DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

M.Sc.(Tech.) H. Katajisto, Dipl.-Ing. A. Mönicke (Componeering Inc., Finland);
M.Sc.(Tech.) J. Väisänen (Aalto University, Finland);
M.Sc.(Tech.) H. Santamala (Metropolia University of Applied Sciences, Finland);
M.Sc.(Tech.) M. Perillo (EnginSoft S.p.a., Italy);

M.Sc.(Tech.) H. Katajisto, Engineering Consultant

THEME

Composites

SUMMARY

This paper describes how a carbon fiber reinforced plastic body of a future city car was optimized to minimize weight. The frame includes numerous parts, some of which have a simple constant laminate structure and some are more complex having additional local reinforcements. The body must meet the different stiffness and load carrying constraints set by the various load cases. The solution time in a standard engineering workstation for a single load case is several minutes due to the model size and particularly due to the use of contact elements. Such a solution time means a waste of engineering resources since the time taken by the computation is hard to fill efficiently with parallel engineering tasks. The multitude of the laminate structures to be determined and the several load cases make the post processing of the data laborious and decision making for design improvements complex. Defining the best performing feasible design is a very challenging task in many respects and, therefore, the problem was solved using numerical simulation tools within a process integration and design optimization environment.

This paper demonstrates how the state-of-the-art simulation tools were applied in practice in an efficient manner for a composite assembly. A successful product development requires skilled engineers. However, for complex composite structures automated design processes with in-built optimization capabilities are crucial. The design-optimization of the frame was made in the

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

preliminary design phase of the project. The challenge of the project was the trade-off between quality and cost. Practical approaches were needed to meet the time and cost requirements and at the same time solid background for the detailed design was essential. Therefore, applicability of the results and restrictions of the model must be understood. These aspects are also covered in the paper.

KEYWORDS

Composites, Optimization

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

1: Project background

The European Union (EU) wishes to limit the production of waste arising from end-of-life vehicles and to increase re-use, recycling and other forms of recovery of end-of-life vehicles and their components. In order to achieve these two objectives, the EU lays down new requirements for European vehicle manufacturers, who should design vehicles which are easy to recycle.

This is a direct quotation from the directive 2000/53/EC of the European Parliament and of the Council of 18 September 2000 on end-of-life vehicles, which requires that priority must be given to the re-use and recovery (recycling, regeneration, etc.) of vehicle components. For the new vehicles the rate of re-use and recycling (in average weight per vehicle per year) should reach 95% starting from 2015 [1].

Applied research and development work in Helsinki Metropolia University of Applied Sciences is carried out as a part of Metropolia's other primary tasks. Metropolia has a strong knowledge and experience of developing concept cars from the drawing board to the drivable demonstration vehicles. Maybe the most remarkable of their projects was the Electric RaceAbout vehicle (E-RA) [2].

The current project, called the Concept Car, focuses on improving the knowledge of the future city car, which is environmentally friendly during its manufacturing and usage, and which complies with the future requirements. Overall examination and research work will increase the knowledge in the perspective of eco-friendly automotive engineering and, therefore, the project will improve the automotive engineering education. Improved knowledge will be shared with the Finnish automotive industry as well. The Concept Car project is funded by TEKES and some private companies like Componeering Inc. TEKES is the main public funding organization for research, development and innovation in Finland. Partner universities are Aalto University, Kemi-Tornio University of Applied Sciences and Tampere University of Technology.

This study focuses on the design and analysis of the composite frame of the city car for structural sizing purposes. The frame consists of mainly fossil composites. Carbon fiber reinforced plastics are widely used due to their high specific stiffness and strength. With a light-weight body operating costs can be lowered. It should be noted that recycling, low carbon footprint and low emissions were considered in other projects of the Concept Car. Improved knowledge will be demonstrated with a manufactured prototype vehicle that will be used for testing and verification purposes. The demonstration vehicle will be fully functional and drivable, and it will be introduced at 2014 Geneva Motor Show.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

The Concept Car project was started in 2011. The design and construction work is mainly carried by students. For example, a Master’s thesis work was conducted in conjunction with the structural sizing project. The schedule of the project for the preliminary design of the composite frame was somehow comparable to industrial projects. As a student work the use of the FEA tool, the model preparation and the pre-design was conducted during Feb-Mar 2012 and the model verification and optimization by Componeering Inc. during Apr-May 2012.

2: Design

The design of the exterior was made by an industrial design student and together with a design engineer they prepared the CAD surface model, which was directly available for the structural designer. For structural sizing purposes only the main frame was included and structurally negligible details were ignored. Examples of such details are the plastic roof window and the trunk door. The structural frame of the Concept Car is presented in Figure 1. The vehicle is in the class of VW Polo and some technological solutions are comparable.

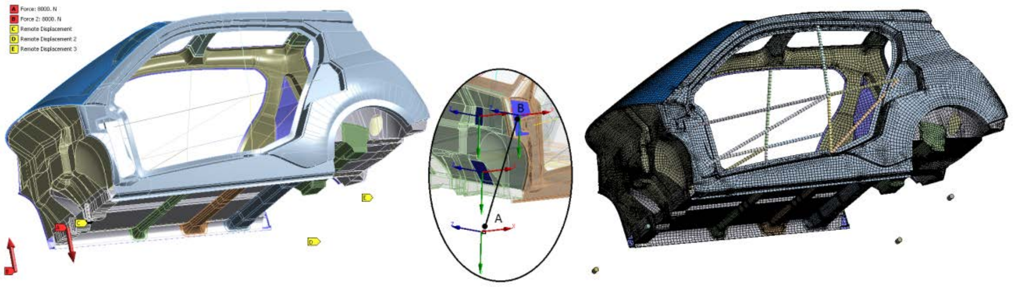

Figure 1: The CAD model of the Concept Car is shown on the left. The load and boundary conditions are shown for the torsional load case as well. Constrained equation between the suspension (A) and the hard mounting point (B) is illustrated in the middle. The element mesh is seen on the right.

