MSC's 1st Worldwide Automotive Conference
Munich 1999

Optimization of Car Components using MSC/CONSTRUCT

Dr. F. Dirschmid
IABG, Einsteinstr. 20, 85521 Ottobrunn, Tel.: (089) 6088 2258, dirschmid@iabg.de

Abstract

New CAE tools are necessary to fulfil the increasing demands for new car components. Typical demands are lightweightness, costs and developing time. Efficient tools and methods have to be used in the very early stage of development to achieve these goals. As soon as the design space, the boundary conditions and the loads are fixed, the topology optimization is an appropriate method to define a first design of the component. The topology optimization is a method that change the density and stiffness distributions in an iterative process to achieve a homogeneous stress distribution. This design proposal must be transferred in a real component by taking into account manufacturing and design points of view. The industrial application of this optimization method using MSC/CONSTRUCT is demonstrated in this paper for different car components.

1 Introduction

At the beginning of the conventionel design process the design engineer defines the shape and the topology of new components using the experience and the results gained from the forerunner. This results in an evolution process which might lead to an optimum design after some iterations and a long period.

Nowadays it is necessary to shorten the development process of new components. Therefore tools are necessary that replace the natural evolution process by an automatic procedure. With MSC/CONSTRUCT it is possible to carry out topology- and shape optimization in the CAE environment. The topology optimization finds an original load balanced material distribution. It is only necessary to know the design space, the boundary conditions and the different loads. With the results of this optimization the design engineer is now able to define a detailed design. Shape and parameter optimization can now be used to minimize local stress maxima. Figure 1 summarizes the whole design process.

There is a wide range of applications for topoloy optimization in automotive industry. Typical examples are suspension components like drag bearings, suspension drags and wheel rims, engine and gear box parts like flexible discs, tension jacks or brackets and car interior and sheet metal components. In the case of sheet metal components it is possible to define the location of beaks or welting spots.

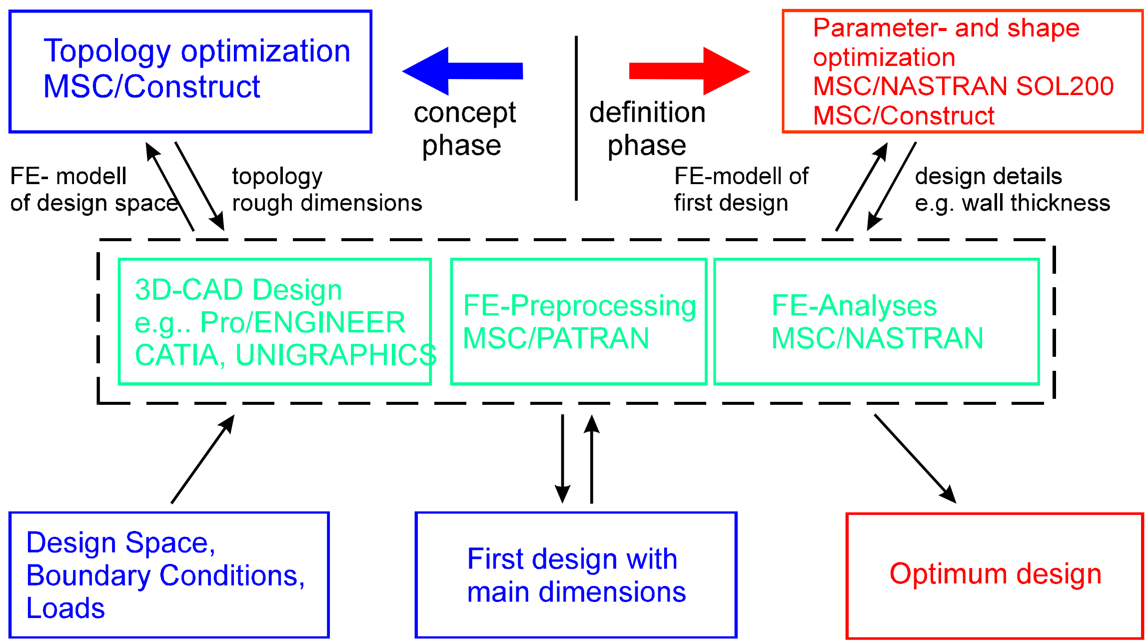

Figure 1: Design process

2 Topology- and Shape Optimization within MSC/CONSTRUCT

Both topology- and shape optimization are based on the theory, that a structure is optimal when the stress destribution is homogeneous [1]. During the topology optimization the Youngs modulus and the density of elements with low stresses are reduced. This results in a „black/white“ structure, with elements having the original Youngs modulus and elements with a almost zero Youngs modulus.The shape optimization changes the surface geometry in a manner that a homogenous stress distribution is achieved. Both methods are parameter free optimization methods contrary to the traditionell parameter optimization. Parameter optmization changes parameters like wall thickness, rib height and fillets radius in order to minimze or maximize objectives like stiffness, stresses or eigenfrequency.

