Topology Optimization of an Engine Bracket Under Harmonic Loads

R. Helfrich[1], A. Schünemann[1]

1: INTES GmbH, Schulze-Delitzsch-Str. 16, 70565 Stuttgart, Germany, www.intes.de, info@intes.de

Abstract: In the past years, topology optimization has been established more and more as standard method to support the load path dependent layout of vehicle and engine parts under weight constraints. In many cases, static load cases are applied. However, studies have shown that the optimal layout under dynamic loads are considerably different from layouts under static loads. Of particular importance are harmonic loads as they are used in typical frequency response analyses. Special difficulties of a dynamic topology optimization are related to the distinct nonlinear dependencies of, for example, displacement amplitudes at certain points of a component on changes of mass and stiffness distribution in the design space.

The paper shows the topology optimization in combination with static and modal frequency response analyses using appropriate examples. Additional manufacturing conditions like release directions, symmetry planes, or frozen regions are also considered. Weight constraints are of special interest. Results are discussed with particular attention on the dynamic background.

The paper will show the specialties of topology optimization under dynamic, in particular harmonic, loads. An industrial example of an engine bracket are used to present the process. Analysis and result evaluation are made with an industrial FEA code (PERMAS with VisPER).

Keywords: Topology optimization, static loads, dynamic loads, free-form shape optimization, stress, compliance, displacement constraint

1. Introduction

Topology optimization is a great means to see how a structure under certain loadings and boundary conditions should look like. The basis is a Finite Element (FE) model of the structure, where the part to be optimized is represented as design space, which uses the maximum geometric dimensions where the optimized part has to fit in. A fine mesh is required, if one wants to see structural details from the optimization. Realistic loadings and boundary conditions are very important to get a usable optimization result. In addition, an idea about the final weight is needed and an idea about the optimization objective. Most frequently, the minimum compliance is used as objective to make the structure as stiff as possible, where compliance is defined as the strain energy in the structure under loading. Additional constraints for the topology optimization are possible and due to a certain intended manufacturing process very important like release directions for cast parts. The design variable of the topology optimization is the filling ratio of each finite element in the design space with values between zero and one. A zero filling ratio indicates an element, which is not needed in the optimized structure, and a filling ratio of one indicates that this element has to be kept in the optimized structure. It is the mission of the topology optimization to get a clear result with filling ratios very near to zero and very near to one. Such a result can then be used for further design steps. Due to the separation between not needed and kept elements, the interface between both areas of elements is very jagged. So, a smoothing process is needed, which generates a smooth surface for the areas of kept elements. With this result, the communication with designers is facilitated and a suitable design can be more easily achieved.

In case, one wants to get not only an idea about the shape of a structure but also about its durability, then we have to state that stress results from topology optimization are of limited value, because the jagged surface of the kept elements is not a preferable basis to calculate stresses. Hence, the results from topology optimization have first to be used to design the new part. Then, for the new structure, a shape optimization can be used to optimize durability values like stresses, strains, and safety factors taking into account additional constraints like weight and compliance.

In the following sections, an engine bracket is introduced as industrial example. This model is used to perform a topology optimization for static load cases. Afterwards, dynamic load cases are added. It is worth mentioning that dynamic load cases only are typically leading non-physical results. As a rule, all dynamic optimizations should always contain at least one static load case (e.g. the weight). Subsequently, a new model is created using the result of the topology optimization with dynamic load cases. This model is very close to the smoothed hull. Then, a freeform shape optimization is performed to minimize the stresses due to the dynamic loads.

2. Model of Engine Bracket

Fig. 1 shows the model for topology optimization, where the fixation points to the engine are supported and the loading point has an offset to the surface of the structure. This offset is modelled using a rigid body connection between loading point and structure. The engine bracket will become a cast part. Therefore, a release direction is specified to describe the manufacturing process. The fixation points to the engine block should not be changed. There, frozen regions are defined, which keep the fixation areas in the design space, but avoid any modification of these areas by the optimization process. In this model, all finite elements are part of the design space. All areas beside the frozen regions are subject to change by optimization.

