Let Simulation Do the Design - Faster

M. Shimozono
(INTES Japan K.K., Japan)

R. Helfrich, R. Fischer
(INTES GmbH, Germany)

Abstract

In times of new 3D printing technology progresses every day, topology optimization has already been identified as an ideal tool to create new design ideas. Additional design constraints can be added in order to take specific process and form conditions into account. Therefore, CAE engineers are dreaming of a design by simulation.

But the result of a topology optimization cannot be directly sent to the printer in many cases, because a number of important conditions are not yet part of the topology optimization. First of all, the surface is not smooth and smoothing changes the structural behaviour. Moreover, intermediate densities are an obstacle against automated interpretation. The classical approach is to forward the smoothed surface to the designer, who then should be inspired by the topology optimization result to find a new and better design. This new design has to be meshed and computed again to hopefully show the intended improvements. Sometimes, several loops between CAE engineer and designer are necessary until a satisfactory design could be achieved. This is the time aspect.

Other aspects are related to points, which are not directly considered during topology optimization, like safety factors and endurance, uncertain material, shape and loading parameters. Such factors make the design workflow more complex and the question is, can we save time and gain quality by using more automatic steps to postpone or even replace the job of the designer?

One possible way to fulfil these required conditions is to use shape optimization as a second step after topology optimization. By this way, final shaping can be performed to fulfil the additional conditions on durability, uncertainties, and other conditions.

The model of an engine bracket is used to show the complete design process. In particular, challenges and their solutions during this process will be highlighted. All computations are carried out using one industrial FE software (PERMAS).

1. The Task

Since topology optimization has entered the simulation scene, the idea has been circulating that this could be a way of getting rid of the designer and there would be a way to design parts and machines by an automatic computational process. This idea was recently fed by the upcoming 3D printing technology, where former complex production processes are replaced by just one production step. So, why not replacing the complex design process by just one design step?

But, of course, a design process is not so easy and one can obviously expect a longer period until the dream will become true. Among others, the complete specification of conditions, which a part to be designed has to fulfil, might not be waiting just around the corner. Also, mere topology optimization is for sure not sufficient and the required steps of design by simulation have to be carefully selected. Questions remain regarding the robustness of the design found and even its appearance.

From this situation, it seems to be wise to proceed step by step to the bright future. Also, restricting the view to design cases where the function is absolutely dominating will facilitate the implementation of the required design steps. Moreover, taking mechanical parts and assemblies under static or dynamic loading will also reduce the complexity of a general approach. Altogether, starting with simple design cases and going on to more complex ones, is an old but successful strategy of engineering.

This paper describes the process of designing an engine bracket as industrial part under static loading. The proposed process is described first, followed by an outline of the bracket model. The steps of the process are illustrated by the related simulation results of the bracket. A conclusion will complete the paper.

2. The Process

The process proposed in this paper is shown in Fig. 1. The steps of this process are explained in the following.

Topology optimization is the principal function for shape finding. To this end, the area of the part, where topology optimization is applied, has to be defined as ‘design space’. It is good practice to use a finer mesh in the design space to better represent the force flow. For the sake of convenience, we assume that the initial FE model already contains the design space. Then, the setup of the optimization is a pre-processing step followed by the optimization itself.

