Industrial applications of shape and topology optimisation and new approaches for shape optimisation considering fatigue

E. Schnack\(^1\), R. Meske\(^2\), J. Sauter\(^2\) and W. Weikl\(^1\)

\(^1\) Institute of Solid Mechanics, Karlsruhe University, Germany
\(^2\) FE-Design GmbH, D-76131 Karlsruhe, Germany

Abstract

The pioneering works of Schnack et al. [1,2] were taken up by Sauter in 1991 with the aim to apply the gradientless structural optimisation to large real-world engineering problems. This led to the development of the computer program CAOSS. The principle of structural optimisation based upon optimality criteria was successfully extended from shape optimisation to topology optimisation [3].

In the presentation, a short introduction and survey on structural optimisation will be given. Moreover, several industrial applications of topology and shape optimisation with CAOSS will be shown. First, an example on shape optimisation of a hyperelastic material structure is shown, where multiple non-linearities occur. In the following, 2D and 3D topology optimisation with contact boundary conditions are demonstrated. A third example considers the application of topology optimisation for redesign in the automobile industry. The results show significant improvements in the criteria weight, strength and stiffness. Finally, an outlook towards further developments will be presented.

In the last part of the talk, a two-scale continuum damage mechanics model for structural optimisation concerning fatigue will be presented, which was recently developed in the group of Schnack [4,5]. Here, the material behaviour is described by defining different partial differential equation systems on micro and meso scale. From this results an optimal algorithm for the shape optimisation of mechanical engineering structures using continuum damage mechanics. Our numerical and experimental results show significant improvements in lifetime maximization.
1 Introduction

Due to the demand to decrease the time-to-market of new products while maintaining a high quality level and reducing overall product development costs, structural optimisation tools have become of significant importance in the virtual product development process.

FE-Design offers with the optimisation system CAOSS an integrated solution for structural optimisation. CAOSS is successfully used in industry with an interface to MSC.Nastran under the product name MSC.Construct since 1997. Now, FE-Design has developed further interfaces to the solvers ABAQUS, ANSYS, IDEAS and MSC.Marc. This product is distributed by FE-Design under the product name TOSCA. Structural optimisation of real-world problems was up to now limited to linear analysis only. By taking advantage of the capabilities of the non-linear solvers like ABAQUS and MSC.Marc, optimisation with non-linear material behaviour and geometric and boundary non-linearities can be performed in the moderate non-linear area [6].

The above described gradientless structural optimisation is well established and approved for statical optimisation problems. But for more general, e.g. dynamic and multiaxial, loading cases also the fatigue behaviour of the material is to be considered, and the optimisation tool must again be extended. Modern approaches use a two-scale continuum damage mechanics model for structural optimisation concerning fatigue. Based on Kachanov's idea [7], a damage variable $D$ is introduced which describes the weakening of the material properties.

In the group of Schnack a modification of the original postulate by Lemaitre [8] is used, describing the material behaviour by defining different partial differential equation systems on micro and meso scale. The resulting coupled elasto-plastic calculation on the micro-level is then solved by using an implicit integration scheme.

2 Shape optimisation of a hyperelastic support

A substantial amount of the developed parts at the Freudenberg Group are rubber-metal components. The computation of rubber components inhibit in general several non-linearities. Because of the large deformation geometric non-linearity has to be taken into account. Due to the incompressibility of the material hybrid elements have to be used. The hyperelastic material law itself is non-linear. Sometimes contact problems occur, which lead to non-linear boundary conditions. These complexities imply that accurate calculations for these components can only be performed with a solver like ABAQUS which is specialized for non-linear problems.

The hyperelastic material behaviour was modelled in this case with a Neo-Hooke model. The initial configuration of the component is shown in Fig 1 on the left side. Due to the symmetry of the component only the right half was modelled. The component is fixed on the lower right side. In the radius there is a steel ring which is not shown in the figures.
Initial design  
Optimised design (10 Cycles)

Figure 1: Equivalent stress and geometry of the hyperelastic support.

Figure 2: Deformation and equivalent stress of initial and optimised design.
A tension and compression load in y-direction was applied on the steel ring. The compression load led to a folding of the component in the initial design, which had a negative influence on the life span.

The nodes on the lower left contour of the component were chosen as design nodes of the optimisation. The optimisation target was the minimization of the von Mises equivalent stress along the design nodes. Because the total stiffness of the component should not be changed too much, a constant volume constraint was defined. The optimisation yields a significantly improved design after 10 optimisation cycles. The modified geometry is shown together with the maximum equivalent stress of both load cases in Fig 1 on the right side. The deformation of the initial and the optimised component is shown for both load cases in Fig 2. The maximum equivalent stress was reduced in both load cases for 26% each. The folding of the component in the compression load case was prevented by the geometry change.

