WEIGHT OPTIMIZATION OF A TRANSMISSION HOUSING

R. Helfrich[1], A. Schünemann[1], S. Weber[2], A. Kienzle[2]

1: INTES GmbH, Stuttgart, Germany, www.intes.de
2: ZF Friedrichshafen AG, Friedrichshafen, Germany, www.zf.com

Abstract: The shape of a transmission housing is derived from function and space of the contained transmission. It can be best characterized by a free geometry stiffened by ribs. From an environmental point of view, the housing provides an oil-tight hull of the transmission. From a structural point of view the housing is a stiff connection between the bearings of the transmission. Due to assembly reasons, housings usually consist of several parts, which are bolted together. Then, after final design, the question is, can we reduce the weight of the housing by reducing the wall thickness of the frequently cast parts of the housing? Additional conditions are a stress limit due to durability reasons and a displacement limit at the bearing positions to keep the overall stiffness of the housing.

The design of a transmission essentially depends on the contact behaviour between gears, in the bearings, and between the bolted parts of the housing. Consequently, a full solid modelling of all parts including gears, bearings, and bolts is required. In addition, the shape optimization of the housing has to use contact analysis as basic analysis method. Due to the free surface geometry of transmission housings, only a nonparametric freeform optimization can be used, which has to be combined with parametric constraints like displacement limits.

The presentation will show the freeform optimization of an industrial example provided by ZF Friedrichshafen AG to reduce weight by wall thickness changes under stress and displacement constraints. The optimization integrates contact analysis in one single software (PERMAS). This drastically reduces the run time for the optimization.

Keywords: Nonparametric shape optimization, contact analysis, weight reduction, stress constraints, displacement constraints, integrated analysis and optimization, Finite Element Analysis (FEA)

1. Introduction

There are two principal approaches for shape optimization in FEA. Either a parametric approach using so-called Shape Basis Vectors (SBV), or a non-parametric approach using optimality criteria. The latter method allows a thickness change at every node of a previously selected surface of the part, 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. This makes the freeform optimization the best approach for complex freeform geometries like housings.

After final design of a transmission, the question is, whether there is a possibility to reduce weight of the housing without changing the designed mechanical characteristics of the transmission. So, weight optimization has to fulfil stress limits in the housing and displacement limits at least. The weight reduction is enabled by a thickness change of the housing walls either on the outer surface or on the inner surface of the housing.

Freeform optimization has been introduced in PERMAS Version 15. Massive extensions have been added in the current PERMAS Version 16 by additional conditions with a new optimization solver. Here, a real industrial example is presented to show freeform optimization with some of these extensions. ZF Friedrichshafen AG kindly agreed to provide a model of a rail transmission for this paper.

2. Basic Model

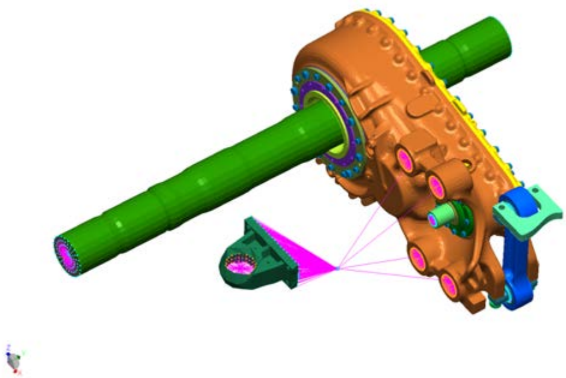

Figure 1: Model of a railway transmission.

Fig. 1 shows the model of a railway transmission. The transmission is directly sitting on the driven axle and it is coupled with the bogie. The driving electric engine is represented by a concentrated mass, which is supported in a suitable manner. Loads are given by bolt pretension, driving torque, and inertia forces. A static contact analysis is performed for this 1 Million elements mesh (TET10) with almost 5 Million unknowns for 10 different loading cases. This analysis runs in about 20 min elapsed time.

Fig. 2 shows the interior of the housing, where all roller bearings are represented by solid models and contact to inner and outer races. Gears are also modelled with solid elements taking into account their interacting forces. All bolts use solid models, too, including the real pretension conditions.

The reason for using a full solid model of the transmission is to represent the realistic stiffness of all parts of the transmission. Therefore, contact analysis between all parts is essential to achieve this target.

