DESIGN OF AN ENGINE BRACKET BY SIMULATION

R. Helfrich, A. Schünemann
(INTES GmbH, Germany)

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. Beside the creation of design ideas, users of topology optimization often want to achieve the final design by this type of optimization based on their material and loading definitions without further design work. These requirements lead to a simulation-driven design process, where topology optimization and shape optimization are combined to obtain the desired design. The key point of the design process is a software concept, where all necessary analysis steps and the optimization are integrated in one single software.

Special focus is put on dynamic loading. Today, topology optimization is often used with static load cases. However, studies have shown that the optimal layout under dynamic loads is considerably different from layouts under static loads. In addition, it is very important to model the boundary conditions of the part to be optimized as realistic as possible. The best way is to bring all connected parts into the model and use reduction methods where possible.

Moreover, because topology optimization has limited capabilities to determine the stress distribution in the optimized part, a subsequent shape optimization has to be considered to complete the design process. Because the shape found by topology optimization has a free geometry, a free-form optimization method is the right choice for this task. Both stress and weight optimization are supported under additional constraints like displacement amplitudes due to harmonic loading.

As an industrial example, the paper shows an engine bracket, where starting from the available design space and realistic loading and boundary conditions a topology optimization generates a design. This design is directly used for re-meshing and subsequent shape optimization to optimize weight and endurance related quantities. Analysis, optimization, and result evaluation are performed with an industrial FEA code (PERMAS with VisPER).

1. Introduction

Topology optimization is a great means to see how a structure under specific 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 loading and boundary conditions are very important to get a usable optimization result. In addition, an idea about the final weight and about the optimization objective is needed. 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 like symmetry and release directions for cast parts are possible and due to a particular intended manufacturing processes very important. 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 close to zero and very close 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 obtained.

In the past, the meshes of the design space were frequently made from TET4 elements in order to get the layout faster. Due to the high stiffness of this element type and due to today’s higher speed of topology optimization methods, there is a clear recommendation to use TET10 elements instead.

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 and dynamic load cases under fixed boundary conditions. Afterwards, the engine is added as main connected part to the bracket. The engine model is reduced by dynamic condensation to limit the model size and the required computational effort for topology optimization. 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.

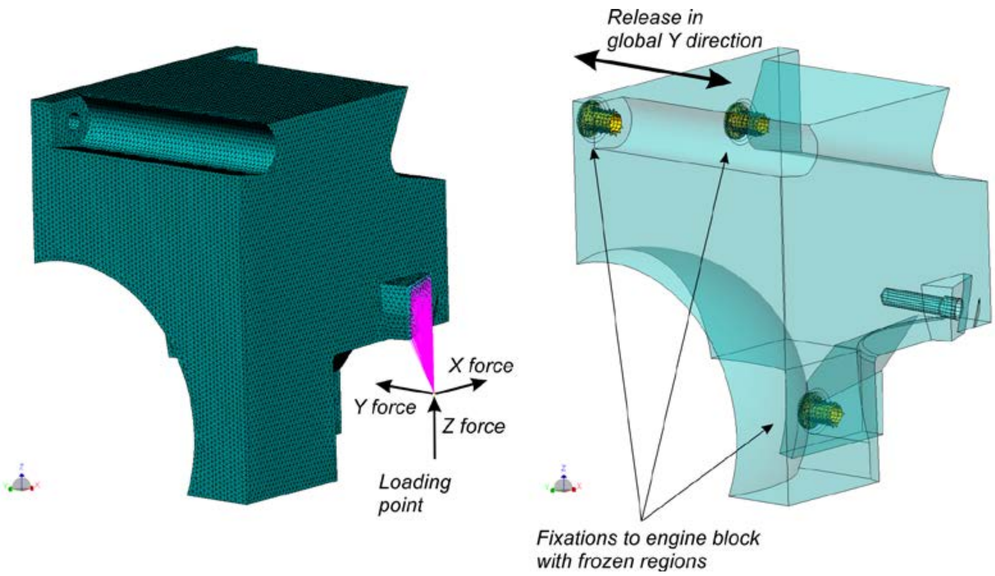

Figure 1: Optimization model of engine bracket

The dimensions of the model are about X/Y/Z = 190/165/230 mm. The model size is about 985,000 nodes and 2,951,000 degrees of freedom.