2.1 Topology optimization

MSC/CONSTRUCT version 2.5 offers two different optimization goals:

- absolute or relativ volume
- global stiffness

When the first option is selected the user has to specify the desired weight of the structure. Typical values are between 15% to 80% of the design space weight. Then MSC/CONSTRUCT finds a structure that has a homogeneous stress distribution and consequently a very high stiffness.

It is also possible to define restrictions in order to take into account manufacturing or assembling points of view. Following restrictions are possible:

- Definition of elements that are not included in the design space
- Definition of frozen elements
- Definition of priorities for element elimination
- Definition of elements where element elimination starts

For a better understanding of this method the optimization of a disc is presented. Figure 2 shows the design space of the disc. The disc is constrained at the center. At the outer ring that is not included in the design space tangential forces are acting clockwise and anticlockwise.

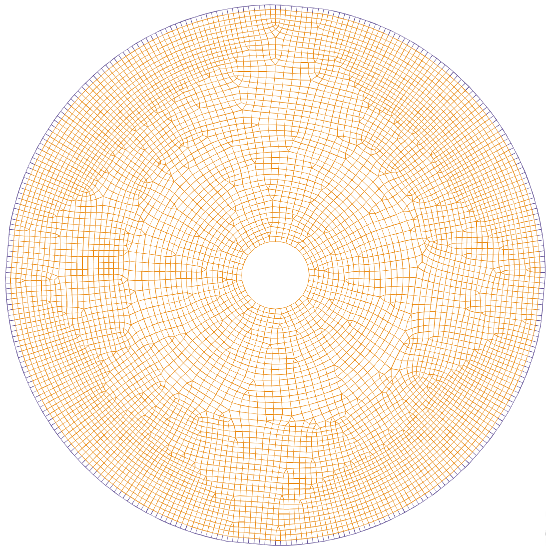

Figure 2:Design space of the disk

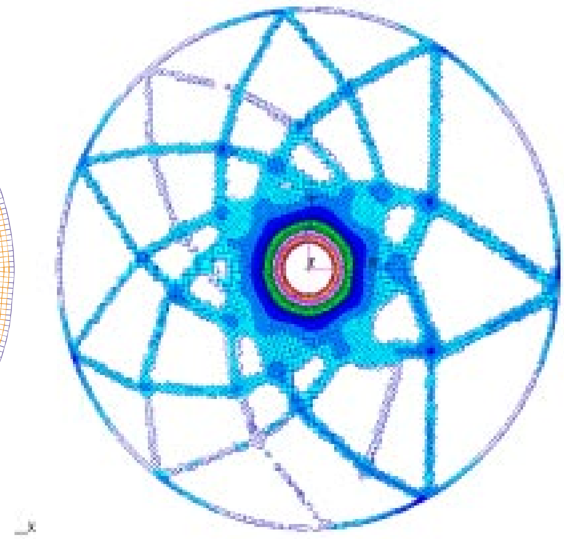

Figure 3: Stress distribution of the topology optimized structure

In figure 3 the result of the optimization is presented. One can see that the design has a very homogeneous stress distribution. Only the center of the disk has higher stresses because of the smaller area at the center.

Volumes can be meshed either automatically by using a tetrahedral mesher or manually with hexahedrons. The first method results in a high number of elements. Additionally a tetrahedron mesh has locally artifical stiffnesses that influence the result of the topology optimization. The second method is very time consuming and in most cases not practible.

A very robust and fast method to mesh the design space is to use voxel technique [2]. It is a method that uses equal cubes to mesh the volume. The only disadvantage is that the surface is not represented exactly but in a stepwise manner. In topology optimization, however, this is of no importance as the result of the optimization process is always stepwise.

As an example the voxel meshing of a sphere is shown in figure 4. In the left half the regular starting mesh is shown. In the right half all elements outside the design space are removed which result in a stepwise approximation of the sphere.

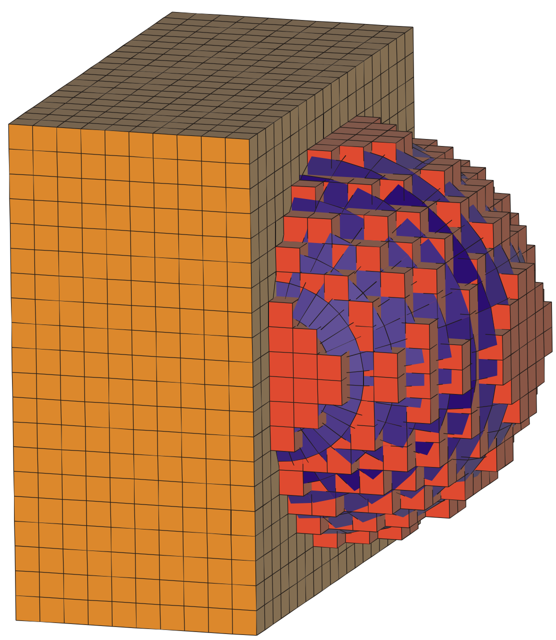

Figure 4: Sphere meshed with voxel elements

2.2 Shape optimization

Shape optimization is used to change the surface (3D) respectively the boundary (2D) of a part. During the optimization the geometry is changed in order to minimize local stress peaks. In principle any finite element model can be shape optimized.

