NEW DISCRETE TOPOLOGY OPTIMIZATION METHOD FOR INDUSTRIAL TASKS

Sierk Fiebig[(a)]

[(a)]Volkswagen Braunschweig, Braunschweig, Germany

[(a)]sierk.fiebig@volkswagen.de

ABSTRACT
Nowadays the development of mechanical components is driven by ambitious targets. Engineers have to fulfill technical requirements under the restrictions of reducing costs and weights simultaneously. Therefore in the last years optimization methods have been integrated in the development process of industrial companies. Today, especially topology optimization methods, have gained in importance and are standard for developing casting parts. Stress or strain-energy information is used for sensitivities in all topology optimization methods. The method SIMP, today's standard in industry, uses continuous material modeling and gradient algorithms. ESO/BESO use discrete modeling and specific algorithms depending on the individual approaches. The new Topology Optimization method uses a discrete modeling, too. The number of modified elements is controlled by the progress of the constraint.
For solving tasks in the industrial development process, a topology optimization method must enable an easy and fast usage and must support manufacturing restrictions.

Keywords: topology optimization, mechanical components, discrete modeling of material

1. INTRODUCTION

Today several approaches exist for topology optimization. The starting point of FEA based topology optimization was at the end of the eighties [Roz01]. Bendsøe introduced first his homogenization method [Ben89]. Parallel to the homogenization method, Bendsøe presented the SIMP approach (Solid Isotropic Microstructure with Penalization) [BenSig03]. This method has become popular, because other researchers use it [Roz92]. Today the SIMP approach is one of the standard methods for topology optimization. For example, the commercial tool Tosca® from FE-Design is based on SIMP. SIMP uses continues design variables. Here the density is used as design variable. The coupled Young-Modulus transfers the modifications of the optimization to the structure results. At the end of each topology optimization, a clear discrete distribution for interpreting the results is needed. Due to this, the SIMP approach penalizes intermediate density values using a penalization factor as a power. In this way, low stiffness values are assigned to intermediate density values [Edw07]. SIMP is combined to a gradient algorithm, e.g. the method of moving asymptotes [Svan87].

Since 1992 another important approach has been developed. The evolutionary structural optimization (ESO) is focused to remove unnecessary material from too conservative designed parts [Que00]. To ESO, it is only possible to remove material and uses a discrete element modeling in comparison to SIMP [HuaXie10]. To enable the opposite, Querin introduces the additive evolutionary structural optimization method, called AESO [Que00b]. AESO adds material to the highest stressed points in order to become an optimal structure. The combination of ESO and AESO is the bidirectional evolutionary structural optimization [BESO] method [Que00] [HuaXie10]. The main idea behind ESO, AESO and BESO is to remove under stressed elements and to add material to higher stressed areas. To designate these elements, two reference levels are defined. During the optimization these levels are adapted to the optimization progress.

All elements under a reference level are removed and all elements above a second level are added. BESO uses here - depending on the individual approach - direct, gradient or interpolated information about material properties to change the structure [HuaXie10].

For industrial usage the SIMP method in combination with gradient algorithm has a large distribution. One main reason for the success of the approach is the integration of manufacturing restrictions. Without these restrictions, it isn't possible, in most cases to get a feasible design for real life problems. At the moment no proposals for the integration of manufacturing restriction to BESO are offered.

2. THE NEW APPROACH FOR TOPOLOGY OPTIMIZATION

The motivation for the new approach is based on three reasons. The main focus is the usage of the method in industry. The overall interest of industry is to recognize parts with lower weight and cost compared to the older reference structure. In contrast to optimization from a mathematical or theoretical view, the task of optimization isn't to find the absolute optimum. In the opinion of engineering and praxis, optimization means the improvement of the result.

This mean, that better optimization results are the first motivation for the new method.
To achieve this and to improve the universal usage, linear and nonlinear FEA analysis should be possible with the new Topology Optimization method. Nonlinear effects are for example plastic behavior of material, nonlinear behavior in bushing and contact problems. Finally the last point, manufacturing requirements should be fulfilled.