3. Dynamic Optimization with Fixed Supports

For the topology optimization under dynamic conditions, a frequency response analysis is used. The optimization objective is to minimize the displacement amplitude in X direction. The other directions are taken as constraints with a 0.3 mm limit. 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.

The fixations to the engine were considered as the supports of the bracket. The next section explains how to take into account the engine stiffness in addition.

There were static loads specified for the bracket. The frequency response analysis uses the same loads but as unit loads with harmonic excitation. The number of frequency points is 100 equidistant points from 10 Hz to 1000 Hz plus all eigenfrequencies in this frequency range.

Beside the frequency response analysis, the static analysis 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. Every optimization loop contains a static analysis and a modal frequency response analysis.

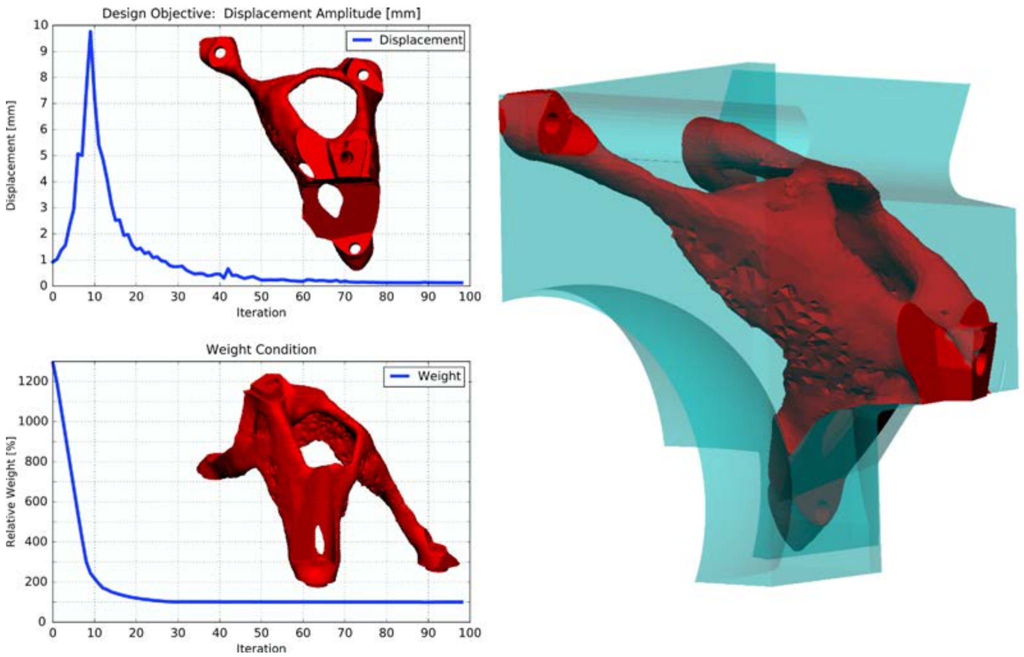

Figure 2: Bracket shape under fixed supports with displacement amplitude history (top) and weight history (bottom)

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 33.0 Nmm to reflect the static part of the loading.

The result of the dynamic 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 initial and final amplitude of the frequency response function is shown in Fig. 3.