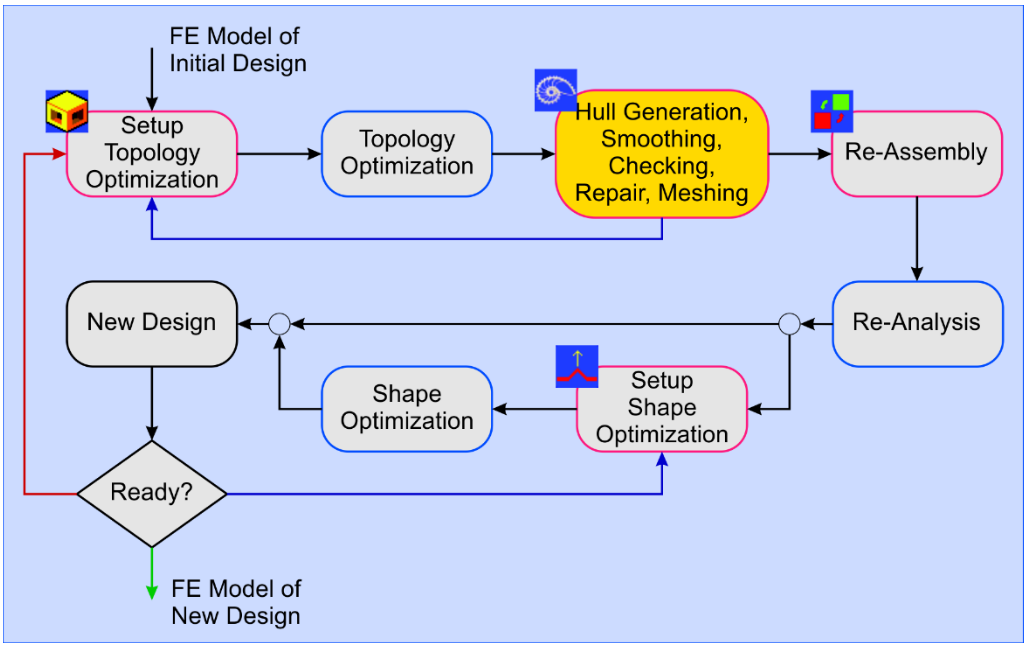

Figure 1: Proposed process to create a new design from an initial design

In static analysis, sufficient loading cases are required to reflect the real operation of the part to be designed. It is good practice to take all coupled parts into account in order to have a realistic stiffness condition for the part to be designed. This could be done by adding the full FE models of those neighbouring parts, or a static condensation of the neighbouring parts could be applied in a substructure approach.

Topology optimization is controlled by the element filling ratio as design variable. A filling ratio of an element near Zero means that this element is not needed to bear the loading. In contrast, a filling ratio near One means that this element is needed to bear the loading. It is essential for the quality of the optimization result that middle values of the filling ratio e.g. 0.5 are avoided, i.e. a Boolean 0/1 distribution is ideal. Then, the optimization result is seen as a fully converged result.

Due to the separation of needed elements (full) and not-needed elements (void), stresses in the design space are not very reliable. The stiffness jumps between ‘full’ and ‘void’ elements lead to stress concentrations which are not best suited for stress optimization by topology optimization. This is reason, why a subsequent shape optimization is proposed in process.

But before a shape optimization can be used, we have to create a new FE model. First, the surface of the ‘full’ elements has to be generated as a hull and subsequently smoothed. Some checking and repair tools are needed to overcome problems from the smoothing process (like very thin structures).

Afterwards, the generated hull has to be meshed, because the old mesh cannot be used any more.

The new mesh of the design space has now to be re-assembled with the rest of the structure. So, connections, supports, loads, and material properties are used from the initial model to create a new FE model. This assembly step is performed by a widely automatized pre-processing step.

The next step is just a re-analysis of the new model with the original loads. This step allows a comparison of the results (like displacements) of the new model with the final results of the topology optimization. The differences should be small, of course. If the results of the re-analysis already satisfy all conditions, the design process can be terminated.

If the results are not fully satisfying or an additional gain in weight is a target, the new model has to undergo a shape optimization. The results of the re-analysis serve as a basis for the setup of the shape optimization. A freeform optimization is suggested, which is particularly suited for stress optimization tasks. So, the relevant surface parts of the new model are selected for the specification of the design space for the shape optimization. In addition, design objective and constraints will be defined. It is for sure useful to use the same constraints as for the topology optimization. Other conditions like for the stresses can be added.