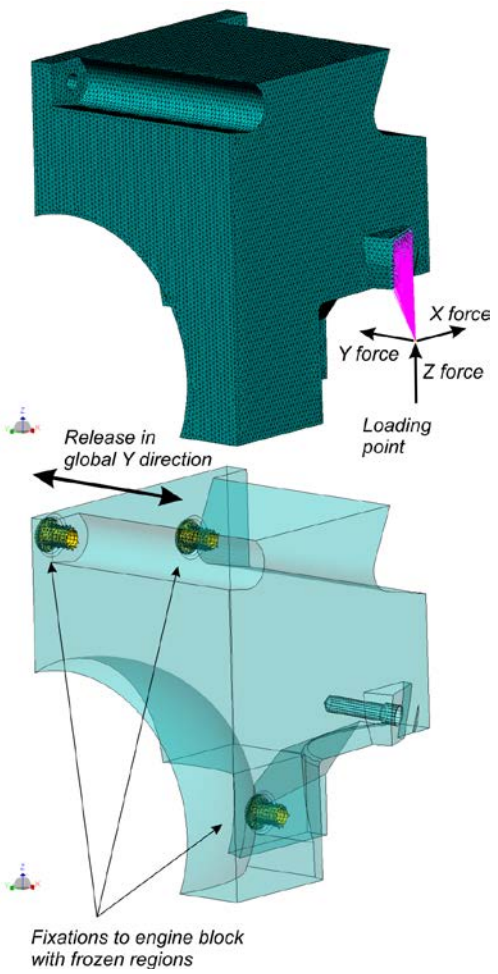

Figure 1: Optimization model of engine bracket

The dimensions of the model are about X/Y/Z = 190/165/230 mm. The model size is about 130,000 nodes and 386,000 degrees of freedom.

3. Static optimization

For the topology optimization under static conditions, a linear static analysis is used. The optimization objective is to minimize the compliance, which denotes the total strain energy in the design space, or in other words, the global (and not local) stiffness of the design space. The weight is chosen as the weight of the predecessor part. Beside the definition of a release direction and frozen regions as shown in Fig. 1, a minimum member size of 10 mm is used. This restriction avoids the generation of very small structures inside the design space and helps to get a producible result.

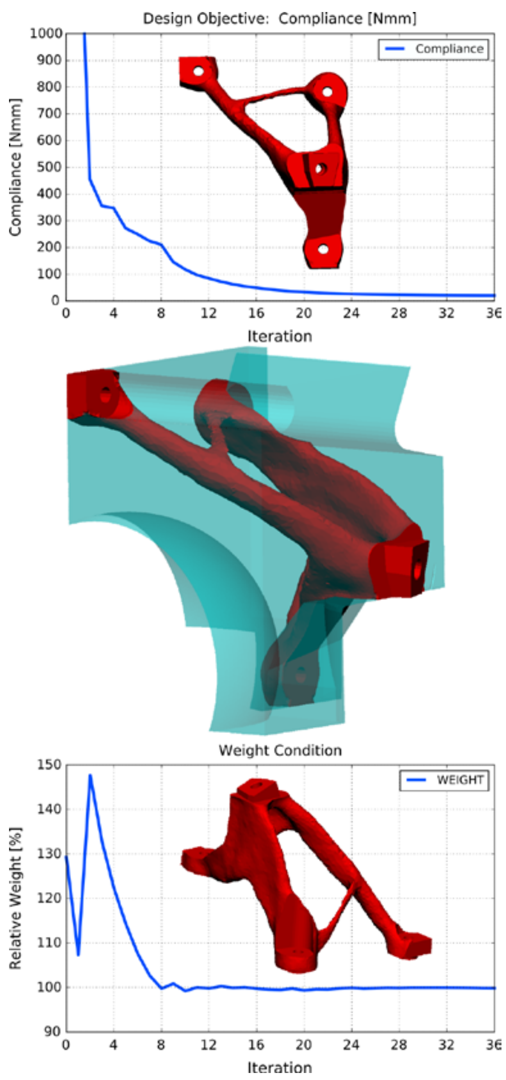

Figure 2: Bracket shape under static conditions with compliance history (top) and weight history (bottom)

The result of the static topology optimization is shown in Fig. 2, where the surface of the remaining elements is smoothed for better impression of the result. Different views also facilitate the spatial impression of the final shape. The final compliance is about 21.3 Nmm.