The vehicle frame consists of separate composite parts (see Figure 2) and they are attached with hybrid joints. Adhesive bonding is the primary joining method whereas rivets are used for a fail-safe structure. Another reason for the rivets is to ease manufacturing. The rivets provide slight compression to the joint in the manufacturing state while the adhesive is curing. Rivets also help in the positioning of the parts.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

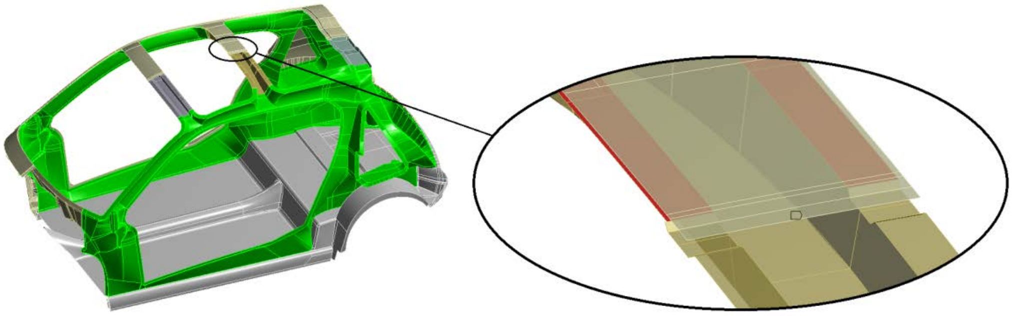

Figure 2: Some composite parts of the assembly and a detailed view of the joint region. The part highlighted in green is the central frame, which is continuous at the rear end. At the roof region and at front the sides of the central frame are connected with beam-like elements. Also the right exterior part of the frame is slightly visible. This right exterior part is seen on the detailed view on top of the beam-like connection element. Red color on the flange of the hat-shaped stiffener represents the contact surface. The counter part of this joint is the bottom side of the right exterior parts seen on top of the detailed view.

In the later phases of the project the geometrical surfaces acted as the base for the lamination. The same surfaces were used for the construction of molds. For the successful assembly the parts need to be accurate. The adhesive layer provides means for some tolerances. In this work the nominal adhesive offset of 2.0mm between the composite panels was used. During the design process the thickness of the panels was determined more accurately and the original frame geometry was modified accordingly in some regions.

The shape of the structure is so arbitrary and the load distribution is so complex that it is difficult to exploit the advantage of unidirectional fibers. It would be possible to achieve higher strength-to-density ratio with unidirectional fibers but in this type of structure it is unnecessarily complicated. Bidirectional fibers with twill weave form were therefore selected for the structure. This allowed for an easier design process. Bidirectional twill weaves bend and drape easily around the double-curved shapes as well.

Mainly low temperature prepregs were used in the manufacture. The laminate stacks were debulked to remove air from the laminate using a vacuum. Some of the parts like the base panel were made with infusion. The curing was made in an oven and the final assembly using jigs.

3: FE model

Modern FEA packages provide handy tools to convert a CAD model to the FEA model. For example, meshing and contact detection have been very much automated. This is very valuable for the user, but on the other hand, the user needs to do his homework properly so that he or she knows what is going on.

In the Concept Car project bonded joints are used as the primary joining method. Contact elements provide means to model bonded joints (see Figure 2). The frame assembly includes tens of individual contacts and, therefore, automated contact detection cannot provide the desired solution without extra consideration. In this project the automated contact detection was applied for restricted parts at a time. Still, manual modification was needed since for some parts the shape is so irregular that in some regions the contact surface was the top side of the part and in other regions it was the bottom side. Bonded contacts were modeled with pure penalty method. In this method the contact pair nodes are connected with contact springs. The default values for the contact stiffness are internally computed by the software using the stiffness of the parent elements. The internal contact stiffness can be scaled by the user using a multiplier. Benchmarking models were created and tested to define a representative multiplier. The benefit of using pure penalty method is that no additional degrees of freedom are added to the model. The downside is that bonded contacts, when modeled with pure penalty method, need iterative solution. However, the number of iterations was forced to one, i.e. the linear static analysis approach was used.

The wheel suspension systems are attached to the main frame via front and rear sub frames. These sub-frames were idealized with constrained equation (see Figure 1). For example, a rigid element was assumed between the points A and B. In the simulation this rigid element was replaced by the multipoint constraint (MPC). All six degrees of freedom at both points were assumed to be the same. This type of approach adds so-called Lagrange multipliers in to the system of equations and increases the model size, respectively. Also, other types of MPC's where included in the model where the number of constrains was less than six. An example of such a joint is a spherical joint. Referring to Figure 1, point B is a node for the constraint equations with node A, but at the same it is a so-called pivot node for other contact pairs. It forms MPC's between all nodes of the shell elements that are overlaid on the blue surface highlighted under "B". Therefore, this blue region can be considered a kind of hard mounting point in the frame. Indeed it is very stiff as aluminum reinforcements are assembled there.

Automated meshing with an average element size of 28mm was used. This was determined from the convergence studies of the deformation for the various load cases. It was understood that this was not sufficient for the accurate strain and stress recovery. Nevertheless, the focus was in the preliminary design of the Concept Car and trade-off was needed with respect to the model size so that the computing time was reasonable. The number of shell and beam elements was 66869 and 367, respectively. The number of bonded and MPC contacts

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

was 78337 and 28, respectively. The total number of elements and nodes was 145601 and 67647, respectively.