Like in topology optimization several restrictions can be defined:

- Definition of nodal displacement direction

- Definition of nodal coupling
- Definition of maximum nodal displacement
- Limited nodal displacement desfined by surfaces or solids

A simple example is shown to demonstrate shape optimization. In figure 5 (left) the finite element mesh and the stress results of a L-bracket with a fillet are shown. The structure is constrained at the top and loaded by a vertical load at the right end. As one expects there are stress exaggerations at the fillet of about 190 N/mm².

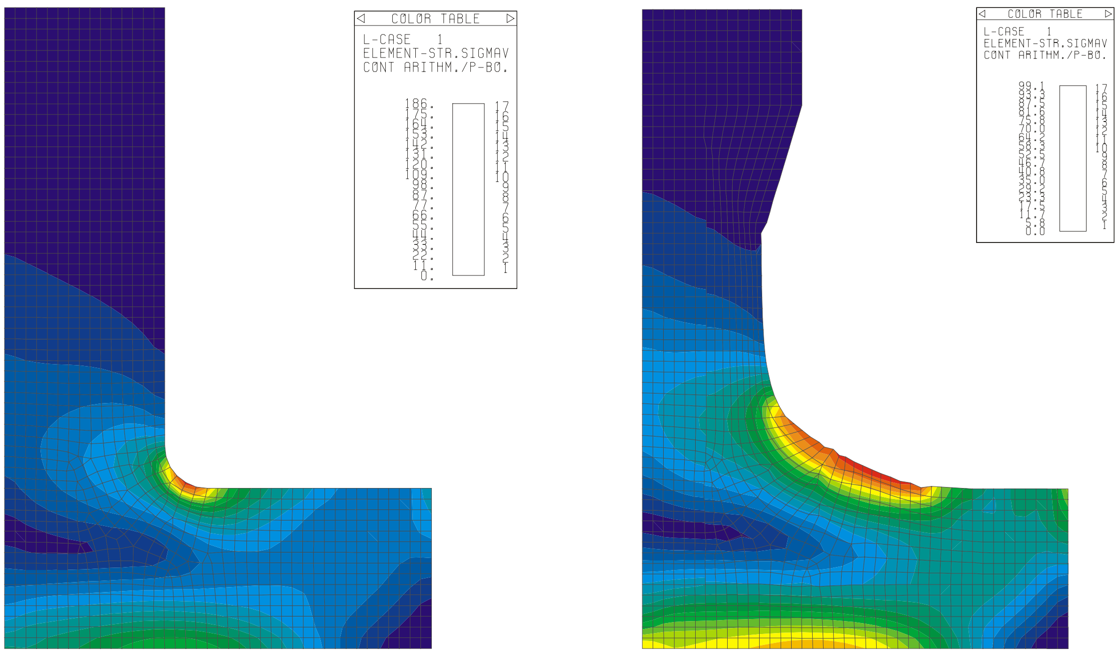

Figure 5: Stress distribution in the bracket before (left) and after (right) shape optimization

The stresses are homogenized by the shape optimization. The local stress maximum is reduced to 100 N/mm². The new geometry and the stress distribution are shown in figure 5 (right).

3 Examples

In the following section three examples are presented. In the first example a bracket is optimized using topology- and shape optimization. The second example shows the optimization of a tension jack. The third example, a seat back rest, demonstrates the interaction of topology- and parameter optimization.

3.1 Weight - and stress optimzation of a bracket

The opbective of this optimization is to find a design for the bracket that has a weight of 50 kg and a maximum stiffness. Five loadcases should be considered. The design space has a weight of 123 kg. Figure 6 shows the structure and the loadcase at which the structure has the highest loads.

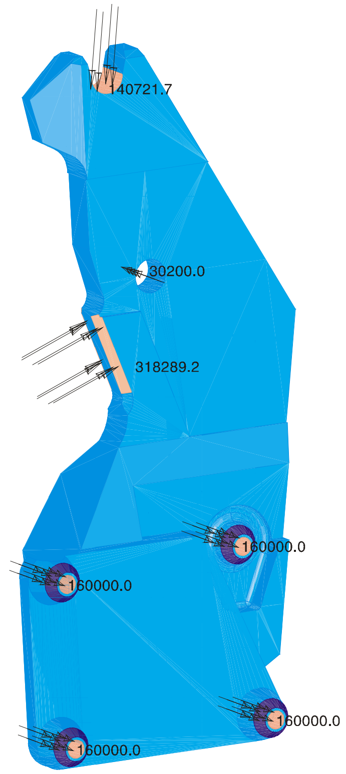

Figure 6: Bracket - loadcase 4

The design space was meshed with the voxel techique described above. The mesh is shown in figure 7. The red areas are frozen elements that cannot be removed during optimization.