The freeform optimization is modifying the coordinates of the surface nodes of the selected design space. The nodes inside the design space are then moved too in order to maintain the quality of the elements.

After a successful shape optimization, a final check has to be done on all conditions. If this check is a passed, the final design is achieved and its FE model is directly available.

3. Model of Engine Bracket

Fig. 2 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.

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.

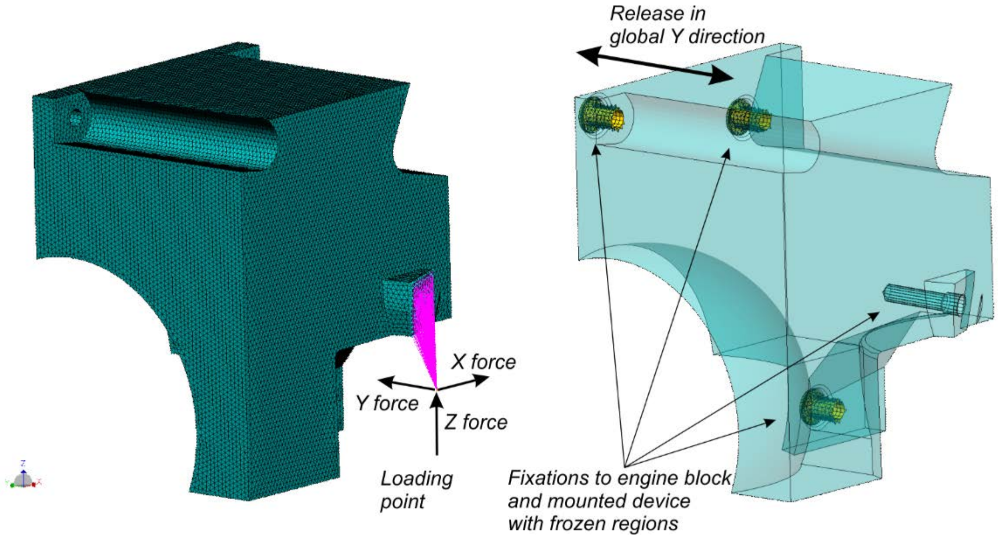

Figure 2: Optimization model of engine bracket

4. Topology Optimization

The first step of the design process is topology optimization. The starting model as shown in Fig. 2 has a much larger volume than the expected result. So, an initial filling ratio of 0.1 has been used. The objective function is compliance, while the weight is used as additional constraint. This weight limit is taken from the weight of the predecessor part. Figs. 3 and 4 show the course of the objective and the constraint, respectively. The optimization used 37 iterations to get the 0/1 distribution of the element filling ratio..