Further potential in the shape optimisation is seen from Freudenberg in the optimisation with user defined material laws. Due to the modular structure of CAOSS it is possible to use own material laws in the FE calculation as long as correct stress values are calculated for the optimisation. This will be subject of further investigations together with Freudenberg.

3 Topology optimisation with non-linear boundary conditions

The exact boundary conditions can very often not be given exactly during the design process of a new component. Non-linear boundary conditions may occur due to contact problems. These boundary conditions can only be transformed into equivalent boundary stresses with a high effort and loss of generality. Therefore it is desirable to allow general contact definitions at the boundary of the optimisation domain in structural optimisation.

![Figure 3: 2D connection rod and bolt with different mesh size and contact definition.](image-url)
As an example for a topology optimisation with contact an optimisation of a tension/compression connecting rod is presented. The FE-mesh of the available design space for the connection rod and the FE-mesh of the bolt is given in Fig 3. The lower and the left boundary are symmetry boundaries. The centre of the bolt is loaded in two load cases with a force of equal magnitude in positive and negative x-direction. A node-element contact is defined between rod and bolt. The boundary of the bolt with the coarse mesh is the master surface, the boundary of the rod with the fine mesh is the slave surface.

The optimisation target was the maximization of the stiffness with a volume reduction of the rod to 50%. No restrictions were assumed for the elements at the contact surface, hence a material reduction at the contact surface was admissible. The optimised material distribution is shown in Fig 4.

The optimisation yields a material distribution along the upper side of the connection rod. The compression force from the second load case is introduced diagonally into the arm of the rod while the load at the middle line is reduced. Therefore less stress is transferred via the contact surface at the middle line and material is removed in this area. As a result, the optimised design has a void area directly at the contact surface. A second smaller void area is created further along the circumference.

This example shows impressively the potential of topology optimisation with contact boundary conditions. A reorientation of the flux of the contact forces happens during the optimisation. This change of boundary conditions is only possible with contact conditions and not with prescribed boundary stresses as equivalent load. If it is not wanted due to design reasons to have void areas at the contact surface, the user can take this into account at the beginning of the optimisation. In this case all elements at the contact surface can be defined as "frozen elements", causing the material density to remain unchanged during the optimisation.

After the capabilities of the topology optimisation with contact conditions have been shown successfully for 2D, the application to 3D will be demonstrated in the following. The conrod and the bolt of the above example were extruded in the 3rd dimension, which gave a model with about 25000 elements and 29000 nodes. The surface at z=0 was defined as additional symmetry plane (see Fig 5).

Figure 4: Optimised 2D tension-/compression conrod (50% volume reduction, 16 iterations)
A contact surface was defined between the rod and the bolt with 1000 contact nodes on the slave surface. Tension and compression forces of equal magnitude were applied in two load cases at the outer node on the middle axis of the bolt, which resulted in a slight bending of the bolt.

The goal of the optimisation was the maximisation of the stiffness with a volume reduction to 40%. The first element layer at the contact surface was defined as “frozen” area, hence void areas at the contact surface were not admitted. The result of the optimisation is shown in Fig 6. It is similar to the 2D example. Due to the loading conditions the material is distributed further towards the point were the forces are applied.
4 Application of topology optimisation for the redesign of a transverse link of an Audi car

A redesign of an existing transverse link (see Fig 7 top) of an Audi car was necessary because of increased loading conditions due to the use of more powerful engines. To achieve the ambitious goal of increasing the strength and simultaneously reducing the weight of the link, topology optimisation with CAOSS was used from the beginning of the design process. A model of the maximum available design space was created. All forces relevant for the strength test were taken as loading conditions for the topology optimisation.

The result of the optimisation is shown in Fig 7 centre. It can be seen clearly that in comparison with the previous design a lot of breakthroughs in the structure have been generated and that several ribs have been at the wrong position.

The design proposal of the optimisation was modified slightly to get a design suitable for casting. Three virtual design cycles with FE calculations were performed in which details like radii and thickness parameters were changed. The basic concept—a frame with struts for the stiffness and breakthroughs for the weight saving—remained unchanged. The final variant is shown in Fig 7 bottom.

Comparison of the previous and the new design revealed that the maximum stress was reduced to 55%, the maximum displacement to 85% and the weight to 88% of the values of the previous design.

Figure 7: Topology optimisation of a transverse link
The mechanical strength tests with the prescribed loading conditions were passed for the first prototype at the first attempt. The consequent application of topology optimisation led to a significant improvement in the criteria weight, strength and stiffness and a very fast and efficient design process.

