008201	🟩 1008201
🟨 1003102	🟨 1007201	🟨 1007201
🟩 1004101	🟦 1005201	🟦 1005201
🟩 1004102	🟥 1006101	🟥 1006101
🟥 1006101	🟦 1005202	🟦 1005202
🟦 1005201	🟥 1006201	🟥 1006201
🟦 1005202	🟦 1005203	🟦 1005203
🟦 1005203	🟥 1006202	🟥 1006202
🟦 1005204	🟦 1005204	🟦 1005204
🟦 1005205	🟥 1006203	🟥 1006203
🟦 1005206	🟦 1005205	🟦 1005205
🟦 1005207	🟦 1005206	🟦 1005206
🟥 1006201	🟦 1005207	🟦 1005207
🟥 1006202	🟨 1003101	🟨 1003101
🟥 1006203	🟩 1004101	🟩 1004101
🟨 1007201	🟦 1001101	🟦 1001101
🟩 1008201	🟥 1002101	🟥 1002101

Legend
🟥 90.0 degrees
🟩 45.0 degrees
🟦 0.0 degrees
🟨 -45.0 degrees

Figure 3: The 3 phases of the optimization process in Optistruct

In phase A it can be clearly seen that different quantities of material with a given fiber orientation are needed depending on the load type and zone. For example, at the lower bolt points, shear loads create a shear field, which increases the need for +/-45° layers in this area. In Composite Size Optimization, the size of the laminate is then determined. This step takes into account the manufactured thicknesses of the individual layers, and the failure criteria of the MLC (multi-layer composite). Here special

Excerpt from ISBN 978-3-00-053387-7

attention must be given to the question of manufacturability. The program does allow stipulations to be made regarding the minimum permitted patch size, but does not rigidly adhere to these constraints. Here the solver uses an integrated automatic patch size definition method based on element size. Manual changes to the “super-plies” therefore often need to be made, for example. This, however, requires relevant manufacturing knowledge, so the viability of the various options must be discussed with the manufacturing department.

In the last step, phase C, the stack sequence of the individual layers in the laminate is optimized. If suitable framework conditions (e.g. stack sequence in the outer layers) are chosen at this point, the solver will “correctly” execute the “Cover” and “Core” functions. A major problem arises at this point because the solver does not follow any generally-applicable rules regarding a symmetrical, well-balanced layer sequence. Therefore this, too, subsequently needs to be compensated for manually. Because this phase of the optimization process provides little help in designing the laminate stack, it is advisable to generate the laminate stack manually based on standard rules and historic values.

2.3.2 Simulating connections

Possible connection types were considered right from the design and development phase. Simulation of these possible connections poses further specific challenges, e.g. selection of suitable modelling techniques, and the question of when it becomes necessary to model the adhesive separately, rather than assuming a simple joint between two materials, as was possible early on in the development process.

2.3.3 Verification

Prior to the detailing phase, design of the component centered on the most relevant load cases and adherence to the development objective, whereas in the detailing phase, calculations and observations need to be made in relation to all applicable requirements. The component often becomes a little “heavier” again at this point, because additional measures need to be taken to achieve these objectives. Various conflicts of objectives generally arise during this phase: e.g. the load case relevant to dimensioning might mean that increased rigidity is needed at a particular point, whereas other criteria demand increased ductility in this area. A vehicle bonnet is an example of a component where this is the case. A balance needs to be found between the aspects of design, optimum use of installation space, and material and fiber orientation, in order to find the solution that best meets all applicable requirements.

3. Summary

In this paper a simulation-supported composite product development process was shown that tries to incorporate all the important topics in order to gain a product that really uses the advantages of composite materials. Another important prerequisite for such a development process to be accepted is that it is usable within a standard industrialized engineering process. It was shown that this is feasible but still needs quite a lot of engineering judgement and experience to deliver a proper product.

Excerpt from ISBN 978-3-00-053387-7