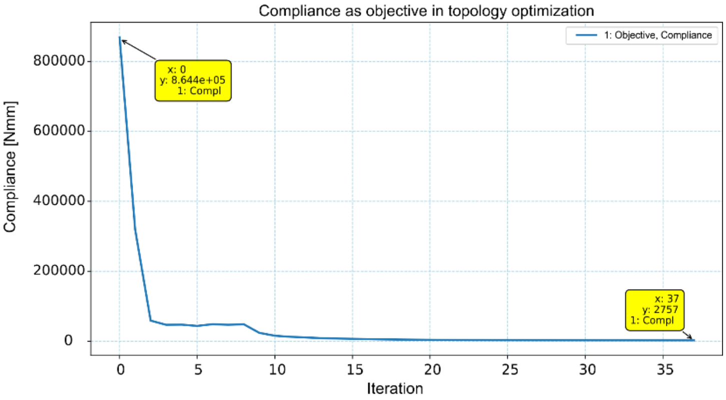

Figure 3: Compliance as objective in topology optimization

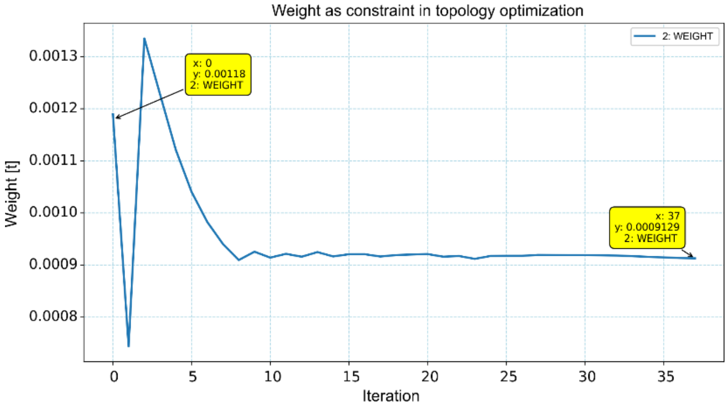

Figure 4: *Weight as constraint in topology optimization*

The primary result of topology optimization is shown in Fig. 5, where all elements with a filling ratio beyond 0.9 are shown together with the length of the displacement vector at the loading point. The displacement is 1.03 mm.

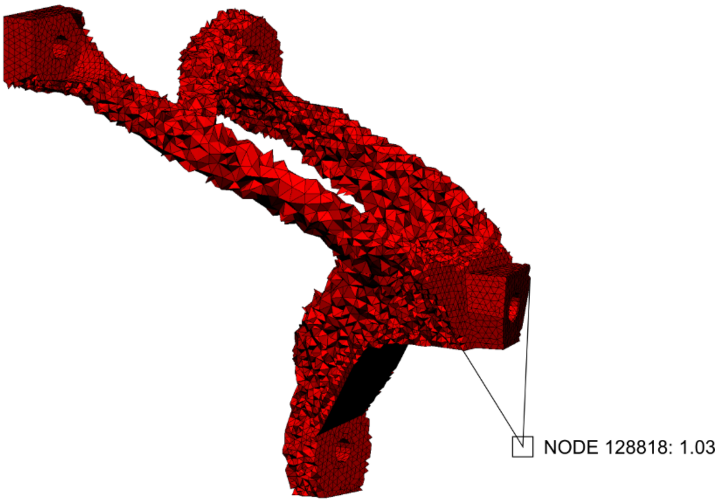

Figure 5: *Topology optimization result showing all elements with filling ratio >0.9*

5. Hull Generation and Remeshing

A design wizard has been created to control the next step of the process, i.e. hull generation and remeshing. To become acquainted with this process step, the design wizard can be used interactively in a graphic environment. Later, when one already feels confident of the result, this process step can be used in an automatized manner. The main points of interest in this process step are as follows:

- The surface level, where the smoothing takes place. This has an influence on the appearance and the weight of the re-meshed part.

- The smoothing level, which determines the quality of the surface.
- The edge length for the surface and solid meshing.
- To create a regular volume (without singularities).

Fig. 6 shows the result of this process.

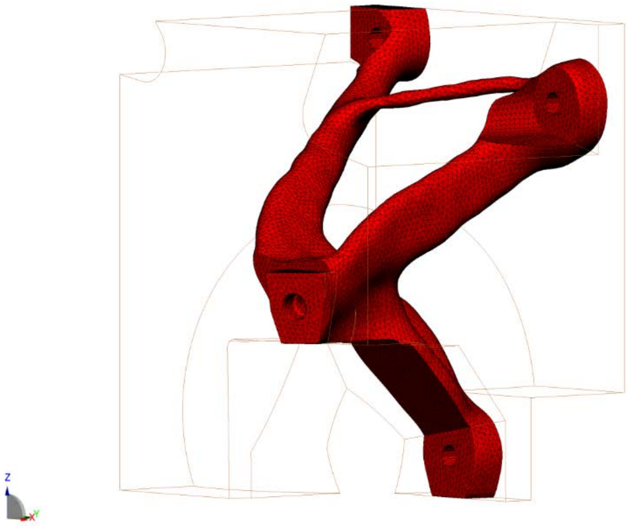

Figure 6: Final shape result after smoothing and remeshing

6. Re-Assembly and Re-Analysis

When having the re-meshed part, then it has to be re-assembled with all other model data from the previous topology optimization model in order to have the same material, boundary conditions, and loads. For this process step, another wizard is provided to handle this assembly.