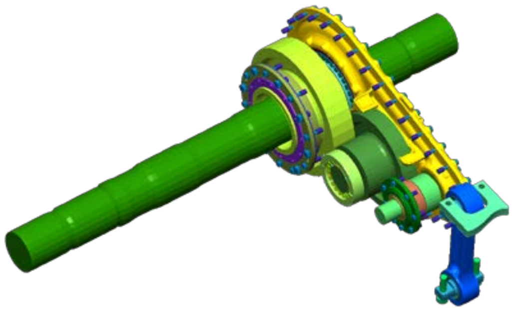

Figure 2: Inside the transmission model.

3. Freeform Optimization

Fig. 3 illustrates how freeform optimization works. The thickness change of a solid structure is achieved by growing or shrinking of the wall thickness perpendicular to the surface. This method requires a relaxation of the mesh in the interior of the solid in order to allow a larger shape change.

On this basis, a typical definition process for freeform optimization has the following steps:

- Selection of surface node set for definition of design space.
- Weight objective for design space.
- Stress limit like von Mises or principal stresses for design space.
- Required displacement conditions to limit local displacements outside of the design space, e.g. the relative or absolute displacements of the bearings of the transmission.
- Required element quality conditions to avoid failing elements during optimization. A good starting mesh quality is helpful.

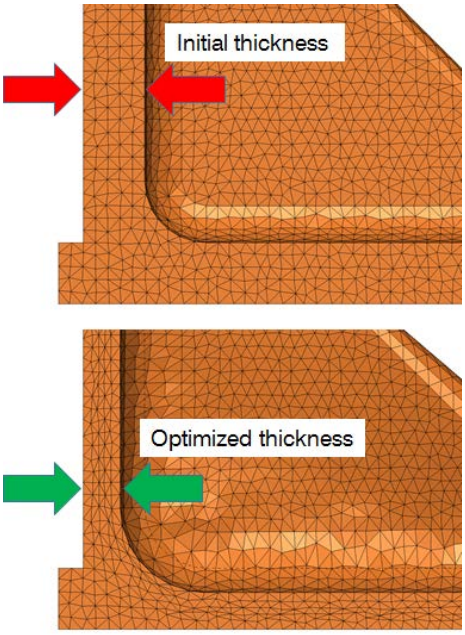

Figure 3: Relaxation of mesh due to thickness change

4. Optimization Settings

The selection of the surface node set for the transmission housing is shown in Fig. 4. The main part of the outer housing surface is taken for modification (top and middle picture). Some inner areas are also selected (bottom picture). The selection has to omit functional and contact regions.

In addition, the following settings were used:

- Minimum initial material thickness is 12mm.
- Assumed thickness change is ±3mm.
- All 10 loading cases are taken into account.
- Assumed von Mises stress limit in design space is 90 MPa.
- Displacement limit is used, which requires no displacement change compared to initial model at selected bearings. Fig. 5 shows an example bearing, where an MPC condition is used to identify the centre of the bearing as dependent node, which should keep the displacements as in the initial model.
- An element quality constraint is used to keep element quality in a safe range. To this end, the classical element tests are combined in one

single number, where a value between 0. and 1. indicates an acceptable element quality, and a value beyond 1. indicates an erroneous element. Here, a limit value of 0.9 has been taken for the design space.

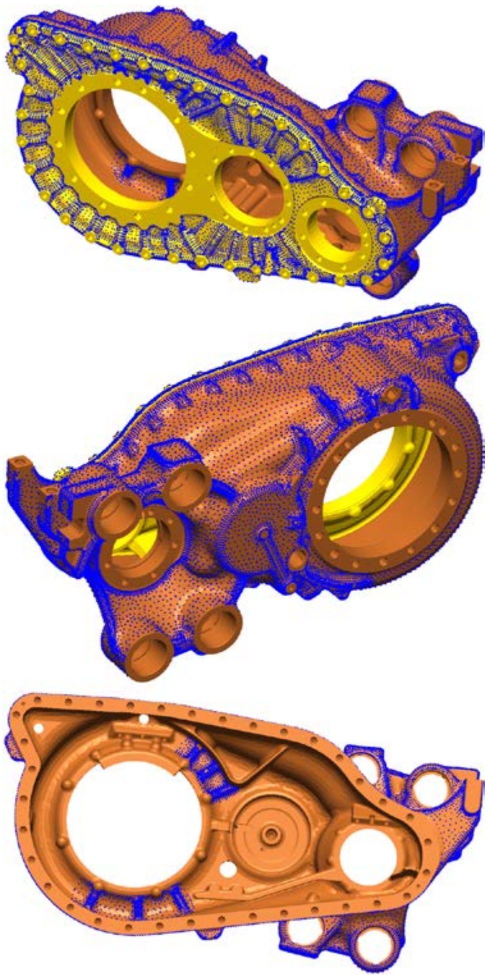

Figure 4: Selected surface node set for design space definition (blue nodes)