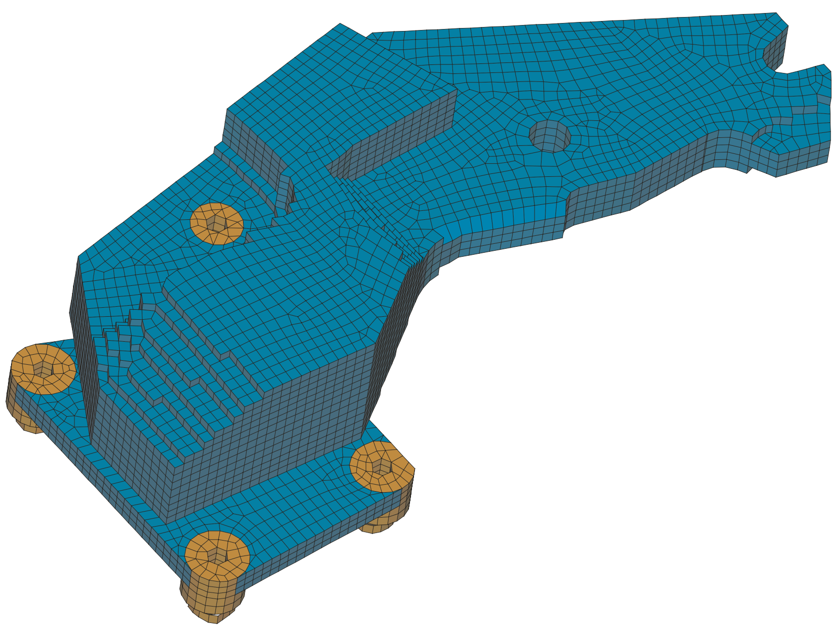

Figure 7: Meshed design space of the bracket

For the optimization a goal of 35 % of the original weight is choosen. The stresses are evaluated by the von Mises stress hypothesis.

The optimization carried out 22 iterations. The result is shown in figure 8. Figure 9 shows the corrresponding stress distribution for the final structure. The stress values are choosen for each element from the maximum of all loadcases. As can be seen the stress distribution is homogenized over the whole structure. Only at the fillet there are still high stresses of aboout 790 N/mm².