Five major load cases were identified and those were used in the design-optimization process. The first one determined the torsional stiffness of the vehicle. This is an important characteristic since it has impact on the driving comfort and the performance of the vehicle. Based on the benchmarking against the previous E-RA project at Metropolia and the general stiffness of serial production city cars [3], the torsional stiffness of the frame was defined to be at least 15 000 Nm/deg in the pre-design phase. The load was applied by setting point forces of ±8000N at the right and left hand side front wheel locations (see Figure 1). Boundary conditions were set so that over constraining was avoided. The required stiffness is obtained while the vertical displacement at load introduction point stays below ±10mm. The resultant displacement of the vehicle when using 2mm thick aluminum for each part is shown in Figure 3. Typically, before the composite design work, the model is verified using an isotropic material. One should pay attention to what kinds of structures are used in that phase, too. For this specific Alu-structure the solution was obtained but with initial composite designs the solution failed. The problem areas were the specific corners of the model where two parts were not in contact (Figure 3). Visual inspection works normally fine but this time it was quite difficult to find the defects. When moving to more flexible composite designs, a threshold was exceeded and the solution failed. One should always pay attention to the solver warning messages, since they give valuable information related to the defects of the model.

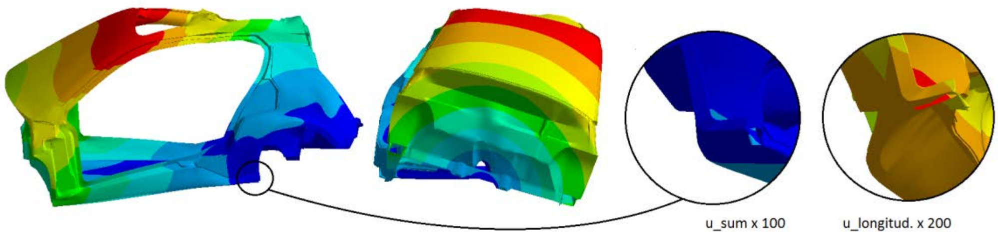

Figure 3: Deformation of the vehicle under torsional load case is shown for the two projections. Displacement scaling factor is 100. A detailed view from the rear end corner is shown. For the same detail a contour plot showing the longitudinal displacement is shown in the right most detail with a scaling factor of 200. This plot revealed a problem with the contact.

The other four load cases where related to driving conditions. Two of the cases simulated the braking conditions and two others a so-called cornering situation. The load cases were defined according to [3]. All driving loads were studied as a steady state load. While converting the dynamic loads to quasi-static loads, empirical “excitation” factors were used in each direction, respectively [3].

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

4: Composite pre-design

After the model verification is done the composite pre-dimensioning can be started. E-RA's monocoque frame consisted of CFRP and thus the obtained knowledge in the E-RA project could be used as a reference.

The modern FEA packages include specific tools for the composites design and analyses. The modeling approach mimics the actual manufacturing procedure where reinforcements are laid on the mold. Before the virtual lamination, the model needs to be prepared. The user needs to know what the program does and vice versa. Figure 4 highlights the specific areas of the base panel where reinforcements are laid on. These include the whole base panel, flat regions of the base panel and some local areas. Already in the conceptual design phase it was determined that the base panel will be a sandwich structure with skin laminates and a core ply. The area indicated on the left is related to the skin laminates and the areas in the middle are related to the core. These geometric items were already defined in the CAD model. For composite designs it is typical that local reinforcements are needed, but the application areas are not known beforehand. First design iterations reveal the places. This type of local regions can be specified afterwards using geometric rules, for example. The initial selection can be the whole base panel from which the elements that reside inside the sphere with the specific origin and radius are re-selected. The areas indicated on the right are related to the local reinforcements that were determined after the first few design iterations. Nowadays, this type of composites modeling is mesh independent for the user. Definitions and selections are based on the names of the geometry items.

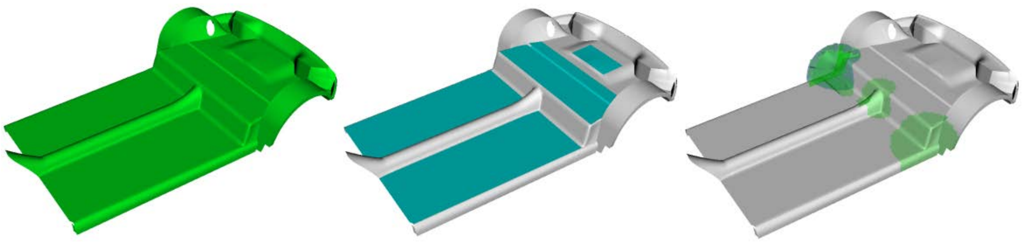

Figure 4: Application areas for the different parts of the base panel: skin laminates (on the left), core ply (in the middle) and local reinforcements (on the right).

In Figure 5 composites modeling related issues are explained more precisely. When the user applies plies on the model, the build-up direction must be known. This is illustrated with the Orientation vector. The underlying shell element has the direction for the Normal, which is determined by the node connectivity order of the element. Eventually, when the lay-up data is passed to the solver, the stack is defined in the coordinate system defined by the nodes of

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

the shell and the shell normal. The Reference direction of the ply needs to be set. This is the direction, which defines the zero direction of the ply. Various techniques to define the reference direction exist, for example, mapping of several coordinate systems or edge follower. In the lay-up phase plies are set in various orientations. Orientations are determined with the Fiber direction and it is specified with respect to the Reference direction. The example shown in Figure 5 illustrates the lamination order: first the Top ply is introduced, then Core and local reinforcement plies and finally the Bottom ply. The build-up direction is determined by the Orientation vector.

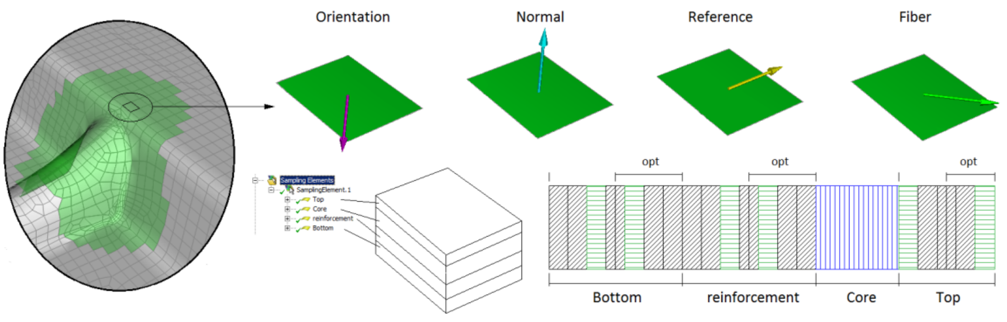

Figure 5: Different coordinate systems applicable in the composites modeling are shown at top. The illustration of the internal structure of the single element is shown on bottom.