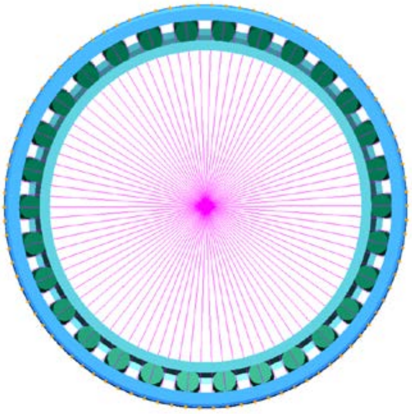

Figure 5: Identification of bearing centre by MPC condition

5. Optimization without Displacement Constraints

In order to see the effect of additional displacement constraints at the bearings, we will show here the optimization results without displacement conditions. Fig. 6 shows the relative history plots for objectives and constraints. Relative history plots show the relation of optimized to initial values. The weight could be reduced by about 9.5%. For the constraints, a deviation of up to 2% is accepted by default. This can be seen from the plots for stress and element quality constraint.

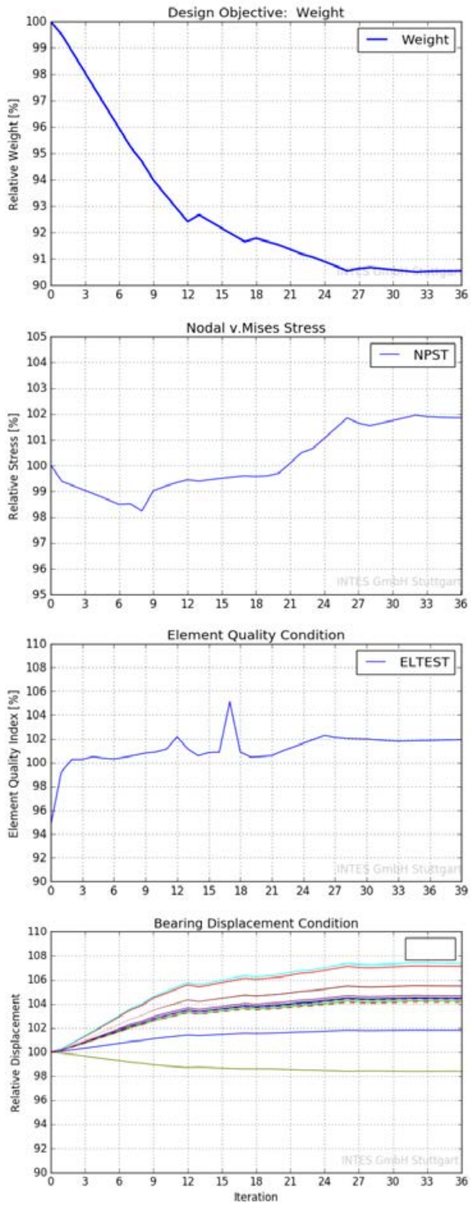

Figure 6: Objective function, stress constraint, element quality condition, and bearing displacement for optimization without displacement condition

The last diagram in Fig. 6 has been generated for comparison reasons. The optimization does not take displacement conditions into account. So, displacements of the optimized housing deviate from the initial design.

To illustrate the result of optimization, Fig. 7 shows the stress distribution for the initial and optimized model for load case 1 together with the normal shape change of the surface over all load cases. Here, the range of thickness changes of ±3mm was not exploited. The range needed was from -2.65mm to +0.68mm.

An interesting effect of thickness reduction is that this leads to higher stresses in the vicinity. To keep the higher stresses in the allowed range, a thickness increase is applied at the location of the high stresses.