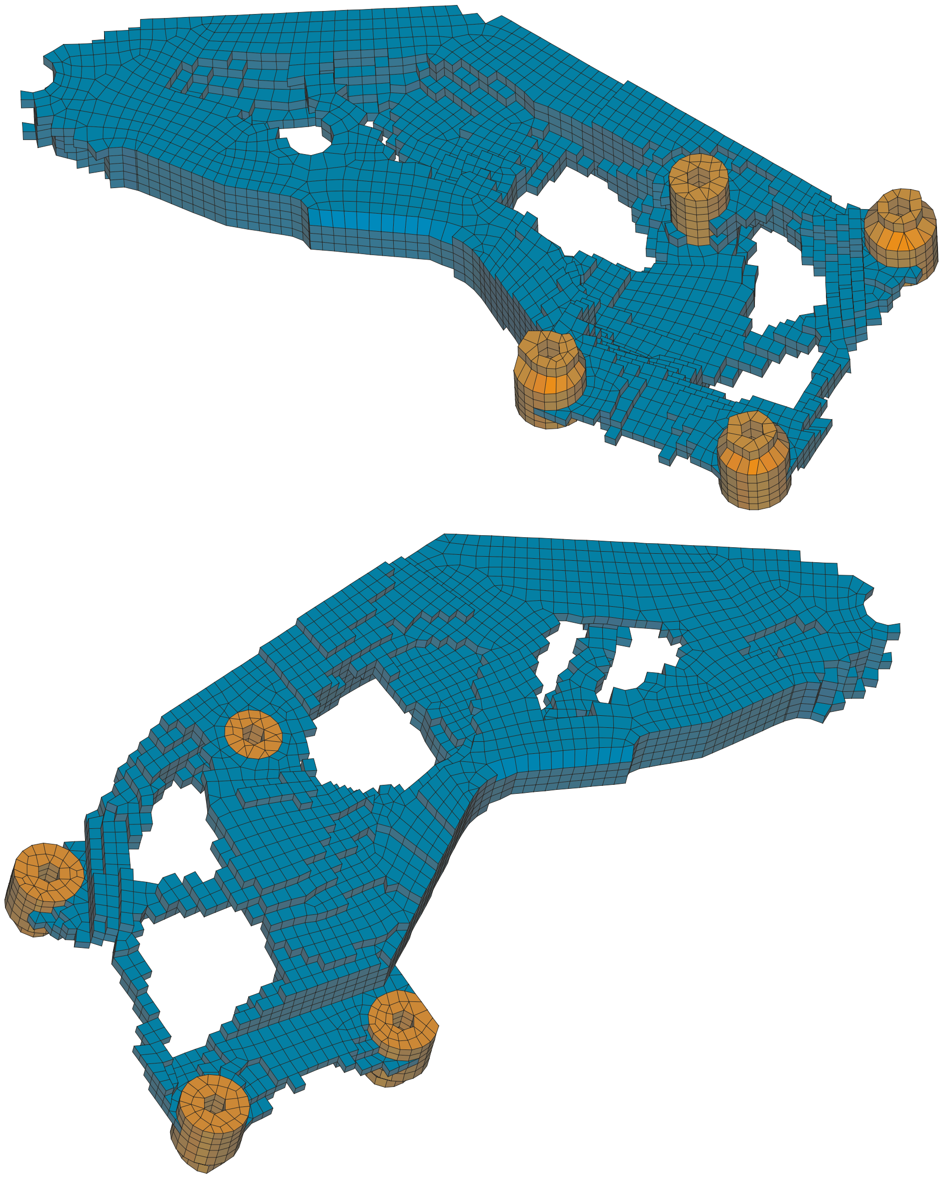

Figure 8: Result of topology optimization

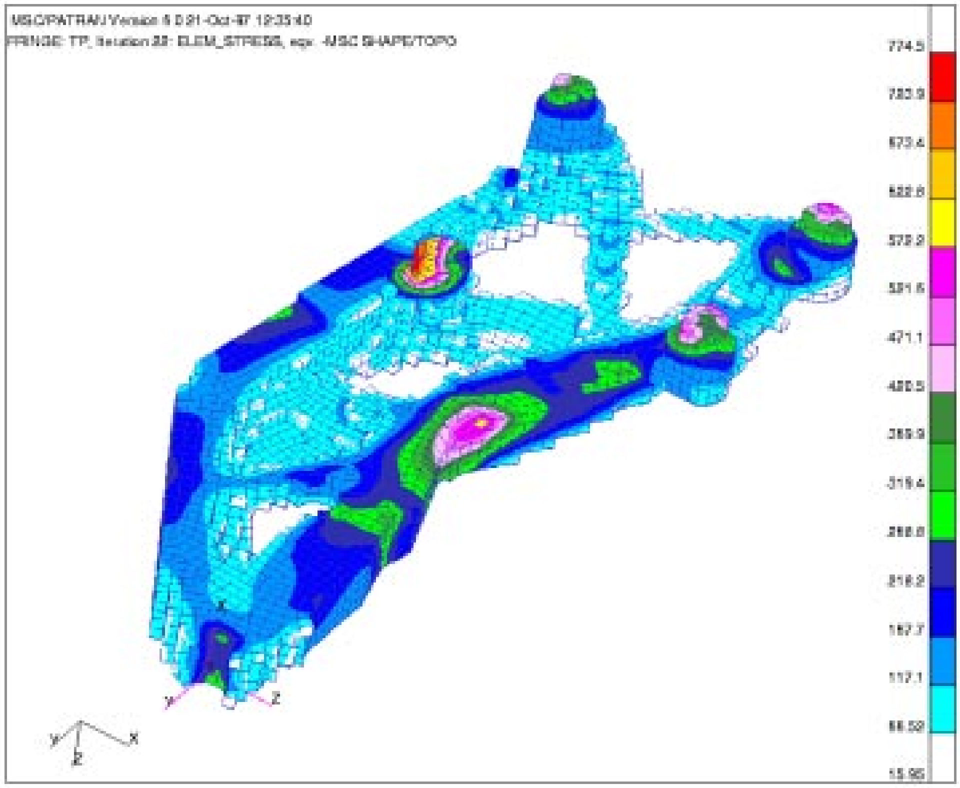

Figure 9: Stress distribution of optimized structure

The high stresses at the fillet can be minimized by a shape optimzation. For this optimization the resulting mesh from the topology optimization is used. The goal for the optimization was to allow a maximum stress of 600 N/mm². Restrictions where used to get a plain fillet over the height and to disallow any displacement outside the starting geometry. Figure 10 shows the optimization result. The stress peak could be reduced to 666 N/mm². Figure 11 shows the difference in the stress distribution before and after optimization.

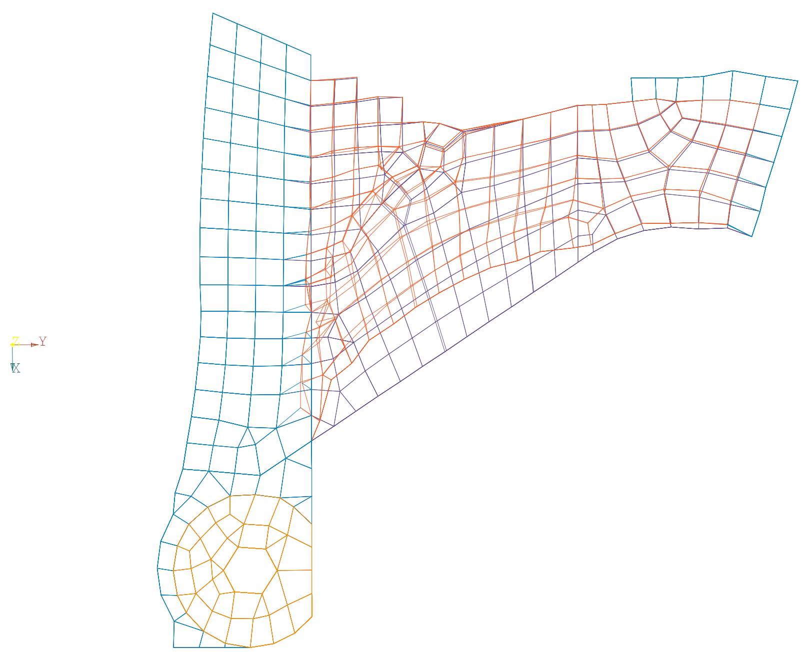

Figure 10: Result of the shape optimization (new geometry red)