Then, it is very important to compare the behaviour of the re-meshed and re-assembled part with the result of the topology optimization. By doing so, one gets a better feeling, what the smoothing process did with the part. Fig. 7 shows the displacement result of the re-analysis, which shows a stiffer result with 0.93 mm than topology optimization with 1.03mm (see Fig. 5). The weight of the re-meshed part is 914.4 g compared to 912.9 g from topology optimization (a plus of 0.16%).

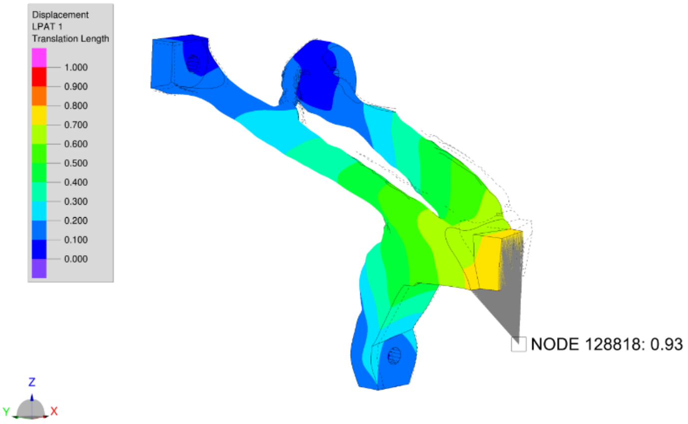

Figure 7: Displacement result of re-analysis

7. Freeform Optimization

To finish the surface, a freeform optimization step follows, which mainly allows to take all kinds of stress conditions into account. But this optimization can also be used for other additional conditions, which could not yet be considered in topology optimization. The method used here for freeform optimization is based on optimality criteria, which is above all best suited for stress adjustment tasks.

For our example, Fig. 6 shows the starting shape of freeform optimization. The weight is used as objective and the first principal stress, the von Mises stress, and the stress gradient normal to the part’s surface are taken as constraints. Fig. 8 shows the weight objective, where the weight could be reduced by about 10%.

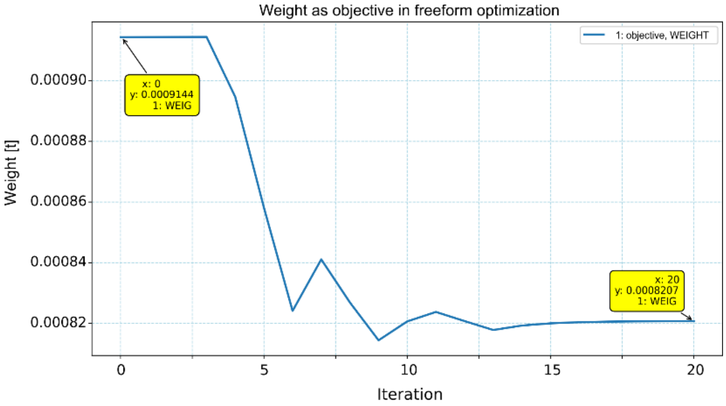

Figure 8: Weight as objective in freeform optimization

The different kinds of stresses are not independent. Therefore, a limit for the von Mises stress of 150 MPa is given and the values of principal stress as well as of stress gradient will be reached automatically. Figs. 9, 10, and 11 show the different stress constraints.