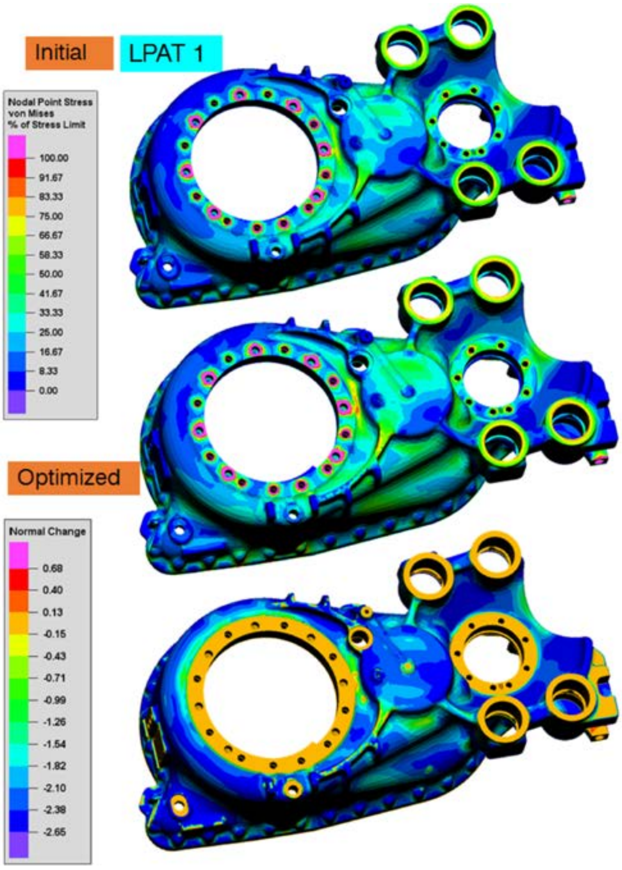

Figure 7: Nodal point von Mises stresses for load case 1 before and after optimization without displacement condition. The bottom picture shows the related normal shape change.

6. Optimization with Displacement Constraints

Now, the bearing displacement conditions are applied and the resulting history plots are shown in Fig. 8. The weight reduction was smaller and reached only about 5.5%. The picture right down at the bottom of Fig. 8 shows the almost perfect holding of the bearing displacements. How this is achieved can be seen from Fig. 9.

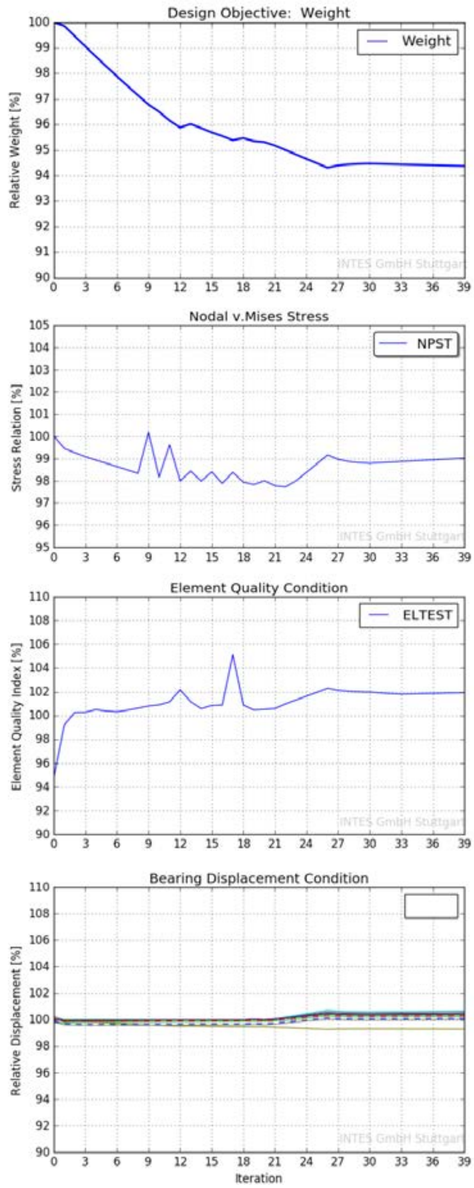

Figure 8: Objective function, stress constraint, element quality condition, and bearing displacement for optimization with displacement condition

Again, Fig. 9 shows the stress distribution for the initial and optimized model for load case 1 together with the normal shape change of the surface over all load cases. Here, the range of thickness changes of ±3mm was not exploited. But the range needed is now from -2.63mm to +2.35mm. While the lower limit is about the same as in the previous case, the upper limit changed from +0.68mm to +2.35mm. This additional material was needed to keep the bearing displacements at the initial level. The reason for not exploiting the lower limit to reduce the weight further lies in the element quality constraint, because a further reduction of the thickness would lead to element quality condition larger than the given limit. By changing this limit, further weight reduction could be possible until the element quality test limits further shape changes.