Because our focus is on dynamics, we performed a frequency response analysis after the static optimization of the bracket. Fig. 3 shows the displacement amplitudes of the frequency response for the final design from static topology optimization.

The frequency response analysis uses the same loads as the static analysis but with harmonic excitation. The number of frequency points is 101 (i.e. 100 equidistant points from 10 Hz to 1000 Hz and one eigenfrequency).

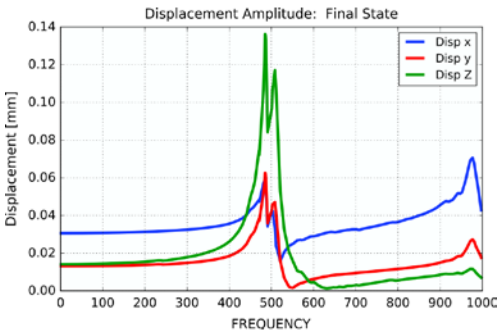

Figure 3: Displacement amplitudes from frequency response for the final shape of static topology optimization

3. Dynamic optimization

For the topology optimization under dynamic conditions, a frequency response analysis is used. The optimization objective is to minimize the displacement amplitude. The weight is chosen as the weight of the predecessor part. Beside the definition of a release direction and frozen regions as shown in Fig. 1, a minimum member size of 10 mm is used. This restriction avoids the generation of very small structures inside the design space and helps to get a producible result.

While in static topology optimization the initial filling ratio was 0.5, the dynamic topology optimization has started with a filling ratio of 1.0. The compliance is used as additional constraint to the optimization with a value of 22.0 Nmm.

Beside the frequency response analysis, the static analysis as used for the static topology optimization is also applied. It is crucial in dynamic topology optimization that static conditions are also applied. Pure dynamic conditions tend to reduce the mass too much, because dynamic conditions are not reflecting static conditions automatically.

The result of the dynamic topology optimization is shown in Fig. 4, where the surface of the remaining elements is smoothed for better impression of the result. Different views also facilitate the spatial impression of the final shape.

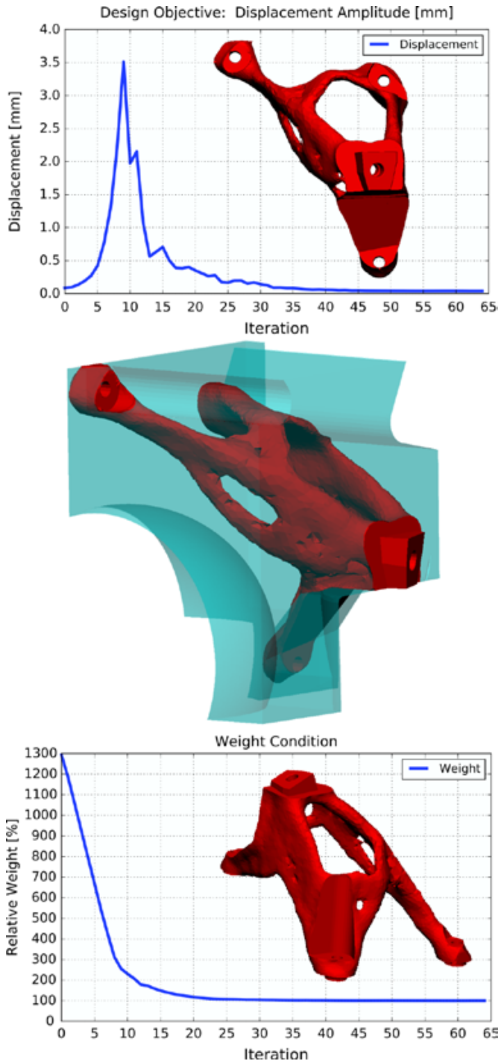

Figure 4: Bracket shape under dynamic conditions with displacement amplitude history (top) and weight history (bottom)

The initial and final amplitude in the frequency response is shown in Fig. 5. There is an additional constraint for the maximum amplitude in the considered frequency range of 0.04 mm. The comparison between the lower graph in Fig. 5 with Fig. 3 shows a big difference not only of the maximum displacement amplitude but also of the resonance frequency.

