# AN APPROACH TOWARDS AN OPTIMAL DESIGN OF COMPOSITE STRUCTURES

B. Goller[a*], M. Gratt[a], C. Elsel[a]

[a]*INTALES GmbH Engineering Solutions, Innsbrucker Str. 1, 6161 Natters, Austria*
**goller@intales.com*

**Keywords:** Optimization, lightweight structures, composite, Finite Elements

**Abstract**

*The present paper will show the development of an efficient, fast and reliable optimizer for composite and metallic parts of lightweight structures. The algorithm aims at identifying the optimal configuration of different structural parts concerning thickness, fibre orientation, number of plies, etc. This leads to mass savings and also a decrease of the development time in the structural dimensioning phase. Due to the limited applicability of classical optimization algorithms like gradient based or evolutionary methods in case of large Finite Element models with a high number of design variable, a novel approach is presented where the optimization problem is tackled by a heuristic adaption procedure on element level. The approach will be illustrated on small numerical examples, and an outlook to the application to industrial problems will be given.*

## 1. Introduction

One main aspect for the competitiveness in the design of lightweight structures is the identification of weight saving opportunities while fulfilling multiple constraints in terms of allowable values for stresses, strains, displacements etc. The variables to be optimized are the number of plies of different parts, the orientation of the fibers and also their stacking sequence in case of composite parts. For metallic parts, the thicknesses are to be optimized, while the diameter represents the design variable related to fasteners. For real structures, the number of variables to be optimized might soon reach a limit where classical optimization algorithms, like e.g. gradient-based methods or also evolutionary algorithms, might not be applicable due to the curse of dimensionality. An infeasible high number of model evaluations would be required in order to identify a possible solution.

The topic of the here described project is the development of a novel approach where the optimization problem is tackled by a heuristic adaption process on the element level. More specifically, the starting point of the algorithm is the element stiffness matrix, which is adjusted according to the temporary state in terms of failure criteria at a certain load level. The external forces are applied in several load increments until the full load level is active and during each of these increments an adaptation process is performed. The process can be compared with the mechanical adjustment of biological materials, like wood and bones, where the stiffness formation process is determined by the acting forces.

The key aspect in this process is the access of the element routines and the development of an adaptation strategy for each element type in order to obtain optimal convergence. A number of state-of-the-art element formulations have therefore been implemented and linked with the commercial Finite-Element (FE) code ABAQUS. In order to manage the independently adjusted properties of each element, an efficient data-management is required which has been solved by connecting an SQLITE-database to the element routines, where the values of each iteration of all variables to be adjusted are written and accessed in the next iteration. Strategies for the type and magnitude of change of the distinct design variables have been investigated where especially the robustness of the solution, i.e. its sensitivity with respect to small deviations of input parameters, is in the focus of the studies.

The main advantage of this approach is its applicability to large and complex FE-models, where also several million degrees of freedom do not lead to algorithmic limitations. In addition, the strategy can be used for models involving different kinds of elements, like shell elements including layered composite, sandwich or isotropic (metallic) shell sections, and also fastener elements.

The structure of the present paper is as follows: after a brief discussion of the theoretical background in Sec. 2, the implementation of the element routines in the framework of ABAQUS with the link to the SQLITE-database is addressed in Sec. 3. The augmentation of the element routines by failure criteria and the respective strategies for adaptation are topic of Sec. 4. The quality of the employed adaptation strategies is judged by means of some small test examples.

## 2. Theoretical Background

The goal of the present project is the optimal design of lightweight structure such that the weight is minimized and all design criteria are fulfilled. This can be expressed as a constrained optimization problem, i.e.

$$\min f(x) \tag{1}$$

$$\text{subject to } g_i(x) \leq c_i \qquad \text{for } i=1...m, \tag{2}$$

where $x$ are the design variables, like thickness, fiber angle, fastener type, etc, $f(x)$ is the objective function, i.e. the total mass, and $g_i(x)$ are the constraints which are defined by failure modes, design rules and feasibility of construction. Classical optimization algorithms might soon reach their feasibility limits since the number of design variables is in the range of hundreds or even thousands for large FE-models.