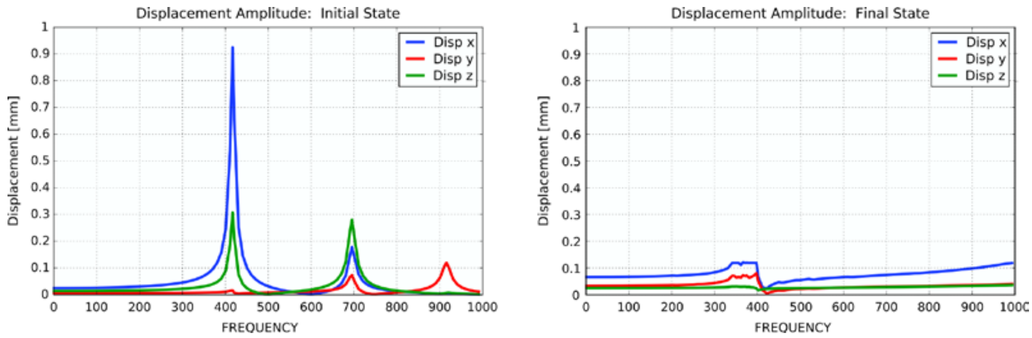

Figure 3: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from dynamic topology optimization under fixed supports

4. Dynamic Optimization when Mounted on Engine

Another topology optimization of the same bracket has been performed to demonstrate the importance of realistic boundary conditions. In the previous section, the supports of the bracket were fixed. Here the complete engine has been used with the real mounting points to model the connection stiffness at the supports.

The engine model is much larger and has much more nodes than the bracket. So, the additional engine model raises drastically the computation time for the topology optimization. The classical solution for this situation is to use dynamic condensation to reduce the size of the engine model. A Craig-Bampton dynamic condensation generated the reduced stiffness, mass and damping matrices of the engine, where the nodes of the engine, which are connected to the bracket, are used as remaining nodes for the static part of Craig-Bampton method. For the condensation, 708 interface nodes have been selected and 50 modes of the engine up to 4,000 Hz have been taken into account.

The conditions for the topology optimization are the same as for the fixed supports.

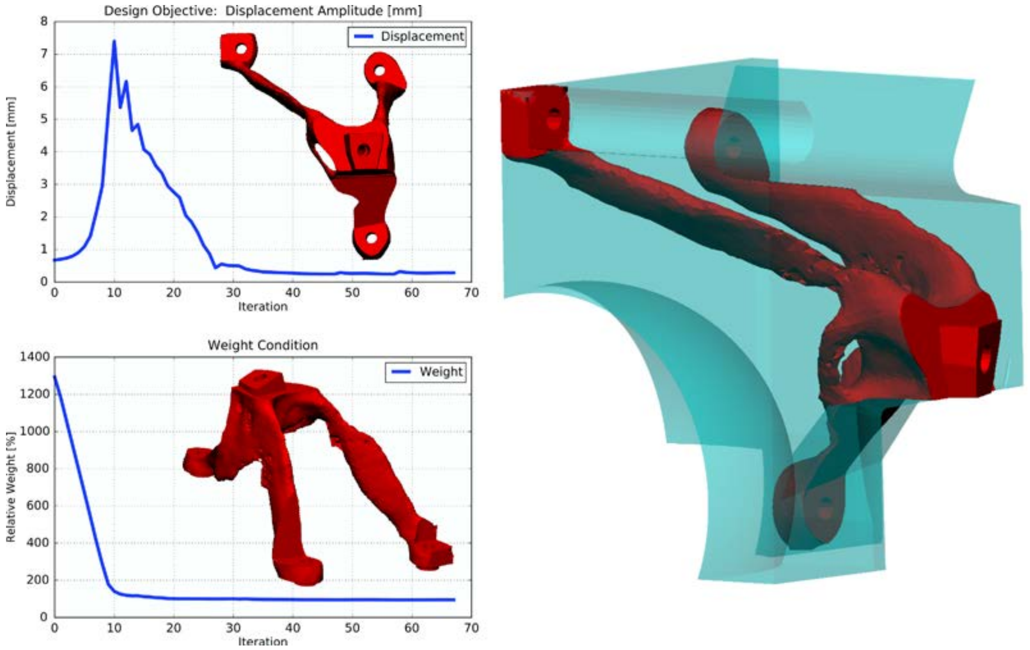

Figure 4: Bracket shape mounted on the engine with displacement amplitude history (top) and weight history (bottom)

The result of the dynamic topology optimization with the engine 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.