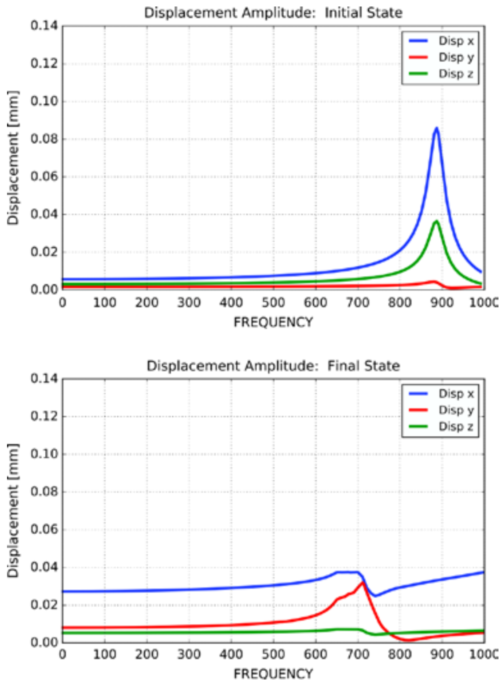

Figure 5: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from dynamic topology optimization

4. Optimization of stresses

As explained above in the introduction, the stress in the design achieved by topology optimization is not suitable for evaluating the durability of the optimized part. Therefore, a subsequent shape optimization is performed.

The first step is to use the final design from dynamic topology optimization for meshing. To this end, the hull of the final design can be exported for meshing. The geometry of the hull is only slightly modified in order to keep the design as close as possible to the topology optimization result. Fig. 6 shows the new part and its boundary conditions, which are identical to the topology optimization. The model size is about 980,000 nodes and about 2,900,00 degrees of freedom.

The optimization method used is a non-parametric free-form optimization based on optimality criteria best suited for stress optimization. This method allows a thickness change at every node of the bracket surface during optimization, while the mesh topology remains unchanged and the node coordinates at the surface and in the interior of the solid are modified to preserve the mesh quality.

The objective used was the weight combined with stress and displacement constraints.The thickness change was limited to ±4 mm. The stress constraint was 35 MPa, the compliance constraint is 38 Nmm and the constraint of the displacement amplitude is 0.7 mm.

As for the previous dynamic topology optimization, the free-form optimization used static and frequency response analysis for optimization.

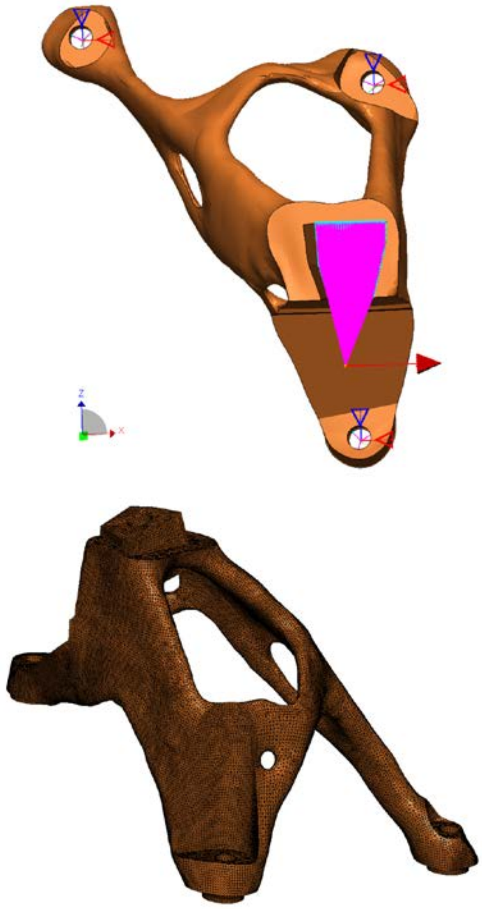

Figure 6: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from dynamic topology optimization

The free-form optimization result is achieved after 6 iterations. The history of the objective function weight and the constraints compliance and stress is shown in Fig. 7. The weight is reduced by about 15%. All constraints are observed in the final design.