Hence, an alternative approach based on an iterative adaption theory on element level is followed in this paper, which is inspired by ideas like "fiber steering" (aligning the fibers along the most effective direction), "computer aided internal optimization" [1], and "multi-domain topology optimization" [2]. More specifically, during the structural analysis the status of each element in terms of fulfilling the investigated criteria is checked and - if necessary - the values of the design variables of the element are changed. This change may imply an increase of the thickness in case of a metal, the change of the draping angle or of the stacking sequence in case of composite elements or also the increase of the diameter in case of a fastener element. In order to consider a possible load distribution when changing the stiffness of the structure due to this local adjustment, the external load is applied iteratively. In this way, the structure can "grow" according to the acting load, which is a concept inspired by mechanical adaptation of biological materials. As it also applies to wood, the grow is only additive, hence, the material is never reduced. In this way, it can be ensured that in case of multiple load cases only the critical load cases are driving for each element, while a lower external forces does not change the element properties.

The crucial point is the definition of the adaptation strategy, which has to be defined for each failure criterion. A measure of the degree to which the limit factor is exceeded for each of the criteria is the so-called reserve factor (RF), which defines the factor by which the load is increased or decreased, respectively, in order to reach the limit state given by an RF equal to 1.0. Hence, this value is a measure by which factor e.g. the thickness of a metallic element or the diameter of a fastener are to be increased. In case of multiple design variables belonging to one element or to one criterion, the strategy becomes more involved and has to be specific to each case. The approaches which have been developed will be described in detail in Sec. 4.

## 3. Implementation of element routines for ABAQUS

In order to implement the here proposed optimization algorithm at element level, an access to the element routine has to be provided and the element properties have to be adjusted according to the evolution of the optimization process. ABAQUS provides interfaces for the user-defined interference in the solution process [3].
The user subroutine employed for the implementation of the element routines is called UEL, where the element stiffness matrix is calculated. The mechanical and geometrical properties, which are required for the definition of the element stiffness matrix, are changed during each iteration of the optimization process. For complex Finite Element models, a large amount of data has to be handled since the current properties of each element have to be stored in every iteration and retrieved in the next iteration. An SQLITE-database provides a suitable means for this problem since data can be stored and accessed efficiently. The user subroutine UEXTERNALDB provides an interface between ABAQUS and an external database.
In order to make the optimizer applicable to a certain region of a model, the type of element used for that component must be implemented as a user subroutine. Hence, the most frequently applied element types have been coded in terms of an UEL, which are three- and four-node shell elements, a sandwich element and a fastener element.

## 4. Development of the optimization procedure

The idea behind the optimization process is the adaptation of the properties of each element according to its state with respect to the failure criteria. Hence, for each failure criterion a strategy for adjusting the element has been developed and implemented in the element routine. Based on the current values of the element forces, moments and/or stresses, the design variables of the analyzed element are adjusted.
In the following, the strategies applied to the different element types are discussed and some examples will illustrate the functionality of the algorithm.

### *4.1. Monolithic composite element*

The first kind of failure criteria which are investigated are composite failure modes. One frequently applied approach is the maximum strain criterion which will be discussed in the following. This strategy can be applied to both four-node quadrilateral and three-node triangular shell element. In addition, also for the dimensioning of the face sheets of sandwich elements this approach is employed.

### *4.1.1 Strain based criterion*

The failure criterion which has been implemented for composite elements is a strain criterion. The requirement is formulated such that the maximum strain occurring in an element must not exceed a user-defined threshold, which is usually in the range of 0.3% to 0.4%. In case the requirement is not fulfilled, the following adaptation strategy is applied: section forces and moment vectors as well as their principal directions are calculated at the element centre. For each of the four directions, the minimum number of plies is determined to meet the unidirectional criterion. From these numbers, the dominating direction, i.e. the draping angle, and the required moment of inertia of the cross-section are identified. Further, a grouping parameter $g$ is calculated from the degree of anisotropy of the stress state. The proposed stacking then follows a strict sequence, $45/90/\text{-}45/0_g$, which is repeated until the required membrane and bending stiffness is obtained.
The approach will be illustrated using quadrilateral monolithic elements. For triangular elements or also the face sheets of sandwich elements, the same strategy can be applied.

### *4.1.2 Numerical example: plate with hole*

The first example used for illustrating the method is a plate with hole under tension load as it can be seen in Figure 1. The structure is clamped at the left end and loaded with constant point loads of 2000 N in positive x-direction at the right edge. The plate is made out of composite (fabric), where the initial configuration is given with 4 plies (45/0/0/45) with a ply thickness of 0.28 mm. The Young's modulus in both directions is given with 62000 N/mm$^2$.