2.1. Basic functionalities similar to ESO/BESO

Using stress or strain-energy information for sensitivities are the basic ideas in all topology optimization methods, see [BenSig03] [HuaXie10] [Mat94]. Depending on this main idea, the new approach uses the stress-values for reducing or adding discrete material in the design space. Another important similarity is the discrete modeling of material.
Following ESO and BESO, the lowest stressed elements are removed from the structure. This is a simple but effective method. This mechanism has also an analogy in nature, during the development and growth of plants [Mat94] and is rooted from experience for solving problems in engineering also.
Due to the fact of discrete material, new elements can only be added to the borders of an existing structure. Without interpolation information the new material is placed to the areas with the highest stress levels, see the AESO method of Querin [Que00].
Depending on the discrete modeling, both methods are possible to handle linear and nonlinear effects in the FEA analysis. The only difference to a regular FEA simulation lies in the surface of the FEA model. Up to now the models from topology optimizations with discrete modeling is not as smooth as a model from a regular simulation.

2.2. Main differences to ESO/BESO

The new Topology Optimization has beside some similarities clear differences to the ESO/BESO methods. The main idea of ESO/BESO is a full stressed design, means all elements receive the same stress level. For this method the compliance-volume product can be assumed as an objective function [Edw07]. Opposite to this, the new Topology Optimization method uses only the volume as target or object function.
For the optimization the new method needs constraints. Remembering the motivation, the new Topology Optimization allows several constraints, e.g. displacement or reaction force. Also the combination of all constraints is possible. Normally a min-max formulation is used. But also other mathematical operators are possible to use, e.g. weighed or distance formulations. In the original approach of BESO, no constraints are used. Only the stress levels are important.
With the main focus to a normalized stress level, BESO adds material by comparing each element stress level to a reference level. Comparing this to the new Topology Optimization method more elements are added in each iteration. The reason is that the new Topology Optimization method adds only at the highest stressed elements(often called hotspots) material.
Starting optimization with infeasible solutions forces the optimization method to add material first to the structure. The BESO method finds the same solution in this case as an optimization run starting from full design space [Que00b]. The new approach offers in this case different solutions, because the process is controlled by the constraint limit. For industrial purposes this behavior is more powerful in later development phases, e.g. when load conditions must be changed to new requirements, the engineer wants to find a new feasible and as light as possible design but with a minimum of changes in the part.

2.3. Main process of the new Topology Optimization method

The flow chart in figure 1 illustrates the main steps of the new Topology Optimization method. The step size controller calculates first a basic rate. Depending on this basic rate, the number of removing and adding elements is defined. After the controller the necessary elements are inserted. In this way, hotspot areas are corrected. After this correction process, the lowest stress elements according to the reduction rate are removed. After adding and removing elements, it is important to check if the structure is connected. All force transmission points must be connected to the supports. If this check fails, the controller modifies the correction and reduction rate in order to produce a feasible structure. In the heuristic steps, non connecting elements are removed from the structure.

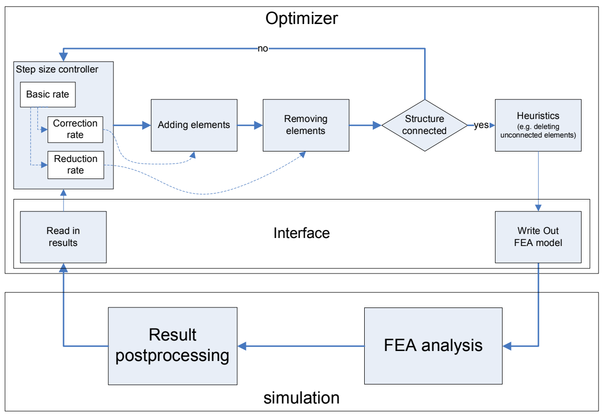

Figure 1: Flow chart of New Topology Optimization Method

The necessary interfaces to the FEA solver are integrated in the optimizer. After finishing all changes and checks, the optimizer writes the element input decks. After the FEA analysis, the result postprocessing evaluates all target functions and constraints. The read in process transfers this information back to the controller.