It is not necessary to introduce plies one by one to the model, which can be really laborious. A so-called sub-laminate can be built in a lay-up tool and then be assigned as a single modeling ply to the model. In Figure 5 internal structures of the different sub-laminates are shown. For example, the bottom sub-laminate has seven plies if we count the two half plies in the middle as single ply, like it is in reality. Plies oriented in 45 degrees are illustrated with black color and plies oriented in zero degrees are illustrated with green color. A sub-laminate can also be a single ply like in case of the core ply.

The initial composite model is constructed area by area using the ply-based modeling approach and utilizing the possibility to apply sub-laminates. Before the solution, the FEA tool internally changes the ply-based model to the zone-based model and assigns new properties based on the layered structure for the shell elements, which initially had the isotropic material composition.

Design criteria were considered as a parallel task. The driving load cases were related to rather extreme situations and, furthermore, the design loads were multiplied by a Factor of Safety (FoS) of 2. For the torsional stiffness study FoS of 1.2 was applied. The load condition is clearly defined and typically displacement results predicted by the simulation are sufficiently close to the experimental measurements.

For the material systems applied in the project only limited test data was available. Mainly CFRP systems were used. For example, for the twill weave Young’s modulus of 62GPa was assumed in the principal 1 and 2 directions, respectively. In-plane shear stiffness was assumed to be 4GPa and out-of-plane shear stiffness 3.2GPa, respectively. A single clear design criterion was needed. When considering high cycle fatigue of composites, the author has considered 0.4% strain level as a critical limit [4]. It was decided to use this strain limit as an acceptance criterion in the plane of the laminate for CFRP’s. Respectively, in the out-of-plane direction 12.8MPa was used as the critical shear stress limit. This stress corresponds to the 0.4% constant shear strain.

Also new material systems were used in the project. Due to the nature of the project these systems cannot be revealed for the moment, unfortunately. A conservative approach was used in terms of the design criterion. For other reinforced material systems, referred here as GFRP, strain limit level of 0.3% was used and interlaminar shear strength of 2.5MPa, respectively.

First design studies were performed and post processed. In the post processing appropriate failure criteria are used. In this project the maximum failure strain criterion was used in the plane of the laminate. Simultaneously, maximum stress 3D criterion was used. For this criterion only the out-of-plane shear stress components were considered. Failure analysis results for the final design are graphically presented in Figure 6 under the specific driving load. The failure analysis considers all plies of the model in each direction. The contour plot is a kind of piercing though the structure and showing the results determined by the most critical ply at each element.

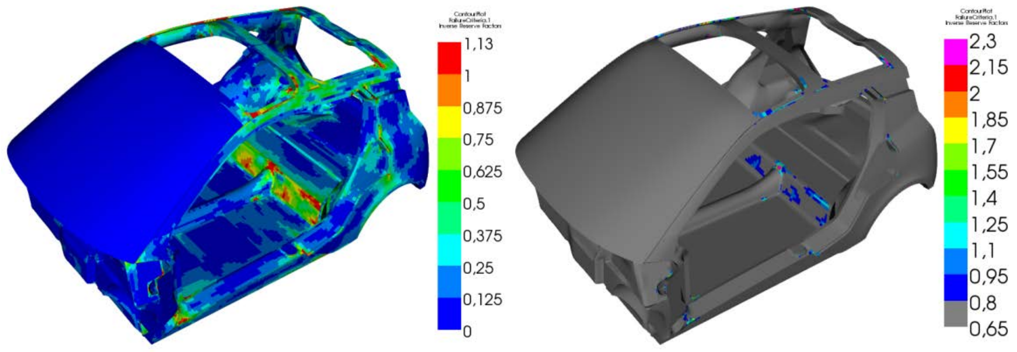

Figure 6: Graphical presentation for the failure analysis results. The result item shown is the inverse reserve factor (1/RF). A value of 1 means that the strains in the laminate are at maximum in the level of 0.4% or 0.3% and out-of-plane shear stresses are 12.8MPa/2.5MPa at maximum depending on the structure concerned. Consequently, a value of two means that the strains and stresses are doubled. Default legends are used on the left. For the image on right modified threshold values have been used both on the lower boundary (0.8) and upper boundary (2) for better visualization.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

Judgment related to the results of the preliminary design was made visually and for a limited set of designs. In the design-optimization process numerical output needs to be extracted. It does not make sense to post process results for elements that potentially give poor results since it would lead to over-dimensioning. The element shape checking feature of the FEA tool was used with default settings to identify elements that exceeded the warning and error limits. These elements were excluded from the post processing. Such elements are shown with yellow color for a section of the model in Figure 7. Removal of the elements did not lead to the desired solution. Still, there were lots of triangular elements that are known to be excessively stiff and give poor results. An example of such element is shown in Figure 7 with red color. This element gave much higher stresses than its neighbors though it was not in the specific stress concentration location. Also, hot spot elements were excluded close to discontinuity locations. The idea was to keep tracking elements necessary for the optimization dispersed throughout the structure so that all true weak points of the model could be considered. The total number of the shell elements in the model was 66869 out of which 2460 belonged to the windshield, which was not considered in the failure analysis of the design-optimization process. 2117 elements were excluded as a result of the shape checking and 537 elements were manually characterized as bad elements based on their shape, size or location.