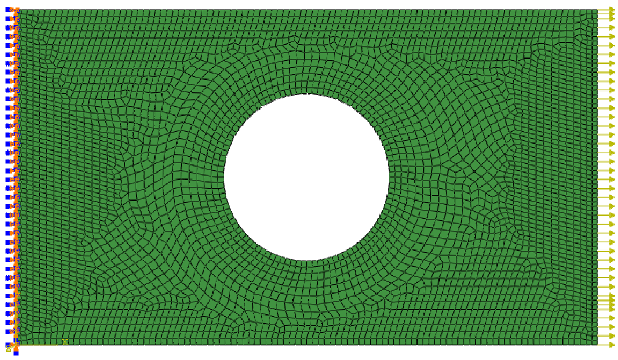

**Figure 1.** FE-model of the plate with hole under tension

The design variables are the number of plies and the draping angle, where the values after the optimization process are shown in Figures 2 and 3. The direction of the fibres is optimal if it is aligned with the direction of the main load path. Since a plate with hole under tension is a well known benchmark example, the direction of the principal stresses is known from analytical solutions. This direction can be recognized also in Figure 2, where however, the curvature of the directions around the hole is not as pronounced as in the analytical solutions. This can be explained by the fact that the material is not isotropic due to the fibres and hence the optimal direction is not equal to the draping angle. The draping angles of the single elements as indicated in Figure 2 represent therefore a plausible solution.
The number of plies is shown in Figure 3 for each element of the model. As it is known from analytical solutions, the highest stresses are located at the top and bottom of the hole. these are also the two regions where the highest thickness increases are performed during the optimization process. the highest value for the number of plies has been set to 50 (user input

prior to the analysis) and this is also the value which has been reached by the elements in these areas. The definition of a maximum values for the plies is motivated by the situation that an increase of the number of plies provokes a higher attraction of loads to this area (due to the higher stiffness) and hence an unrealistic thickening would occur around stress peaks. By limiting the number of plies to a user defined value, a certain re-distribution of the stresses around the location of the peak values can be obtained.
The increase of the number of plies in the two regions from the hole to the support reflect the direction of the forces in the structure. Also at the support a certain increase can be seen due to the higher stresses.

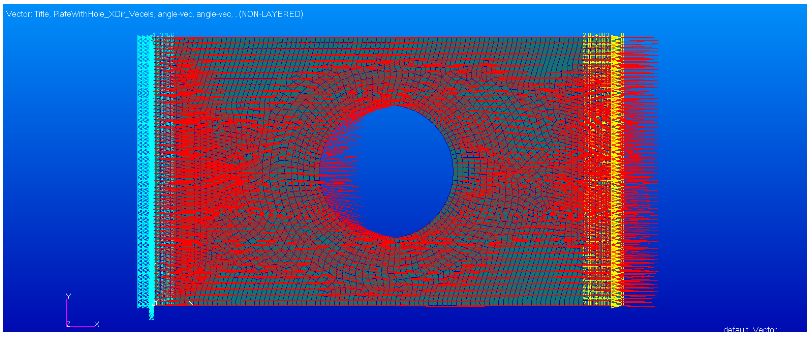

**Figure 2**: draping angle after optimization

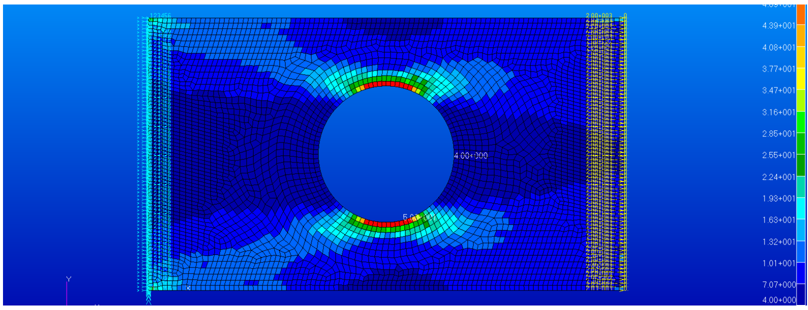

**Figure 3**: number of plies after optimization

*4.2 Fastener element*

The second kind of failure criteria which are investigated are fastener related failure modes. More specifically, three criteria concerning bolt failure due to axial and shear load, pull through failure and a bearing criterion are implemented and discussed in the following.