The initial and final amplitude of the frequency response function is shown in Fig. 5.

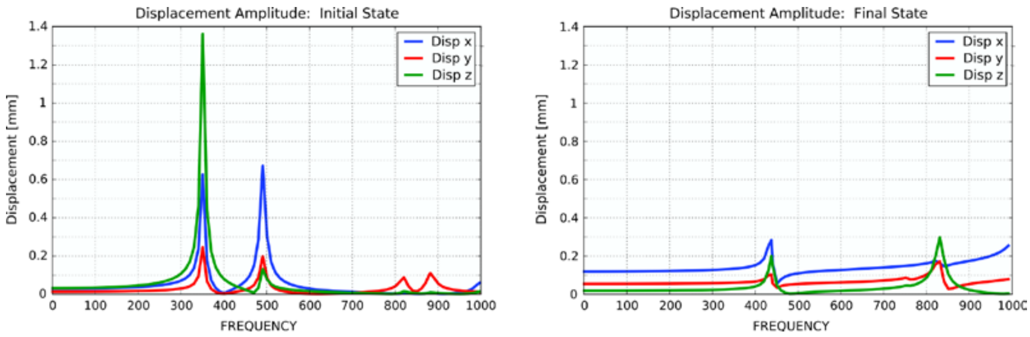

Figure 5: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from dynamic topology optimization with engine

Comparing this layout with the layout of the previous section there are significant differences of the shape and the response behaviour. While the final mass and compliance is the same, the final displacement amplitudes are larger than with fixed bracket supports.

5. 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.

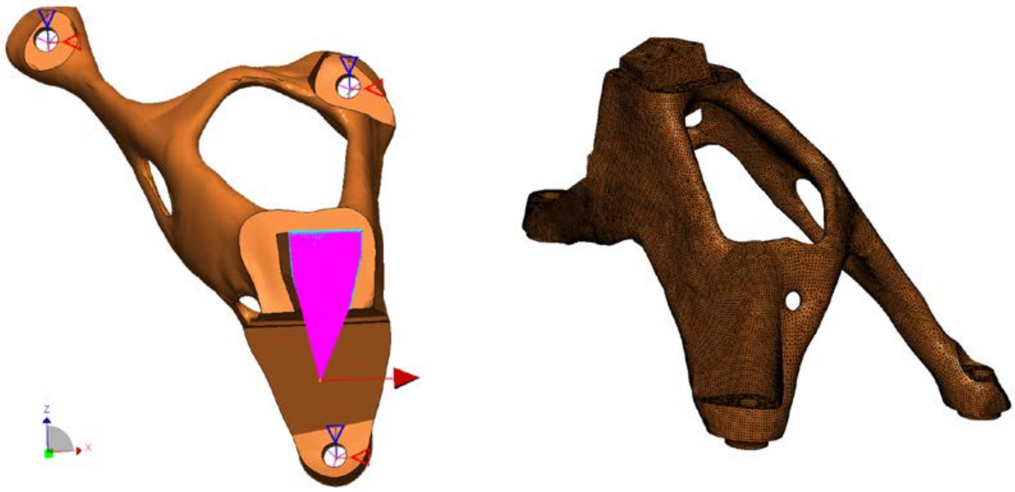

Figure 6: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from dynamic topology optimization

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.

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.