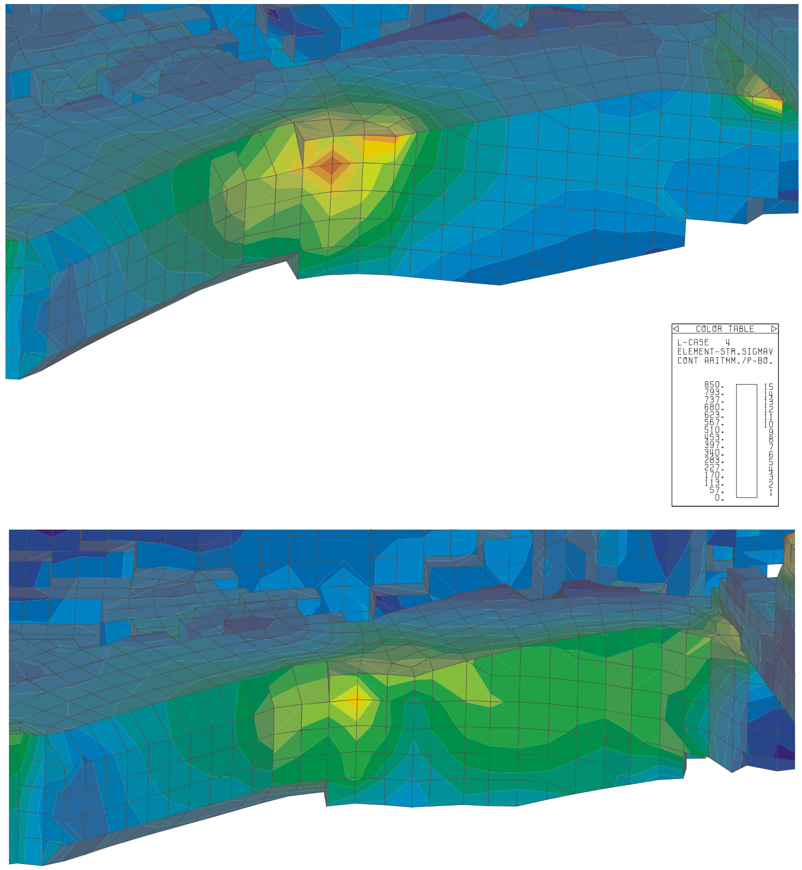

Figure 11: Stress distribution before (above) and after (below) shape optimization

The conversion of the topology and shape optimization results in a real CAD design is shown in figure 12. The final design is like in most cases a compromise of the optimization results and the manufacturing process. Typical restrictions for casting and forging are the minimum and maximum wall thickness or the position of holes.

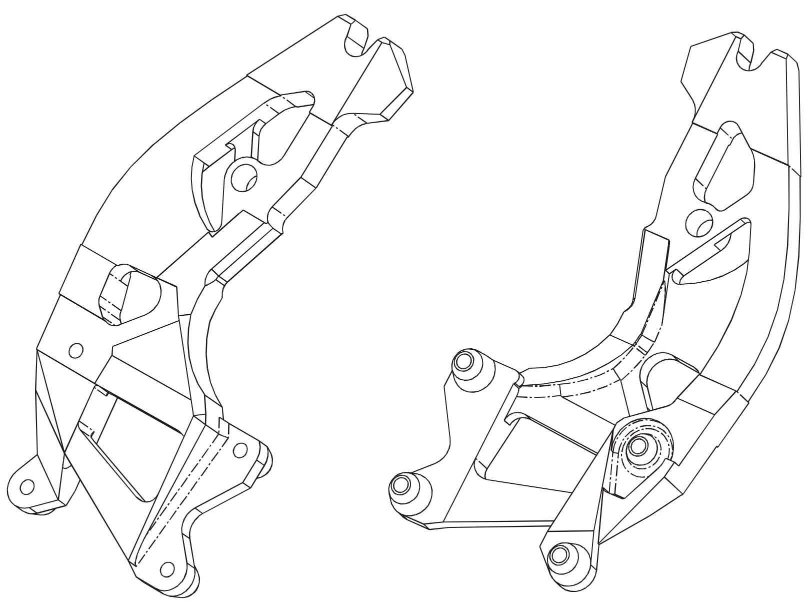

Figure 12: CAD-design of the bracket

3.2 Stiffness optimization of a tension jack

The next example demonstrates the optimization of an engine component. For a tension jack the stiffness is an important criterium. To optimize the stiffness a topology optimization is carried out. Figure 13 show the design space of the component.