2.4. Integration into the industrial development process

Several steps are needed for the procedure of a topology optimization. Normally a topology optimization is based on FEA analysis. Due to this, the topology optimization must be coupled with a FEA solver.

One basic idea of this new approach is the integration in the standard development process, especially simulation process. Through this, a external FEA solver should be used. This demands interfaces to read and write the special formatted input decks of the solver. The optimizer supports two FEA solvers: Abaqus from Simulia® and Nastran.ND® from MSC®. Other FEA solvers can be integrated. Only the necessary interfaces have to be programmed in C++.
To minimize the complexity of the development, the new Topology Optimization doesn't manage the process of the topology optimization. The workflow is controlled through an external program, such as Optimus® from Noesis®.

2.4.1. Preprocessing

The preprocessing can be divided into two parts. First the normal FEA preprocessing has to be done. In figure 2, the two steps: meshing the part and define loads and structure supports are illustrated.

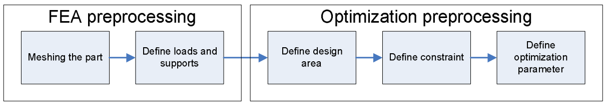

Figure 2: Preprocessing

After this, for the optimization preprocessing, different files in ASCII® format are used. The design areas, the constraint and the optimization parameter are chosen. All is flexible and can be adapted to the specific problem.

2.4.2. Interface between the optimizer and the FEA solver

The optimizer works internal with a data grid, see figure 3. The information form the internal data grid, called "matrix", can be transferred to the FEA model. On the initial run of the optimizer the elements are mapped to the matrix. This mapping is fixed over the whole optimization. The optimizer changes the status of the matrix. Status 0 means no material, Status 1 means material. After finishing the optimization steps, this information is mapped to the FEA model.
Only the elements with status 1 are written to the FEA data file. No other elements are available for the FEA solver.
Therefore, the results of the FEA analysis, apart from the aliasing effects at the border, have the same result quality as a normal analysis.

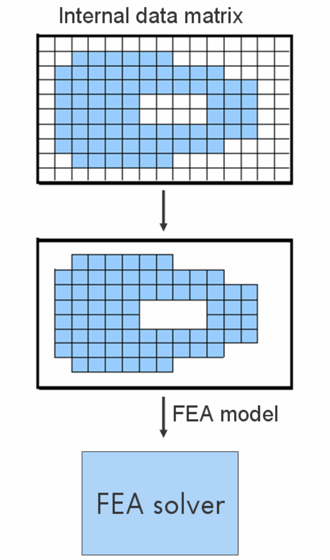

Figure 3: Interface between optimizer and FEA solver

2.4.3. Optimization Workflow

For the workflow Optimus® is used. The optimizer is integrated as User Algorithm. The optimizer writes the FEA input decks on his own. To optimus a reference to this file is transferred. During the process, optimus transfers files, starts the FEA solver and the postprecessing scripts to evaluate the stress values and the constraints. The process is shown in figure 4.

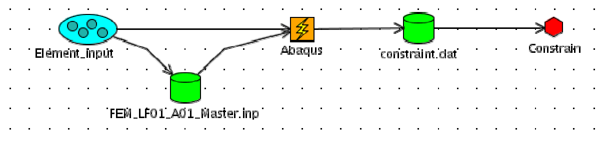

Figure 4: Preprocessing

2.4.4. Interface between the FEA solver and the optimizer

The figure 5 illustrates the process between FEA postprocessing and the interface of the optimizer.