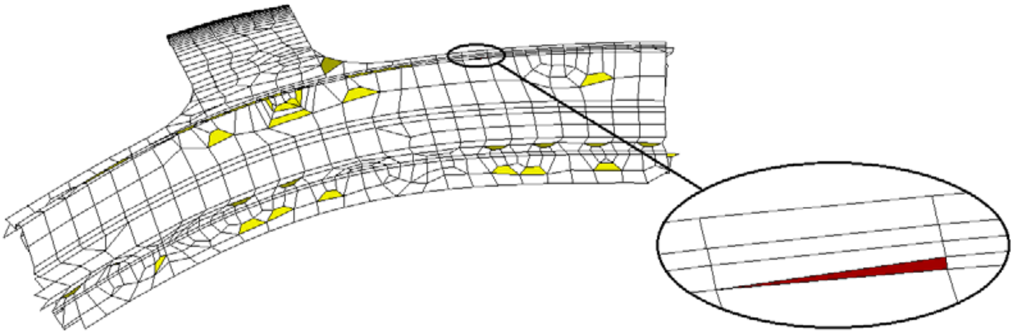

Figure 7: Failure analysis was made for the tracking elements shown in white color. Shape checking was used to discard yellow elements. Red elements were manually rejected.

In the failure analysis the frame of the vehicle comprised 16 groups where each group represented a part of the structure like the floor or the inner frame, for example. For each group the most critical element determined if the certain design passed the specific load case or not. After the first design-optimization runs it became obvious that meeting the target weight budget required loosening of the initial design criteria. Safety was included both in the load specification and material allowables. However, both material and load specifications were kept original. Instead, for each group the original acceptance level of one for the inverse reserve factor was reconsidered. After re-evaluation this number was set between 1.2 and 2.2 for all groups except for the floor for which the value of 3.0 was allowed. The stress concentrations of the floor were further investigated during the detailed design phase.

5: Design-optimization

Generally optimization aims to the selection of the best, or a set of best suited designs, with respect to one or more objectives and a set of constraints. For engineering purposes the optimization process and its results can be much more valuable than just finding the minimum or maximum of a function. Gaining deeper insight in the behavior of the model and identification of the design drivers and critical elements are a few of the possible benefits. Knowledge on the dependence of a certain output variable, such as reserve factors or displacements, on certain input variables, such as layer orientation and thickness, was also important in this project when designing details and making decisions for the final body configuration.

Based on the given goals and requirements combined with the knowledge of the composite pre-design phase, the optimization design loop was set up. The loop consists of five major parts: input variable definitions and lay-up creation, laminate export in FE understandable format, FE calculations with applied lay-ups, output files and variables, as well as the optimization algorithm which evaluates the designs and creates new input variables (see Figure 8).

After the pre-design the possible lay-ups were decided for each sub-laminate, as shown in Figure 9 (table), so that feasible designs could be found and there was enough potential to fit the weight budget. Owing to the limited computation resources and the time frame of the project, the amount of options for each sub-laminate had to be reduced. Thus, only main orientations of 0° and 45° were used. However, due to the equal stiffness and strength properties of the twill in principal 1 and 2 directions, the combination of those two orientations are a practical selection for this problem.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

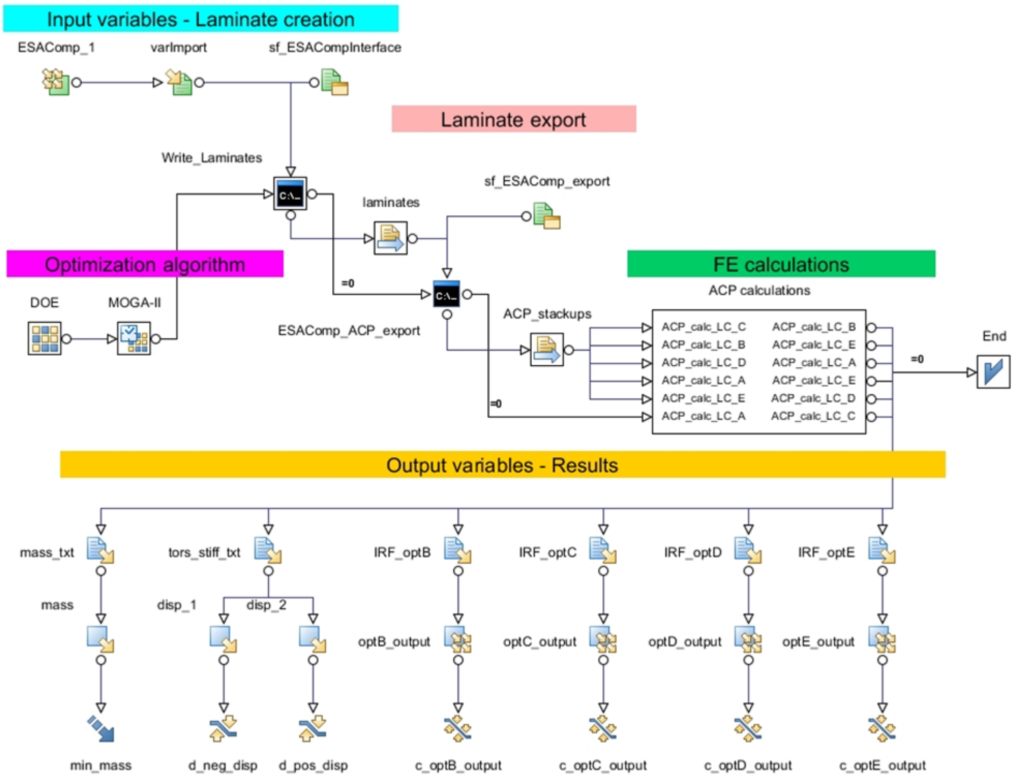

Figure 8: The optimization workflow used in the project.