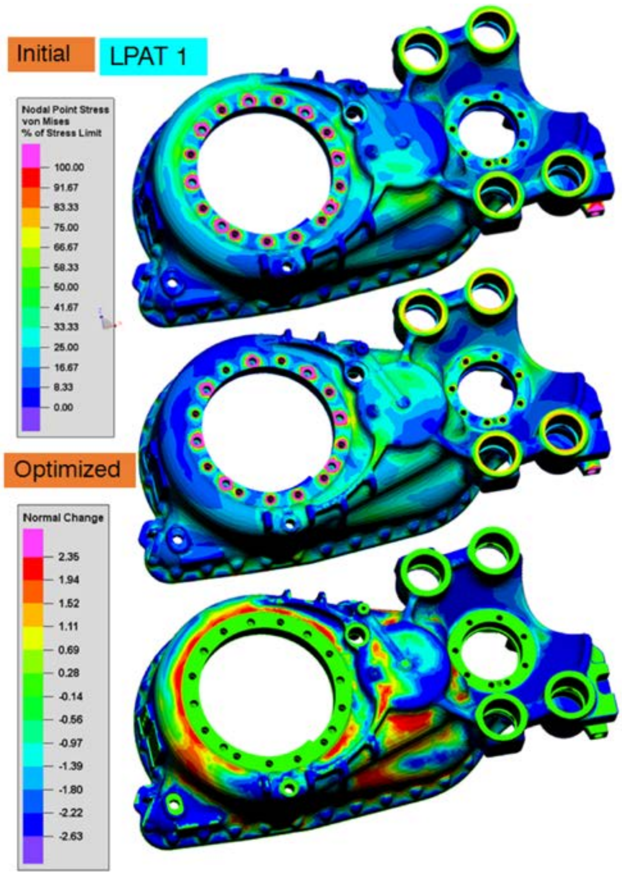

Figure 9: Nodal point von Mises stresses for load case 1 before and after optimization with displacement condition. The bottom picture shows the related normal shape change.

7. Comparison

The weight reduction in the case with displacement conditions is less than in the case without displacement conditions (see Figs. 6 and 8). This is in accordance with the expectation that additional constraints reduce the capability of a structure to lose weight.

The comparison of the bearing displacements shows that it is possible to keep the displacements of the initial design during optimization (see Figs. 6 and 8). Of course, less weight savings are achieved, because some weight has to be spent to increase the stiffness to keep the bearing displacements unchanged.

8. Conclusion

Freeform optimization with additional design constraints opens a new wide field of applications. It is possible to use weight or stress as objective function. Additional constraints are displacements or stresses outside the design space.

The optimization works with nonlinear static contact analysis. Nonlinear material behaviour can be added and even effective plastic strains can be used as constraint or objective function. Additional dynamic load cases are also possible [1].

Single load cases or several loading steps are supported as well. Also, load case combinations to neglect pretension effects in the stress field are possible.

Manufacturing constraints like symmetry or release directions to avoid undercuts complete the functional range of freeform optimization.

A previous topology optimization [2] can be combined with a freeform optimization, where the result of the topology optimization is used to generate a new mesh for subsequent freeform optimization [1,3].

Finally, third party software results can be included in a freeform optimization like fatigue life or safety factors [3].

9. References

[1] R. Helfrich, A. Schünemann: "*Topology Optimization of an Engine Bracket under Harmonic Loads*", SIA International Conference « Automotive NVH Comfort », Le Mans 19-20 Oct 2016.

[2] J. Marchesini: "*Optimisation topologique avec contact - Exemples d'application*", SIA – Journée d'etude – Optimisation topologique, ENSTA PARISTECH - Palaiseau 24 Nov 2016.

[3] M. Schreck, A. Schünemann, R. Helfrich: "*Design and endurance of an engine bracket*", Finite Elements in Engineering Applications, Proceedings of the 13th PERMAS Users' Conference, Stuttgart, Germany, 14-15 April 2016.