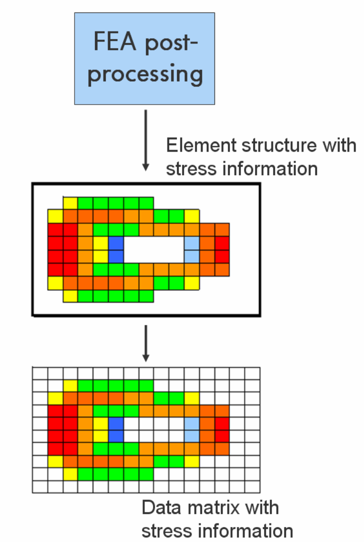

Figure 5: Postprocessing and interface to the optimizer

The result of the postprocessing is a list of all elements. Each line of this list represents one element. The line begins with the element id. Second entry is the stress value of the element.
The first step of the interface maps the stress value of each element to the internal element list.
After this, the stress values of the elements are mapped to the internal data matrix.

2.5. Integration of manufacturing restriction

For a feasible industrial casting part design, numerous manufacturing restrictions have to be fulfilled. Besides minimum and maximum material strength, normally a forming direction has to be taken into account. Special production processes - especially forging - need closed structures, at best, without any holes.
Additionally it is sometimes necessary to design symmetric parts, maybe for using the same part on the left and right side of a car. As well as a minimum strength restriction, casting directions, forging and symmetry restrictions are implemented.

2.5.1. Casting direction

Due to a casting direction, no material inside the structure can be deleted. In this way, no undercuts exists.

The figure 4 illustrates the differences between a part with active and non active casting restrictions. Without casting restriction, all elements can be removed from the structure. The optimization starts with the lowest stressed elements.

With casting restriction only the visible elements can be removed from the structure. After removing one element, the next element becomes visible. In each step, the current lowest element is deleted.

The red line in figure 6 shows the maximum element number in one row and which is possible to remove each iteration. Due to this, the algorithm can repair too large cuts in the next iteration.

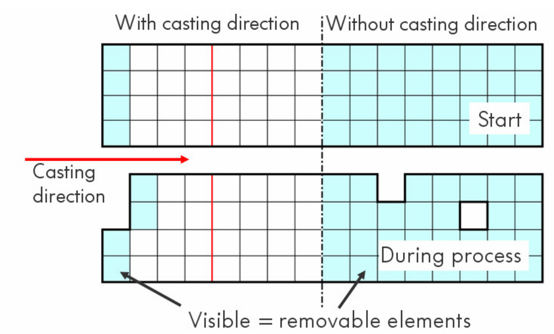

Figure 6: Casting direction

2.5.2. Forging

This restriction avoids parts with holes in the structure. To implement the mechanism, the last elements in one row are blocked for adding them to the visible group. Without getting visible, theses elements can't be removed from the structure, see figure 7.

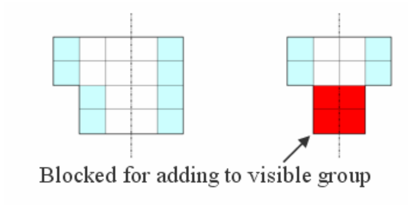

Figure 7: Forging

2.5.3. Symmetry

At the moment, only a plane symmetry is implemented. But other symmetries, like point symmetry, are easy to add. The mechanism for the plane symmetry can be directly transferred to them.
For a plane symmetry, all elements are divided into two groups. The first group allows modifications. After adding and removing elements in this group, the changes are mapped to the second group, illustrated in figure 8.