*4.2.1 Combined Failure Criterion*

The combined criterion is formulated as follows:

$$\left(\frac{\sigma_n}{\sigma_{n,allowed}}\right)^2 + \left(\frac{\sigma_s}{\sigma_{s,allowed}}\right)^2 \leq 1, \tag{3}$$

where $\sigma_n$ and $\sigma_{n,allowed}$ are the acting and allowable normal stresses of the fastener, and $\sigma_s$ and $\sigma_{s,allowed}$ define the acting and the allowable shear stresses. Based on this criterion, the following reserve factor can be calculated:

$$\left(\frac{RF\cdot\sigma_n}{\sigma_{n,allowed}}\right)^2+\left(\frac{RF\cdot\sigma_s}{\sigma_{s,allowed}}\right)^2=1 \quad \Rightarrow RF=\frac{1}{\sqrt{\left(\frac{\sigma_n}{\sigma_{n,allowed}}\right)^2+\left(\frac{\sigma_s}{\sigma_{s,allowed}}\right)^2}} \tag{4}$$

In case the reserve factor assumes a value smaller than 1.0, the diameter of the fastener is increased such that RF≥1.0. According to the theoretically necessary fastener diameter, the diameter with the next larger size which is available for the selected fastener type is chosen.

*4.2.2 Pull-through failure criterion*

The second criterion which is analyzed is the pull through criterion, where the stresses acting in the area between the fastener head/nut and the base material of the connected plates are compared with the allowable pull-through stresses. The acting stresses are given by:

$$\sigma_{PT}=\frac{P}{\frac{1}{4}(D-d)^2\pi}, \tag{5}$$

which is calculated for both nut and head side. The reserve factor is obtained by comparing the acting stress with the allowable stress, i.e.

$$RF=\frac{\sigma_{PT,allowed}}{\sigma_{PT}} \tag{6}$$

In case this comparison yields an RF smaller than 1.0, both fastener diameter and number of plies are modified such that the criterion is fulfilled. The diameter of the fastener which has been determined in the previously discussed combined criterion is thereby taken as a starting value.

*4.2.3 Bearing Failure Criterion*

The third criterion which is analyzed is the bearing criterion. For this reason, the stresses which are transferred from the collar of the fastener to the plate material, are compared with the allowable bearing stresses. More specifically, the acting bearing stress is given by

$$\sigma_{BEA}=\frac{P_{shear}}{d\pi t_0}, \tag{7}$$

where $P_{shear}$ is the shear force acting in the fastener element, $d$ is the diameter of the fastener, and $t_0$ denotes the thickness of the thinnest connecting plate. In order to compute the reserve factor, this stress is compared with the allowable bearing stress, i.e.

$$RF=\frac{\sigma_{BEA,allowed}}{\sigma_{BEA}}. \tag{8}$$

Analogously to the pull-through criterion, the required fastener diameter and the thickness (i.e. number of plies) are calculated in case of an RF value smaller than 1.0. The fibers are oriented such that the draping angle is aligned with the direction of the shear force.

### *4.2.4 Numerical example*

The example employed for testing this procedure is depicted in Figure 4. The model consists of about 1400 shell elements (4-node, quadrilateral element), where all parts are made out of composite. The material of the upper and middle part is tape, while for the lower part a fabric is used. The material names are HTS_977-20_268 (thickness of 0.25 mm), IMS_RTM6_268 (thickness of 0.25 mm) and AS4C_M21_285 (thickness of 0.37 mm), respectively, where the initial layups are defined by 8, 10 and 4 plies for the upper, middle and lower plate, respectively. The model involves a total number of 6 fasteners, where there are three single-lap shear fasteners connecting the upper with the middle plate and three connecting all three plates (double-lap shear fasteners). Their locations are indicated by the origin of the coordinate systems in
Figure 4**:** , where the orientation is perpendicular to the plates for all fasteners. For the three double-lap shear fasteners one type, namely prEN6115K/ABS1738K (bolt/collar) with an initial diameter of 11.13 mm is selected. The types of the three single-lap shear fasteners are prEN6115B, prEN6114K and prEN6115V for the collar, where the bolt is given by ABS1738K for all three fasteners. The initial diameters are 6.35 mm (prEN6115B) and 11.13 mm (prEN6114K and prEN6115V).
The upper and middle plates are clamped on the left end and loaded with a uniform traction of 300 N/mm and 500 N/mm, respectively, on the right end. On the lower plate a uniform compression load of 50 N/mm acts along the right edge.
The design variables are the diameter of the fasteners, the number of plies and the draping angle of the elements which are connecting or surrounding a fastener (2 element rows around the fastener position).