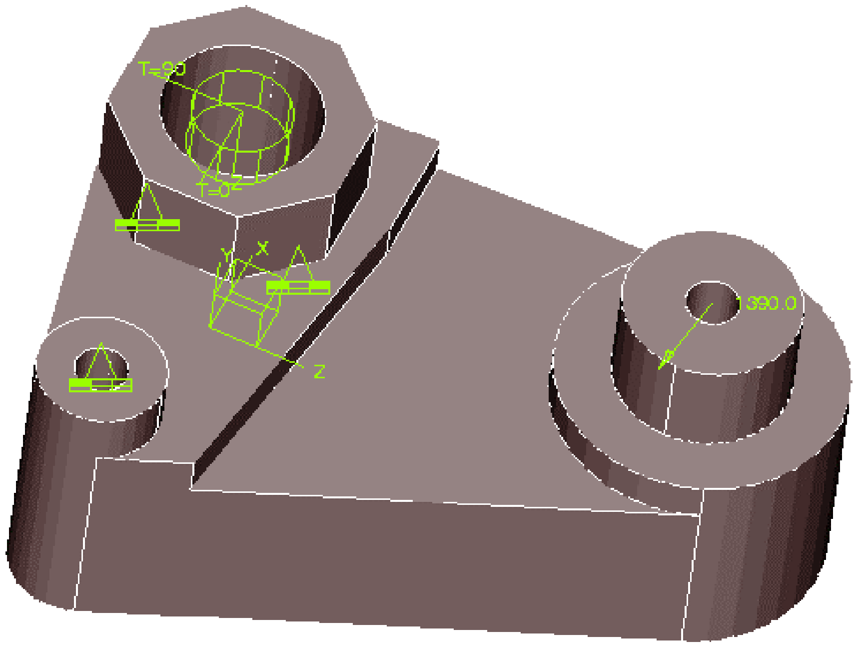

Figure 13: Design space of tension jack

The result is shown in figure 14. For a better visualization of the optimized design the result is smoothed. Within MSC/PATRAN it is now possible to export the geometry as a vrml-file. This is a neutral 3D geometry description format that can be visualized with simple viewer (e.g. internet browser).

The reult of the CAD conversion is shown in figure 15. Again, a compromise between the optimization result and manufacturing points of view has to be found.

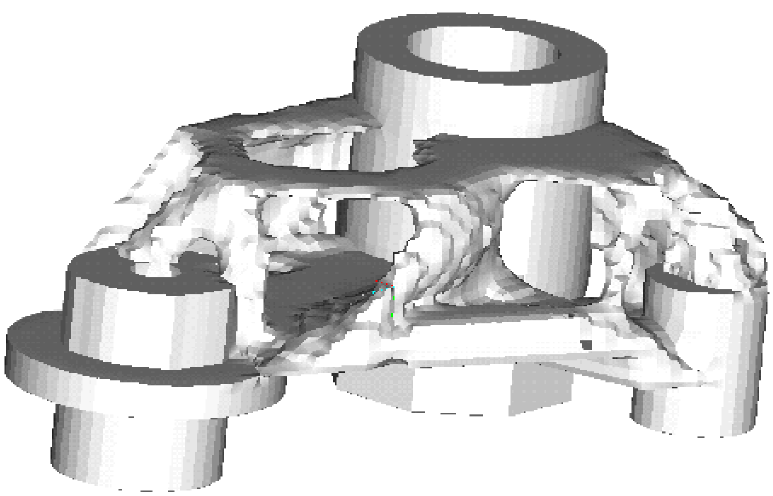

Figure 14: Result of topology optimization

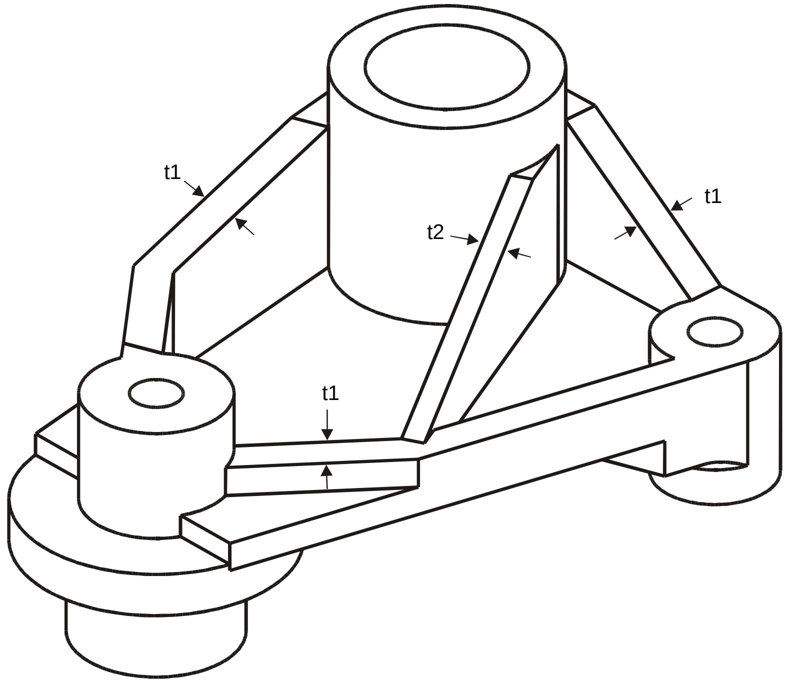

Figure 15: CAD-design of tension jack

3.3 Weight optimization of a seat back rest

This example demonstrates the combined application of topology and shape optimization.