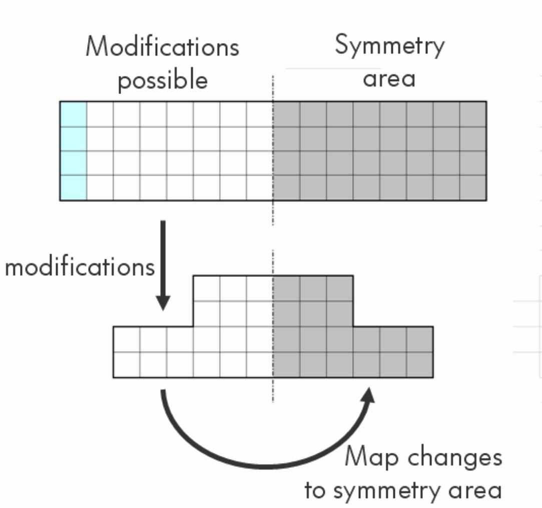

Figure 8: Plane symmetry

2.5.4. Minimum Strength

To avoid too small structures, which aren't possible to manufacture, normally filters are used. To this reason, discrete element modeling and using the half length of the minimum material strength for the elements length, the structure has a natural material strength. If the strength of the structure is smaller, the risk of collapsing in the FEA analysis is too high. This is demonstrated for example in figure 9.

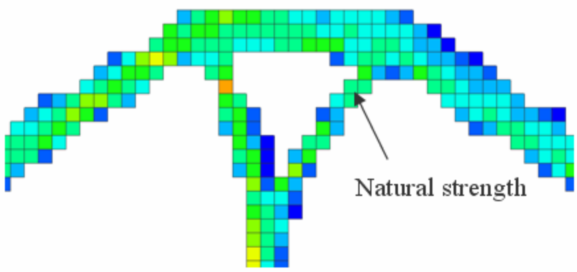

Figure 9: Minimum material strength

2.6. Example: Cantilever example with plastic material

One classical problem for testing topology optimization is the cantilever problem. In this case, it should directly demonstrate, how the new Topology Optimization method works using a nonlinear FEA analysis with plastic material. The material characteristic in this example has the specification of steel. On the left, two fixed supporting elements form the boundary as indicated in figure 10. In the middle of the plate on the right an enforced displacement of 20 mm at an angle of 90° to the main describes the load. In FEA simulation nonlinear geometry is activated and a real flow curve is used. As constraint function the reaction force at the node where the enforced displacement is applied, is used. The part should be optimized to a level of 10 kN. The target function is the minimization of weight, measured in elements.

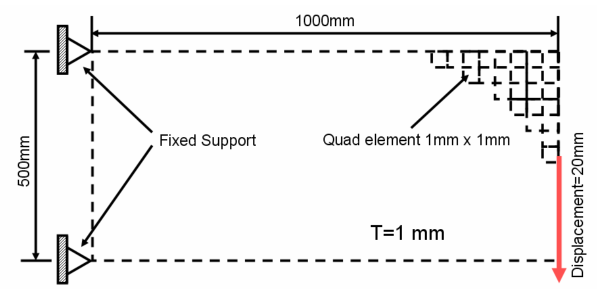

Figure 10: Cantilever problem with plastic material behavior

The optimization starts with a full design space with 125000 elements and an inertial basic reduction rate of 0.1. The general optimization process can be divided into three phases. The first phase is described by large reductions of elements up to the moment, where the constraint function rises strongly. In this second phase the constraint rises to the point where the constraint limit is reached. At this point two following iterations violate the limit. The optimization makes a cutback caused by the control mechanism and adds nearly 20% of elements to the structure. As a result the constraint offers the possibility to reduce the elements once again. Up to iteration 36 the optimization run reaches the point, where the cutback was made. Now the optimization control function is under the constraint limit. So the optimization progress enters the third phase. Characterized by slow step sizes, the optimization run offers improvements in detail. Through the oscillation around the constrain limit the last unnecessary elements are removed.

In figure 11 and 12 the optimization process is described through the number of elements and a normalized constraint. The value is the reaction force through the constraint limit of 10 kN. At ~25000 elements the cutback level is reached. After the second phase the optimization minimizes the weight to 20.35% of the starting value. In iteration 100 the optimization ends with a final value of 19.75%.