	orientation	0	45	0	45 and 0	45 or 0	45	SE		
	ply thk	1	1	1	1	0.5	0.5			
Sub-lam	**Material**	**n1**	**n2**	**n3**	**n4**	**n5**	**n6**	**n_min**	**n_max**	**NOTE**
1	CFRP_1	1,2,3	0,1	0	0,1	1	0	3	13	
2	CFRP_2	0,1	0,1	0	0,1	1	0	1	9	
3	CFRP_3	0,1	0,1	0	0,1	1	0	1	9	similar to 2
4	CFRP_4	0,1	0,1	0	0,1	1	0	1	9	similar to 2
5	CFRP_5	1,2	0,1	0	0,1	1	0	3	11	
6	CFRP_6	1,2,3	1,2,3	0	1,2	0,1	0	8	21	
7	CFRP_7	1,2,3	0,1	0	0,1	1	0	3	13	similar to 1
8	CFRP_8	1,2	0,1,2	0	0,1	0,1	0	2	13	
9	CFRP_9	1,2	0,1,2	0	1	0,1	0	6	13	
10	CFRP_10	1,2	1,2	0	0,1	1	0	5	13	
11	CFRP_11	1,2	0,1,2	0	1	0,1	0	6	13	similar to 9
12	CFRP_12	0,1,2,3	0,1,2	0	0	0	1	1	11	
13	CFRP_13	0,1,2,3	0,1,2	0	0	0	1	1	11	similar to 12
14	CFRP_14	0	1,2	0,1,2	0,1	0,1	0	2	13	
15	CFRP_15	0,1,2,3	0,1,2	0	0	0	1	1	11	similar to 12
16	CFRP_16	1,2	0,1,2	0	0,1	0,1	0	2	13	similar to 8
17	CFRP_17	1,2,3	0,1	0	0,1	0,1	0	2	13	
18	CFRP_18	1,2	0,1,2	0	0,1	0	0	2	12	
19	CFRP_19	1,2	0,1	0	0,1	1	0	3	11	similar to 5
20	GFRP (reinf. side)	1,2	0,1,2	0	0	0,1	0	2	9	
21	GFRP	1,2,3	0,1	0	0,1	0,1	0	2	13	similar to 17, different material
22	GFRP **(reinf. mid)**	0	**0,1**	**0,1**	**0,1**	**1**	0	1	**(7)** 9	
23	GFRP **(top)**	**1,2**	**0,1**	0	0	**0,1**	0	2	**(5)** 7	
24	GFRP **(bot)**	**0,1,2,3**	**0,1,2**	0	**1**	**0,1**	0	4	**(7)** 15	
25	GFRP	0	1,2	0,1	0,1,2	0,1	0	2	15	inactive in simulation
26	CFRP_2	1,2	0,1,2	0	0,1	0,1	0	2	13	similar to 8, different material
27	CFRP_3	1,2	0,1,2	0	0,1	0,1	0	2	13	similar to 8, different material
28	CFRP_4	1,2	0,1,2	0	0,1	0,1	0	2	13	similar to 8, different material

Figure 9: Possible layups for the sub-laminates of the Concept Car body. For sub-laminates 22 to 24 the bold values refer to the final design. Sub-laminates 22-24 are also illustrated with the corresponding names in Figure 5.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

Defining the range of all 110 input variables which are necessary to create the 27 dependent lay-ups manually in the optimization environment would have been very laborious and error prone. Therefore a different approach was used. The potential layups are defined in a proprietary ESAComp XML code which allows for both definition of the variable ranges and creation of lay-ups from a set of given input variable values. The variable ranges can be easily imported to the optimization environment and, vice versa, a set of values for a certain design transferred to lay-ups. Additionally, features like balance or symmetry of a sub-laminate can already be defined in the XML code. Sets of input variables for similar sub-laminates can be created at once, which means less coding and debugging work.

Each lay-up in Figure 9 can be written in the form of

[n1 x 0 / n2 x 45 / n3 x 0 / n4 x [45 / 0] / (½) n5 x 45 or 0 / (½) n6x 45]s (1)

with layer multipliers n1 to n6. A multiplier vector $\vec{n}$=[0,1,0,1,1,0] for sub-laminate 24 in Figure 9 would thus result in [45 / 45 / 0 / (½) 45]s. The subscript s denotes symmetry and the indicator (½) means that a half thickness layer is used. Whether a 0° or 45° layer is multiplied with n5 is decided with an extra variable. With the above equation it was possible to include the stacking sequence optimization into the laminate lay-up formulation. Based on the pre-design, a single sub-laminate included three or four active variables and the rest were set to zero. The purpose was to guarantee that both layer orientations existed in each sub-laminate.

In the next step the lay-up information is passed to the FE models. After the first step a list of layers with only material name and orientation existed for each lay-up. With the help of the batch capabilities and FE interfaces of ESAComp, which also served as the material database, this list was translated for the FE tool containing the necessary density, stiffness and strength properties of each material as well as the stacking sequences of the sub-laminates. Then the load cases were calculated and values of output variables transferred to the optimization framework.

The goal of the optimization was to minimize the weight while meeting the constraints. Mathematically the problem can be described as single objective optimization

$$\min_{\vec{x} \in S} m(\vec{x}) \qquad (2)$$

with the vector of design variables $\vec{x}$, the mass of the frame m and the feasible set S defined by a number of equality and inequality conditions $S = \{\vec{x} \mid \vec{g}(\vec{x}) \leq 0, \vec{h}(\vec{x}) = 0\}$. The torsional stiffness was determined by displacements u_1 and u_2 incorporated as $-u_1 - 10 \leq 0$ and $u_2 - 10 \leq 0$. The strength related constraints of the body were implemented based on the inverse reserve factors as

$$\frac{1}{\overrightarrow{RF_i}} - \frac{1}{\overrightarrow{RF^c}} \leq 0 \qquad (3)$$

where $\overrightarrow{RF_i}$ is the vector of reserve factors for i=2,…,5 load cases and $\overrightarrow{RF^c}$ is the vector of critical reserve factors which contains the critical values for all 16 groups. During the optimization, which utilized mass, displacements and inverse reserve factors, also element number, reason for failure (stress/strain component) and failing layer were kept track of for each of the groups and load cases. Different body configurations could be compared at certain level of detail.

The last part is the optimization algorithm, also called the scheduler. The optimization problem contains discrete variables, so the chosen algorithm had to be suited for that. In previous projects of the authors [5, 6, 7], the performance of several algorithms like Particle Swarm Optimization, evolution strategies and genetic algorithms, was compared. From that experience the multi-objective genetic algorithm II (MOGA II) provided by the optimization tool was chosen for this project as it has reliably provided good results for similar problems.