The shape change is visualized in Fig. 8, where the position change of the nodes normal to the surface is shown. The maximum allowed values of ±4 mm are not exploited. The maximum thickness change is about 2 mm growing and shrinking.

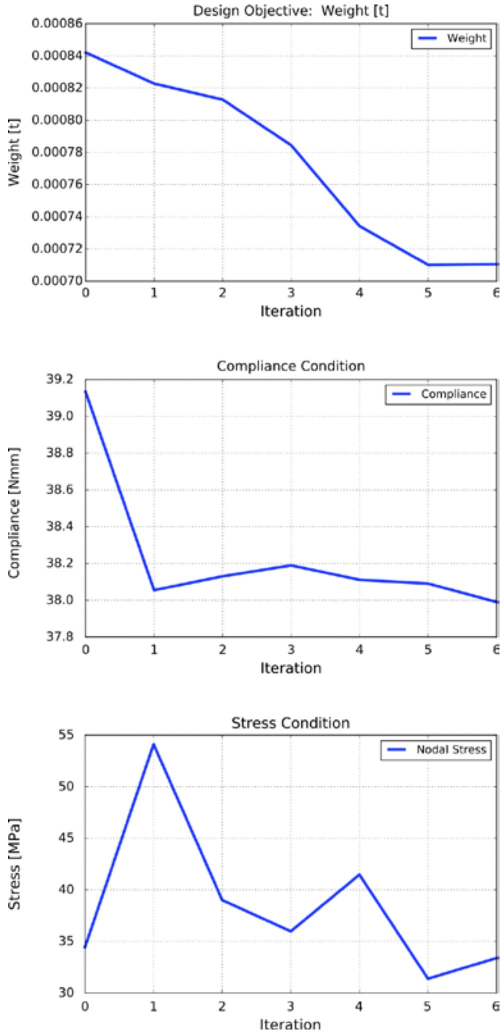

Figure 7: Histories of objective function (weight) and the constraints compliance and stress (von Mises)

The stress distribution before and after the free-form optimization is shown in Fig. 9. The stress constraint is not violated before optimization. Nevertheless, stresses are widely reduced through other constraints like compliance. A detail with high initial stress before the optimization is shown in Fig. 10, where the stress is reduced from 34.5 MPa to 18.5 MPa.

Finally, the initial and final amplitude in the frequency response is shown in Fig. 11. The constraint condition for the maximum amplitude is observed and does not exceed 0.7 mm.