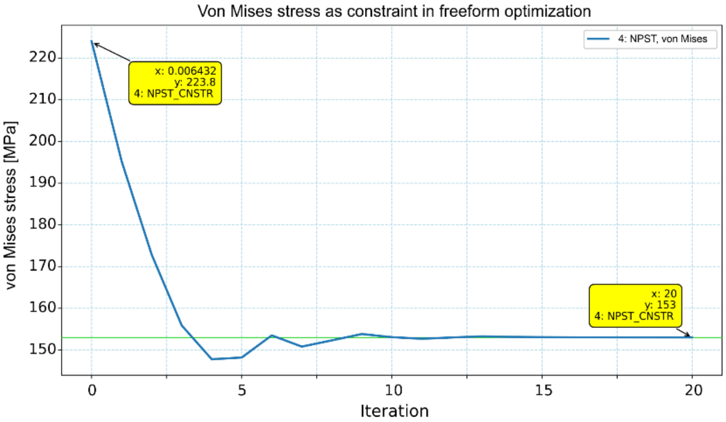

Figure 9: Von Mises stress as constraint in freeform optimization with limit 150 MPa

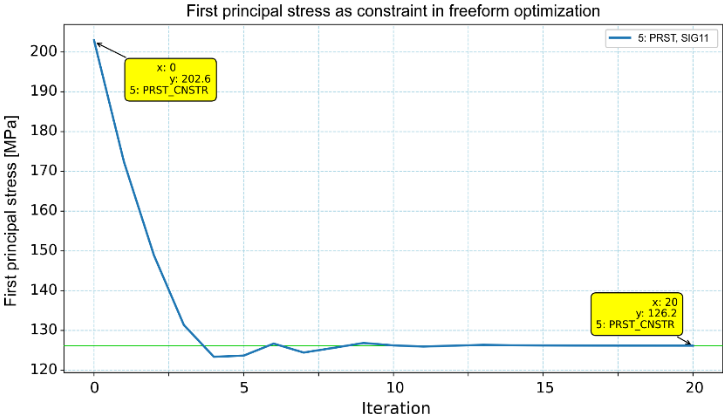

Figure 10: First principal stress as constraint in freeform optimization

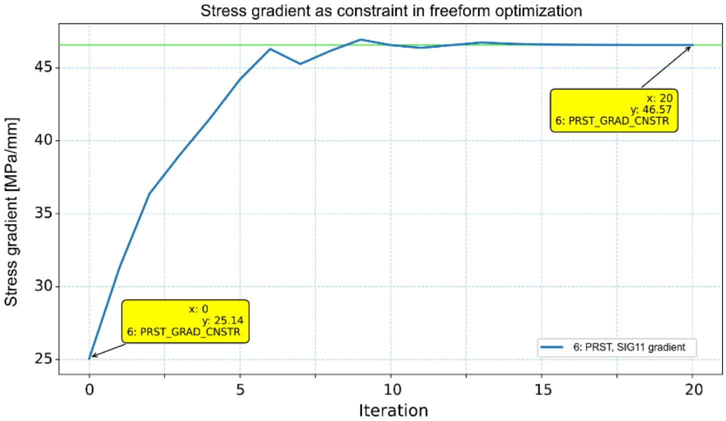

Figure 11: Stress gradient as constraint in freeform optimization

For the shape change a growing or shrinking of ±3 mm has been allowed. Fig. 13 shows the normal coordinate change with a maximum of about 2.3 mm. So, the allowed limit is not exploited. The reason for this is an additional constraint, which indicates the element quality in the design space. The element quality is important, because the coordinates of all elements in the design space are modified in every iteration. In order to avoid a stop of the optimization due to elements becoming erroneous, the element test constraint is used to avoid such situation. All values less than 1.0 indicate elements with acceptable quality. All values equal or greater 1.0 indicate an erroneous element. Here, the element test constraint is limited by 0.9 (see Fig. 12). With a good element quality, the stress results are also reliable. (see Fig. 14).