Genetic algorithms use an initial set of designs (generation) and create offspring generations based on the fitness (derived from objective values and how well constraints are met). The input variables of a design can be compared to genetic code of an animate being. Different operators are used on this code to create new designs:

- Cross-over: Properties of two designs are combined.
- Selection: The design is used unchanged in the next generation.
- Mutation: Part of the input variables of a design of the parent generation is randomly changed within the range of the input variables.

The first generation is the starting point for the optimization. Thus it should provide diverse designs for the scheduler to create further generations from and not get stuck in a local optimum. A rule of thumb for the size of an initial generation is to use twice the number of input variables multiplied with number of objectives [8], which gives 220 designs for the car body. However, from experience it was known, that the minimum required generation size does not scale linearly with the amount of design variables and for a high number of input variables lower values can be chosen. As can be seen in Figure 10 several generations are needed before designs converge towards the goal. Keeping that in mind and considering the time frame of the project, the initial generation consisted of 75 designs.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

The algorithm for creating the designs was a random design of experiments. Therefore, the design space is filled randomly, which usually results in diverse designs, but has the risk of clusters because such are not actively prevented. Uniform Latin Hypercube Sampling could be an option to investigate in future projects. Designs are also created based on random sampling, but in a way which leads to a more uniform distribution over the design space.

6: Results

The design-optimization environment provides various tools to follow the process. The mass-history plot shown in Figure 10 is one of the very basic output forms, which indicates how the mass of the frame developed as a function of the design ID. The results are in line with our previous optimization projects of the same size. It takes 25 to 30 generations to reach the leveled state. In this project more generations were needed since the first feasible designs were obtained only during the 4th generation. If all aspects of the problem have been properly considered, the optimization process produces results that require only very little post processing. If there are several equally light feasible designs, the selection can be based on the torsional stiffness of the body, for example.

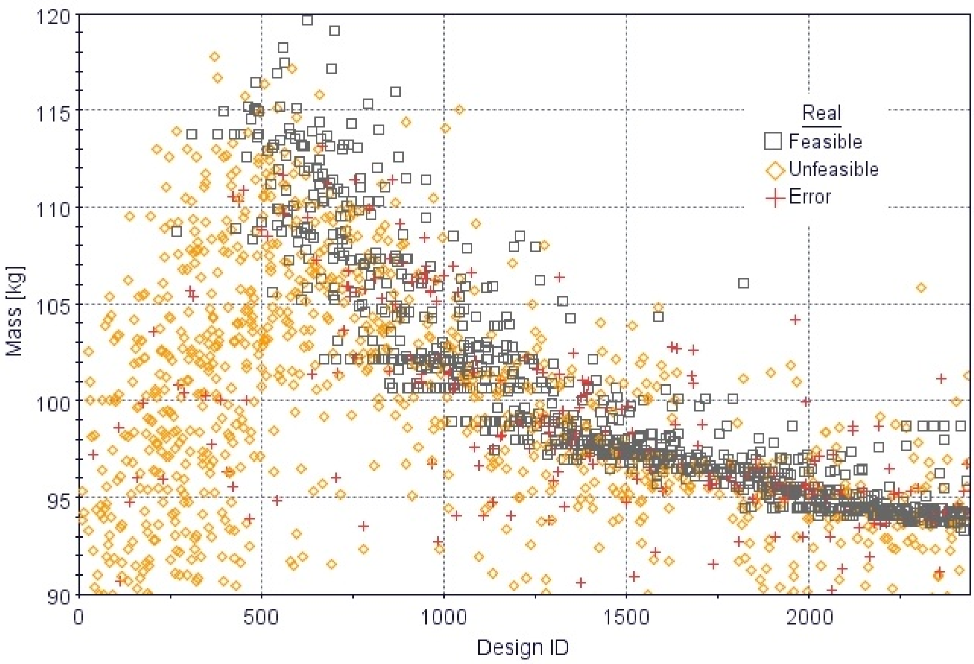

Figure 10: The mass-history of the process indicating feasible, unfeasible and error designs. For unfeasible designs at least one constraint is violated.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

Detailed post-processing is always needed. The specific design can be thoroughly evaluated after reading the existing lay-up information. The thickness of the laminate at different locations of the model can be visualized, for example. Typically the solution needs to be re-run since the result files are not saved during the process. This is required for the detailed post processing of strains, stresses and failure analysis results. Detailed post processing is made for the hot spot elements before the design is accepted (see Figure 11). Ply-based modeling approach guarantees that the step from the simulation to manufacturing is smooth.