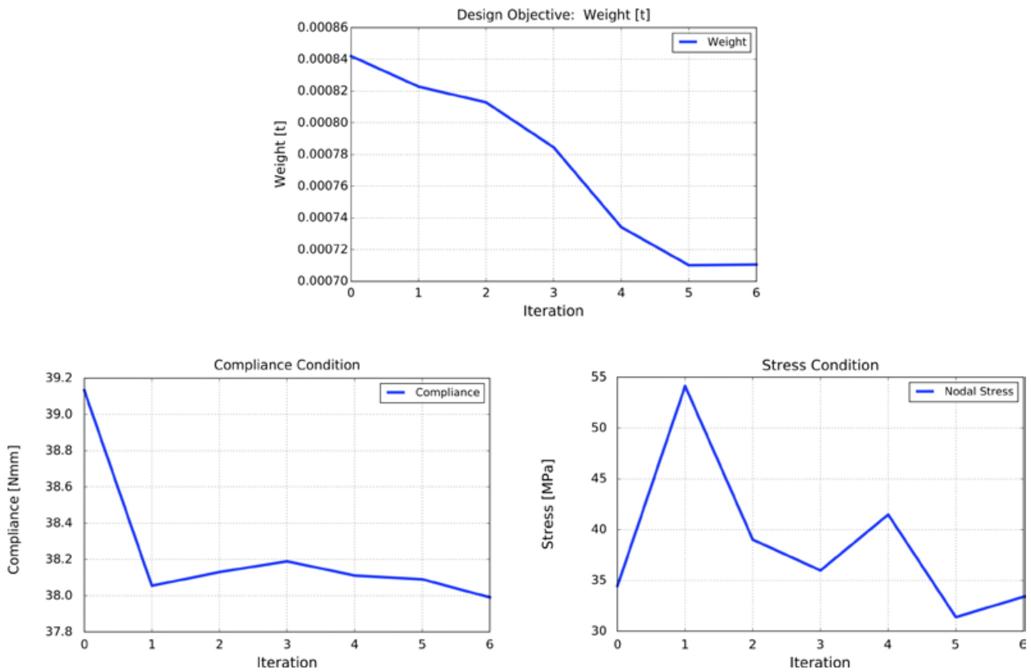

Figure 7: Histories of objective function (weight) and the constraints compliance and stress (von Mises)

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.