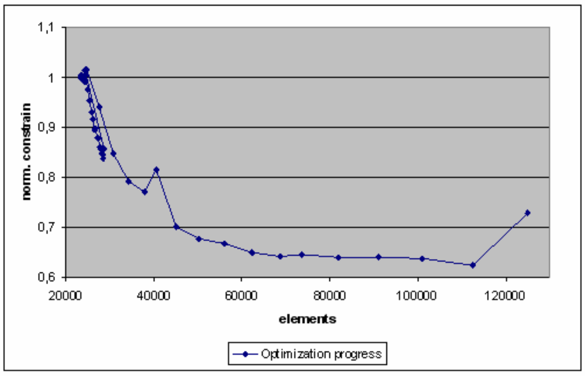

Figure 11: Optimization process of a cantilever problem with plastic material behavior described in Figure 10

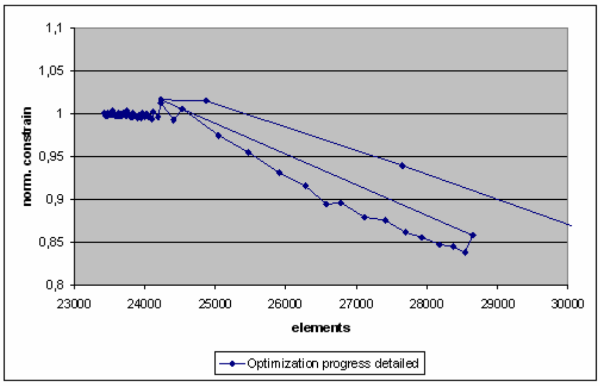

Figure 12: Detailed optimization process of a cantilever problem with plastic material behavior from Figure 10

The changes of the structure and the stress plots during the optimizer are illustrated in figure 13.

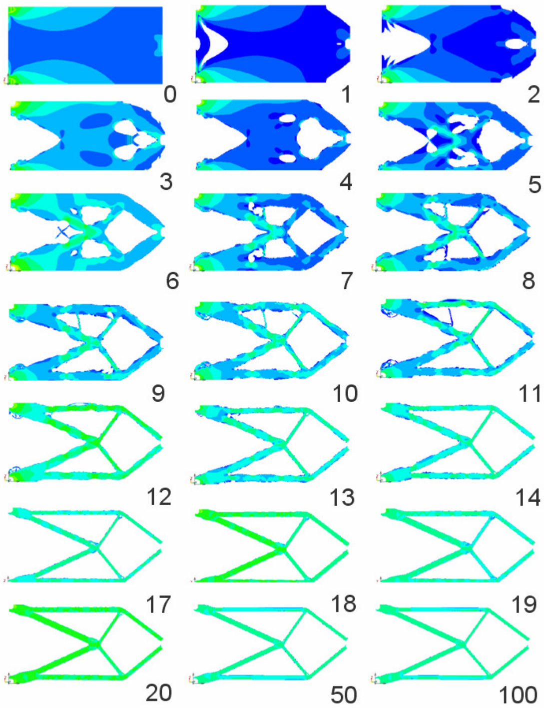

Figure 13: changes in optimization process of a cantilever problem with plastic material behavior from Figure 10

Figure 14 demonstrates the result quality of the new approach. Using nonlinear FEA analysis during the optimization run, it is possible to dimension all areas in the structure correctly. Due to this, the result shows a very good utilization of material in nearly all areas. Another positive aspect is the simplicity of the structure. No ramifications are proposed. This structure is easier to be constructed and manufactured.

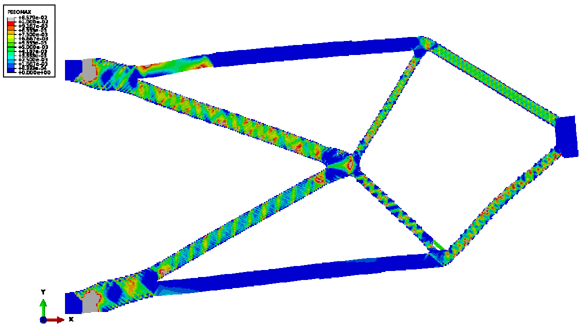

Figure 14: Nonlinear FEA analysis of optimization results from the example in Figure 10

The figure 15 shows the reaction force of the final iteration. In the figure is the progress of the reaction over the displacement illustrated. The final value at 20mm displacement is 10008,8 N. This demonstrates, that the optimizer has the ability to deliver a result exactly to the necessary constraint limit.