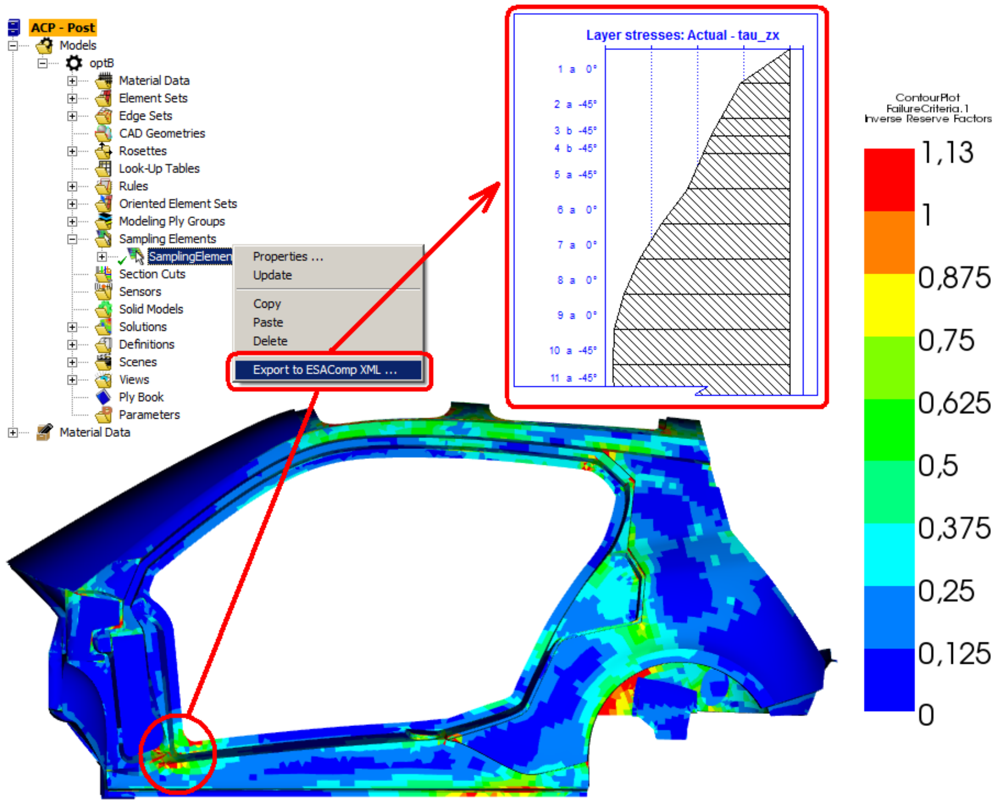

Figure 11: Detailed post processing for a hot spot element is made in ESAComp, which provides a suite of tools for design, analysis and reporting.

7: Conclusions

For complex composite structures the number of design variables easily builds up and the best performing solution cannot be found just relying on the engineering expertise and intuition. From the beginning it was clear that an environment that simultaneously provides the integration of the simulation

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

tools and the optimization capabilities was needed. Design-optimization of composite structures with genetic algorithms can utilize parallel computation, but this option was not possible in this project. Limited computational resources were compensated by defining what is practical by focusing on the most relevant issues related to a workable solution. The number of design variables was constrained to the level of 100. The number of design variables reflects the number of designs needed for the reliable optimization. A decent preliminary design before the optimization is very important. Limits for the design variables can be reliably estimated and feasible designs are obtained just from the beginning. This project was a learning process and the preliminary design phase could have been made in a better way. In this project we started to obtain feasible designs regularly only after the first 500 designs.

The size of the FEA model in terms of the number of nodal degrees of freedom and constrained equations was limited to the practical level so that the solution time for a single run was in the level of five minutes. The size of the FEA model reflects to the level of details that can be included in the model. Figure 12 shows a detailed view for a part of the geometry model and the associated FE mesh. A radius of 25mm is generally idealized with two bi-linear shell elements. Such idealization increases the disturbance forces at the edges of the elements. On the other hand, large elements are not capable of capturing stress concentrations.

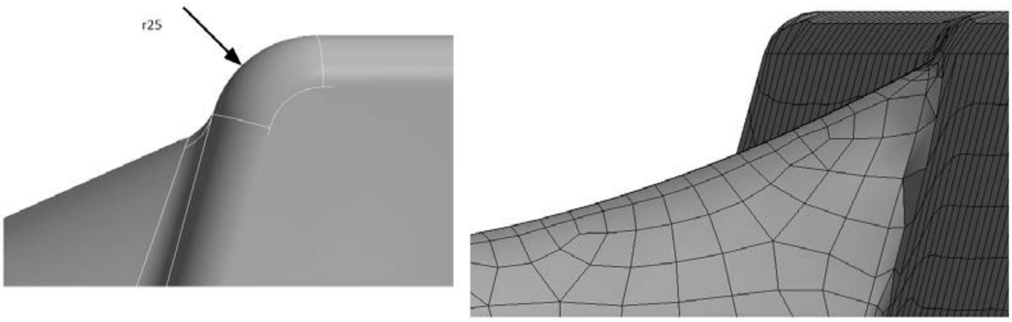

Figure 12: A detail of the geometry model and associated element mesh illustrating the mesh density.

The purpose of this project was to find the best performing feasible solution in the preliminary design phase using design-optimization. The issues related to the stress concentration could be better assessed in the detailed design phase. This topic was out of the scope of this study. Five necessary load cases were identified, i.e. solving a single design took roughly 30 minutes. After running 2500 designs the problem had converged mass of the frame being below 95kg. From the experience of previous similar projects it could be assessed that mass savings in the scale of 15% were reached while comparing to the traditional hands-on approach. Certainly, this achievement paid back the effort needed in setting up the system.

8: Acknowledgements

ANSYS, Inc. is acknowledged for providing the FEA tools for the project through their Scandinavian distribution partner EDR-Medeso and through CSC - IT Center for Science Ltd.

EnginSoft S.p.A. is acknowledged for providing modeFRONTIER, a multidisciplinary and multi-objective software for the design-optimization, for the project.

DESIGN OPTIMIZATION OF A COMPOSITE CAR BODY

REFERENCES

1 Summaries of EU legislation, http://europa.eu/legislation_summaries/environment/waste_management/l21225_en.htm

2 E-RA homepage, http://www.raceabout.fi/era/

3 Hapian-Smith J., An Introduction to Modern Design Vehicle, p. 125-155, Butterworth-Heinemann, 2000, ISBN: 0 7506 5044 3

4 Fatigue in Composite, Bryan Harris (Editor), ISBN: 0849317673, November 3, 2003

5 Multi-Objective Design-Optimisation of Composite Structures, Katajisto H. et al., Proceedings of the NAFEMS World Congress 2007, May 22-25, 2007, Vancouver, Canada

6 Engineering Oriented Formulation for Laminate Lay-up Optimization, Mönicke A. et al., Journal of Structural Mechanics, Vol. 41, 2008, Issue 3, ISSN 1797-5301

7 Design-Optimization of Cylindrical, Layered Composite Structures Using Efficient Laminate Parameterization, Mönicke A. et al., Proceedings of the European Conference on Spacecraft Structures, Materials & Mechanical Testing, SP-691, 20--23 March, 2012, Noordwijk, The Netherlands

8 modeFRONTIER User Manual. 2003, ESTECO s.r.l. (ed.)