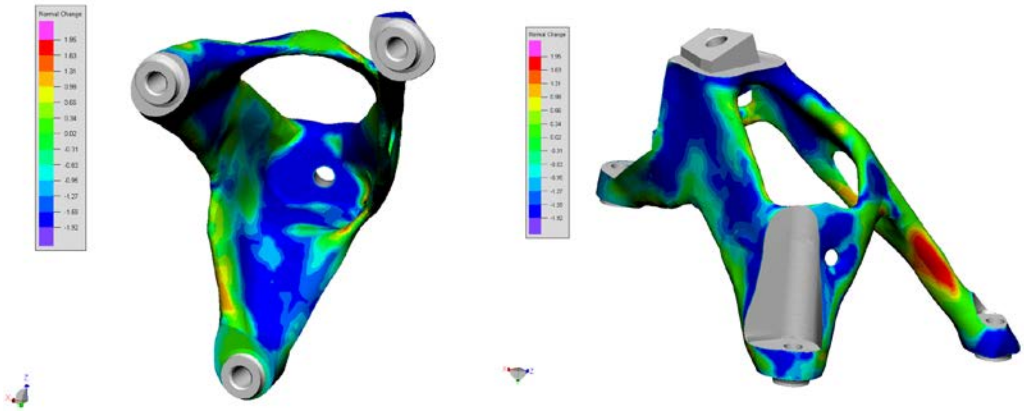

Figure 8: Normal thickness change by free-form optimization

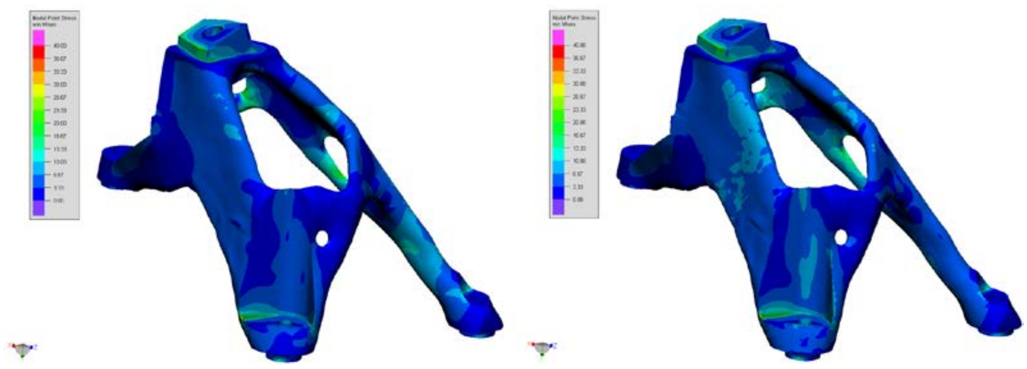

Figure 9: Von Mises stress distribution before and after free-form optimization

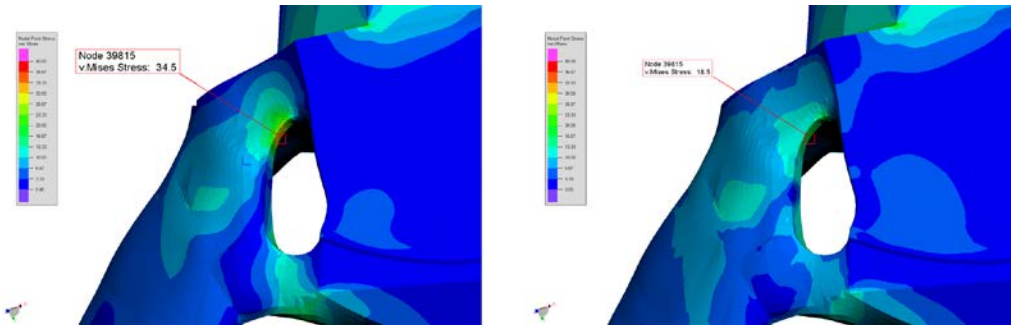

Figure 10: Von Mises stress distribution before and after free-form optimization for a detail with high initial stress

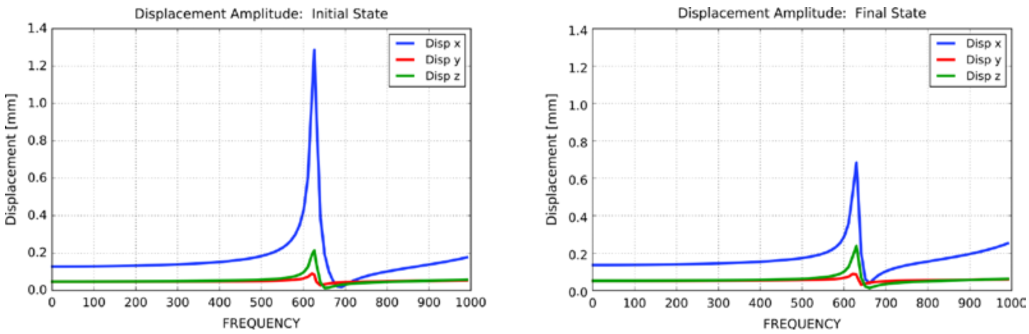

Figure 11: Frequency response amplitudes for the initial (top) and final optimized (bottom) shape from free-form optimization

6. Conclusion

An engine bracket as industrial example has been used to show the influence of connected parts in dynamic analysis on the layout found by topology optimization. It is highly recommendable to take into account connected parts during topology optimization. In addition, static loading should not be neglected.

In order to reduce the model size and the related computation time, the connected parts were reduced by dynamic condensation using the Craig-Bampton method.

The smoothed 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.

The described process of topology and free-form optimization is capable to provide a geometry, which is fully generated by simulation. It depends on the design and cost requirements whether the generated geometry is accepted for production.

7. References

Schreck, M., Schünemann, A. (2016). *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.

Helfrich, R., Schünemann, A. (2016). *Topology Optimization of an Engine Bracket Under Harmonic Loads*: SIA International Conference « Automotive NVH Comfort », Le Mans 19-20 Oct 2016

In case of questions, please contact first author by e-mail to Reinhard.Helfrich@intes.de