5 Two-scale continuum damage mechanics model

In mechanical engineering we are also often concerned with dynamically loaded constructions, i.e. machinery parts. In this case, special interest has to be taken on the fatigue behaviour of the material. In the following a two-scale continuum damage mechanics model will be presented which was recently developed for structural optimisation concerning fatigue in the group of Schnack [4,5]:

At first a two-grid model for the micro- and the meso-scale is introduced, with \( h \) being the gridsize of the microelement and \( H \) the gridsize of the meso-element. The Lin-Taylor hypothesis [9] is applied, i.e. the strain tensor for the micro and meso scale at the border between both elements is the same. For the yield stress on the micro-structure, the fatigue limit of the meso-level is used. The fatigue limit of the micro-grid is given by the yield stress on the micro-level modified by the relation between fatigue limit and yield stress on the meso-scale.

Concerning the relationship between micro- and meso-level, the original postulate by Lemaitre [8] states that if there is failure on the micro-level, only then there is complete failure at the same time on the meso-level. This has been modified by Grunwald [5] in the following way: If there is failure on the micro-level, it increases step by step and thus accumulates also on the meso-level. If finally a critical value on the meso-level is reached, then there is failure of the structure. This modification has been a result of our computer simulation for shape optimisation of dynamically loaded machinery parts.

Finally, a damage evolution equation for the damage variable \( D \) is given, including the rate of the accumulated plastic strain \( p \), together with the strain energy release rate \( Y \), the material parameter \( S \) and a step function:

\[
\dot{D} = \frac{Y}{S} \dot{p} \Theta( p - p_D ).
\] (1)

In the following a summary of the material law and the constitutive equations on the micro-scale is given which have to be solved in the numerical procedure:

\[
\varepsilon_{ij}^h = \varepsilon_{ij}^{eh} + \varepsilon_{ij}^{ph},
\]

\[
\varepsilon_{ij}^{eh} = \frac{1 + \nu}{E(1-D)} \sigma_{ij}^h - \frac{\nu}{E(1-D)} \delta_{ij},
\]

\[
\varepsilon_{ij}^{ph} = \begin{cases} 
\frac{3}{2} \frac{\delta_{ij}^{bd}}{\sigma_f} p^h & \text{if} \quad f = 0 \land \dot{f} = 0, \\
0 & \text{else},
\end{cases}
\]

\[
\dot{p} = \begin{cases} 
\frac{\sigma_f^2 R_p}{2 E S} & \text{if} \quad p^h \geq p_D^h \land f = 0 \land \dot{f} = 0, \\
0 & \text{else}.
\end{cases}
\] (2)
6 Numerical and experimental results

With the above described continuum damage model several numerical and experimental tests have been carried out at the institute of Schnack, using an SQP optimisation scheme. For the specimen a tension loaded stripe has been used with circular notches on both sides as start shape.

As a starting point the optimisation procedure was tested for static optimisation, i.e. minimization of maximum von Mises stress at the free boundary. The static optimisation was carried out without substructuring, whereas during the dynamical optimisation procedure we have worked with an adaptive mesh generation. The mesh of the statically optimised part was also used as start geometry for the damage optimisation. It was necessary to double the nodes on the free boundary (65 nodes) to achieve a stress error of $e_s < 5\%$ in this case.

![Image of comparison of circularly notched and optimised shapes](image)

Figure 8: Comparison of circularly notched and optimised shapes

In the four diagrams of Fig 8, numerical and experimental data are shown. In a.) there is the stress distribution for the start shape and the statically optimised one, and in b.) the distribution of the damage parameter $D$. In c.) the damage-parameter optimised profile is compared to the statically optimised one, whereas in d.) the statically optimised profile is compared to the damage equivalent stress optimised profile.

The differences between the classical static optimisation and the dynamical optimisation are rather small, but can well be seen in the different oscillations along the free boundary. This is probably due to local effects, which we have considered in the continuum model on the micro-level.
In numerical S-N-diagrams it was shown that for a special stress level the lifetime of the structure can be increased by a factor of about three, if working with the above described shape optimisation for dynamically loaded mechanical engineering structures.

7 Conclusions

The use of structural optimisation tools in the early stage of the development process offers new potential in the process chain. The development process becomes faster and more efficient by using topology and shape optimisation. This results in structures which are lighter, stronger and more durable which constitutes a competitive advantage for the companies that use these solutions.

The optimisation system CAOSS provides an integrated solution for structural optimisation of real-world problems. It has reached wide spread acceptance in industry and offers interfaces to all major FE-solvers. With the interfaces to the non-linear solvers ABAQUS and MSC.Marc it is now also possible to perform structural optimisation with moderate non-linearities.

Under dynamic loading conditions, the two-scale continuum damage mechanics approach by the group of Schnack provides a suited tool for shape optimisation under consideration of the fatigue behaviour, leading to a higher lifetime of the optimised structure.

8 References