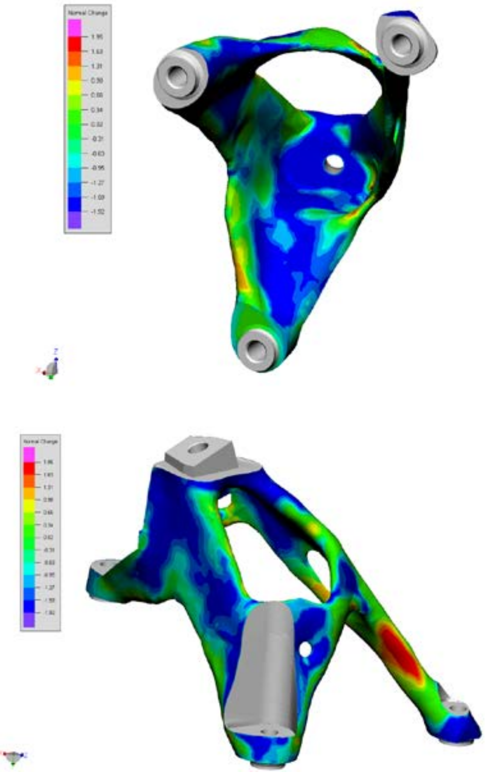

Figure 8: Normal thickness change by free-form optimization

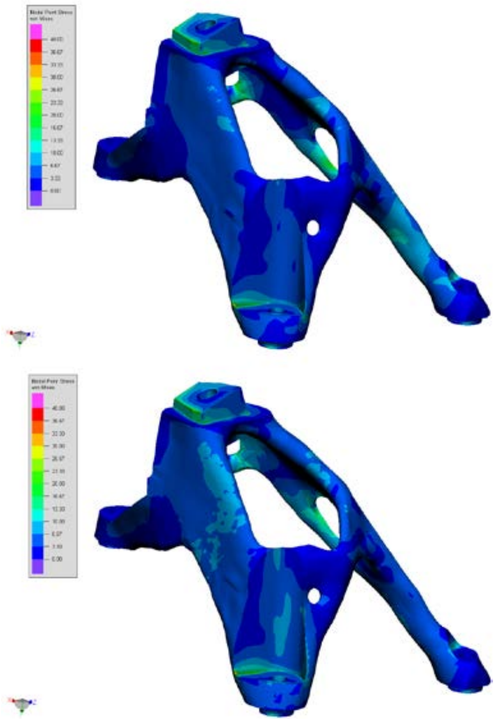

Figure 9: Von Mises stress distribution before and after free-form optimization

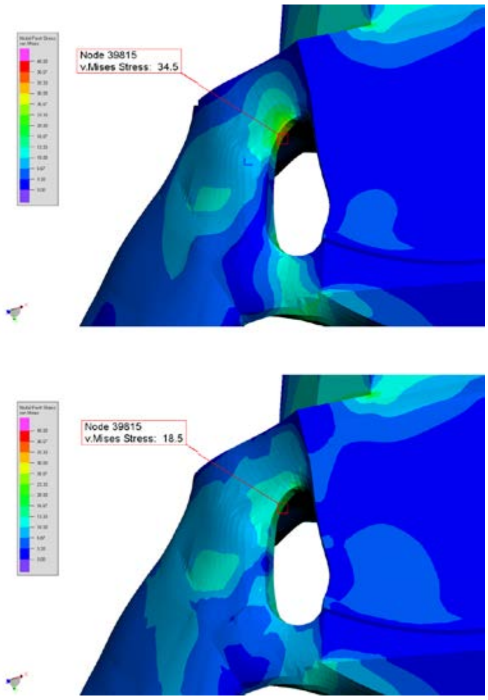

Figure 10: Von Mises stress distribution before and after free-form optimization for a detail with high initial stress

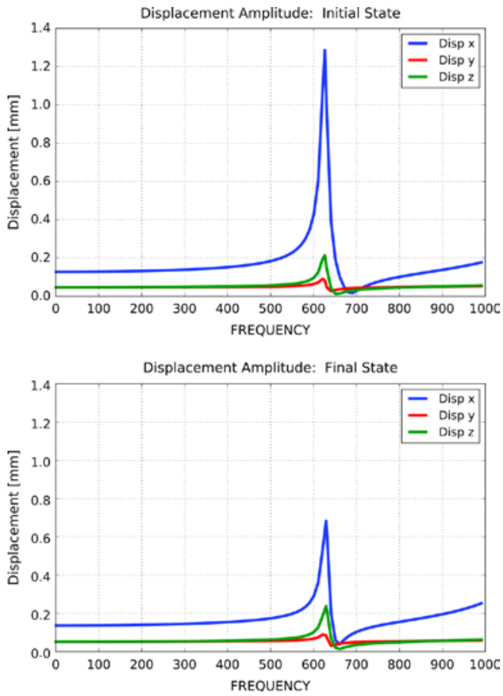

Figure 11: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from free-form optimization

5. Conclusion

An engine bracket as industrial example has been used to show the influence of dynamic analysis on the layout found by topology optimization. It is highly recommendable for dynamically loaded parts to take into account the dynamic loading during topology optimization. Vice versa, static loading should not be neglected. At least the weight of the part and attached parts can be a significant loading.

The layout from topology optimization has been used to generate a new geometry and design of the bracket. A new mesh was generated.

Stresses are difficult to be limited by topology optimization. Therefore, a free-form optimization is used to optimize the shape of the final design. A non-parametric free-form optimization was used with weight as constraint and additional constraints like static compliance and dynamic displacements as well as stresses. Static analysis and frequency response analysis are taken into account.

6. References

[1] M. Schreck, A. Schuenemann: "Design and endurance of an engine bracket", Proceedings of the13th PERMAS Users Conference, Stuttgart, Germany, 14-15 April 2016, ISBN 978-3-926494-16-0.