Finally, we compare the length of the displacement vector at the loading point (see Fig. 14) and find that after freeform optimization the displacement is 0.99 mm compared to 1.03 mm after topology optimization and 0.93 mm after re-analysis. So, the weight reduction during freeform optimization results in a slightly higher deformation than achieved after remeshing but still smaller than after topology optimization.

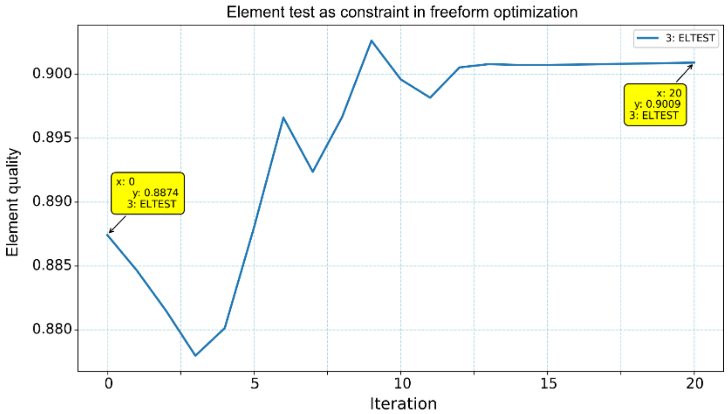

Figure 12: Element test constraint to control element quality

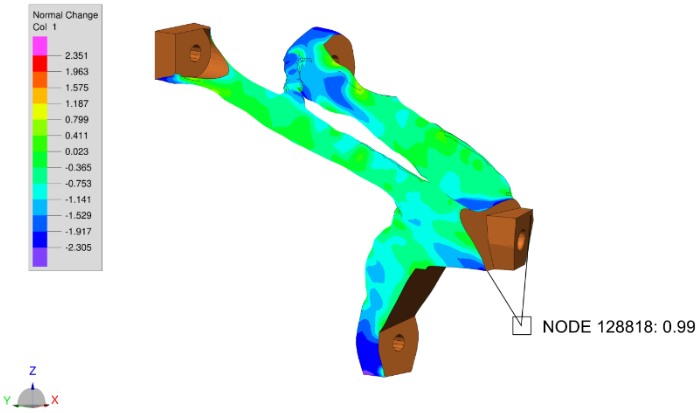

Figure 13: Normal coordinate change

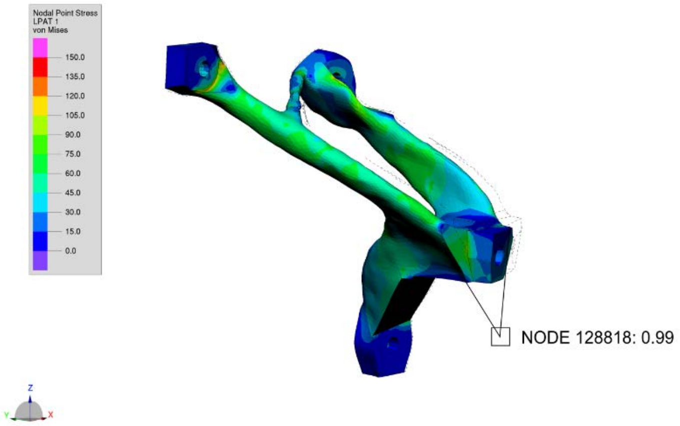

Figure 14: Final von Mises stress distribution with displacement indication at loading point

8. Conclusion

The process used in this paper targets to find a final design by simulation, which fulfils all given requirements and is ready for production (e.g. by 3D printing). After describing all steps of this process, one can say that this target can be achieved. The design wizard plays an important role in this process, because the semi-automatic hull generation and re-meshing saves a lot of time compared to classical parametric CAD re-construction and subsequent meshing processes. The combination of topology optimization and freeform optimization ensures a good result from a stiffness and displacement point of view as well as from a stress point of view.