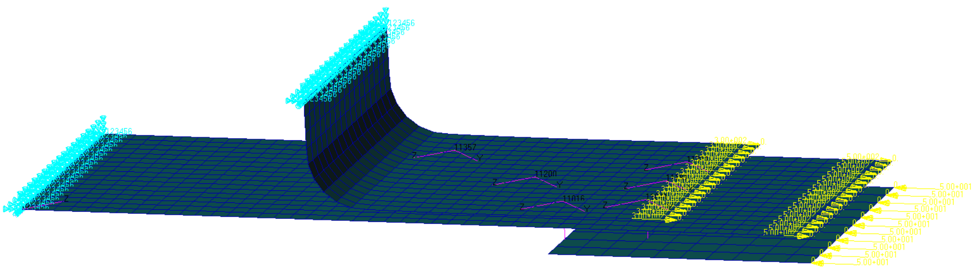

**Figure 4**: Numerical example used for testing the fastener optimizer

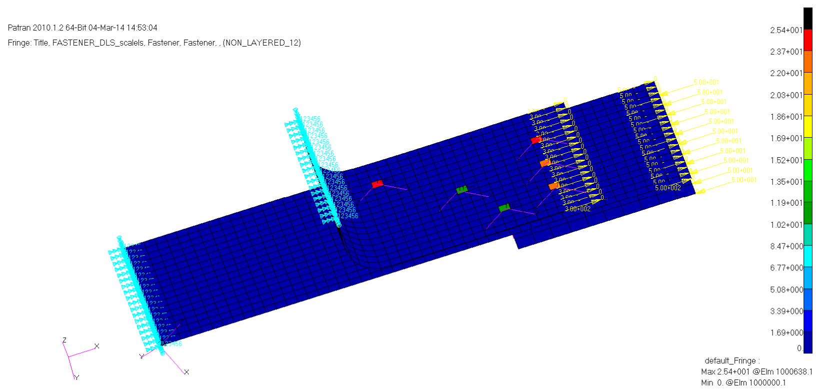

**Figure 5:** Optimized fastener diameter (illustrated by means of the closest shell element)

The diameter of the fasteners which have been determined after ten iterations can be seen in Figure 5, where the shell element closest to the fastener is colored in the respective manner. The largest diameters (25.4 mm) are required for the leftmost single-lap shear fastener and for one of the three double-lap shear fasteners. For these two fasteners, the required number of plies (depicted in Figure 6) are highest, where for the first the bearing criterion is more critical since 30 plies are needed in comparison to 19 plies needed for the second most critical location.

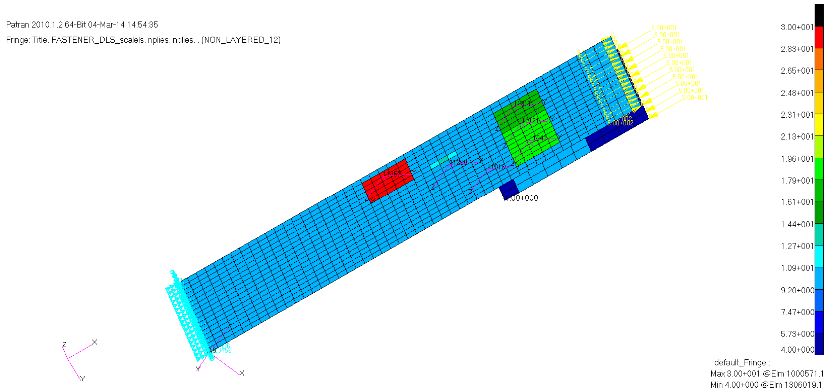


**Figure 6:** Optimized number of plies of connected and surrounding elements of middle component

## 5. Conclusions

In this paper, a novel approach for optimization of lightweight structures has been presented and applied to two illustrating examples involving composite parts and fasteners. The advantage of this approach is its applicability to FE-models involving a high number of design variables. The results which are shown in this work have been obtained after a total of ten iterations, which suggests its usability in case of large numerical models.

## Acknowledgments

This financial support of the Austrian Space Applications Programme of the Austrian Ministry for Transport, Innovation and Technology (BMVIT) is deeply appreciated by the authors.

## References

[1] R. Kriechbaum, J. Schafer and M. Mattheck. CAIO (computer aided internal optimisation): a powerful method to optimise fiber arrangement in composites. In *First European conference on smart structures and materials*, 281-284, 1992

[2] Z.D. Ma, N. Kikuchi, C. Pierre and B. Raju. Multidomain topology optimization for structural and material design. *ASME Journal for Applied Mechanics* 73(4): 565–573, 2006

[3] Dassault Systèmes. *ABAQUS 6.12 Analysis User's Manual.*