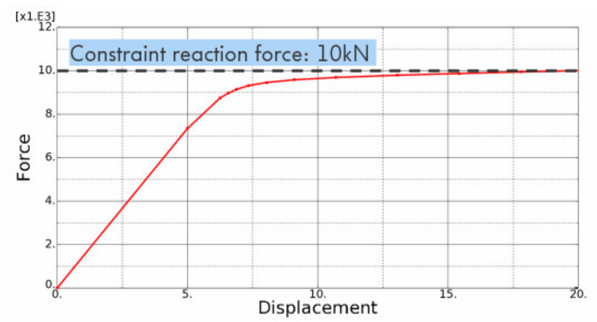

Figure 15: Reaction force of final iteration from the example in Figure 10

3. 6. CONCLUSION

In this paper an approach for a new Topology Optimization method is proposed. The method is developed based on requirements from the automotive industry. The main focus is the combination of finding a minimum weight and the best material distribution in one optimization run. To fulfill this task a discrete approach to include all nonlinear effects, e.g. plastic material behavior was chosen.

The example demonstrates the quality of the new optimization method. For the cantilever problem the new approach shows an advantage of 30% compared with a conventional industrial gradient based topology optimization methods. The new developed method shows the usability for real life development problems. The quality of the results is significantly increased compared with conventional gradient based topology optimizations, especially in cases with nonlinear effects, e.g. plastic material behavior, in the FEA simulation.

Finding a satisfying solution in topology optimization reduces the necessary development time in a development department. The first designs in CAD based on the optimization runs indicate very competitive weight and fulfill immediately the technical requirements and manufacturing restrictions. So development loops and development costs can be saved. New target and constraint functions will increase the usage and more problems can be solved in less time.

REFERENCES

G.I.N. Rozvany: Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics, *Struct Multidisc Optim 21, 90–108, Springer-Verlag*, Berlin, 2001

M.P Bendsøe: Optimal shape design as a material distribution problem, *Struct. Optim. 1, 193–202*, 1989

M.P. Bendsoe and O. Sigmund: Topology Optimization: Theory, Methods and Applications, *Springer-Verlag, Berlin*, 2003.

G.I.N. Rozvany, M. Zhou, T. Birker: Generalized shape optimization without homogenization. *Struct. Optim. 4, 250–254*, 1992

C. S. Edwards, H. A. Kim, C. J. Budd: An evaluative study on ESO and SIMP for optimising a cantilever tie–beam, *Struct Multidisc Optim (2007) 34:403–414, Springer-Verlag, Berlin*, 2007

K. Svanberg: The method of moving asymptotes-a new method for structural optimization, *Int J Numer Methods Eng 24:359–373*, 1987

Q.M. Querin, G.P. Steven, Y.M. Xie: Evolutionary structural optimisation using an additive algorithm, *Finite Elements in Analysis and Design 34 (2000 291-308, ELSEVIER*, 2000

X Huang, Y.M.Xie: Evolutionary Topology Optimization of Continuum Structures, *Wiley*, Chichester, 2010

Q.M. Querin, V. Young, G.P. Steven, Y.M. Xie. : Computational efficiency and validation of bi-directional evolutionary structural optimization, *Comput. Methods Appl. Mech. Engrg. 189 (2000) 559-573, ELSEVIER*, 2000

C. Mattheck: Design in der Natur, *Rombach Verlag*, Freiburg im Breisgau, 1997

AUTHORS BIOGRAPHY

Sierk Fiebig studied mechanical engineering at the TU Braunschweig. After his studies, he started research work for his Ph.D.-thesis at Volkswagen in optimization methods for the development of mechanical optimization methods. Since joining the company he has worked as a simulation engineer in the chassis development at Volkswagen Braunschweig.