Figure 16 shows the design space and the boundary conditions of the seat back rest. The loads result from maximum peak loads during frontal and rear crash. The weight of the design space is 380 kg assuming a aluminium structure. The desired weight is 8 kg. The maximum stresses should be below 500 N/mm². The minimum sheet metal thickness is 0.5 mm.

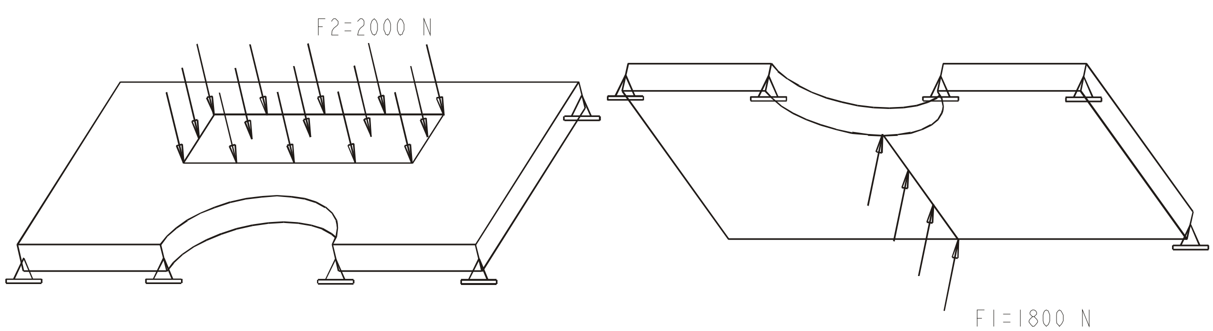

Figure 16: Model of design space

An optimization in one step is not possible because of the low ratio of 2 % of desired weight and design space weight and the small wall thickness of 0.5 mm. Both points would presuppose a very fine mesh and as a consequence in an unacceptable high number of elements.

Therefore a multistep approach is used. In the first step the result of a topology optimization of the volume was transferred in a surface geometry. This structure is optimized with the help of a parameter optimization. Then an additional loop consisting of topology and parameter optimization is performed. For the parameter optimization SOL 200 of MSC/NASTRAN was used [3]. Because of the symmetry only half of the strucure is examined.

- **Step 1: Topology optimization of design space**

Objective in this step was to reducea weight to 30 % of the design space weight.

Figure 17 shows the results of the topology optimization after 24 iterations. The result consists of a front and a back plate, ribs at the boundary and a curved center rib.

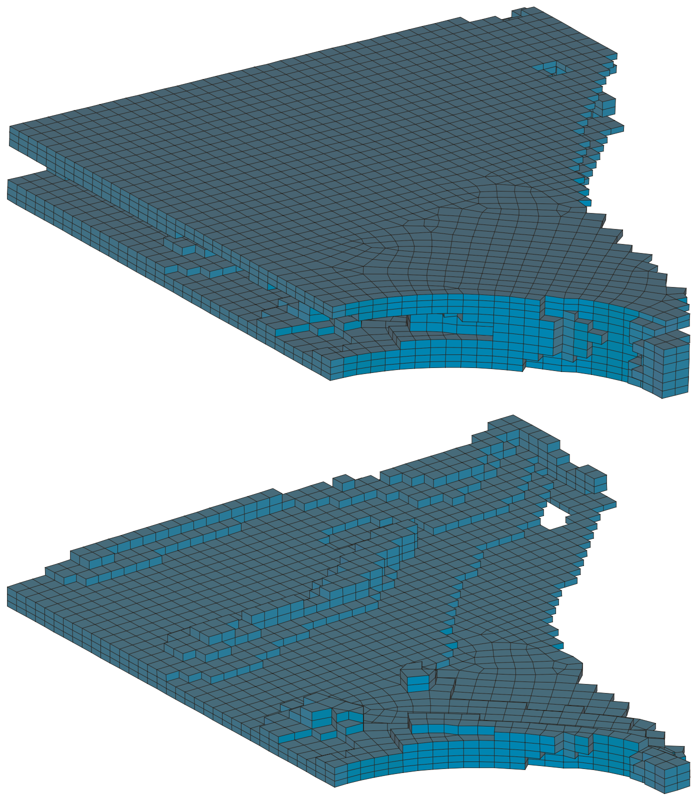

Figure 17: Result of first topology optimization - total structure (above) and half structure (below)

This result are transmitted into a shell representation of the structure (Figure 18). The thickness of all shells is presumed 1.0 mm. The weight of the structure is now 9.8 kg.

- **Step 2: Parameter optimization of wall thicknesses**

The design parameters of this optimization step are the wall thicknesses of the front and back plate including the boundary ribs, the rib at the curvature and the center rib. The thickness can be changed between 0.5 mm and 10.0 mm. Goal is to minimize weight under the restriction of maximum stresses of 500 N/mm².

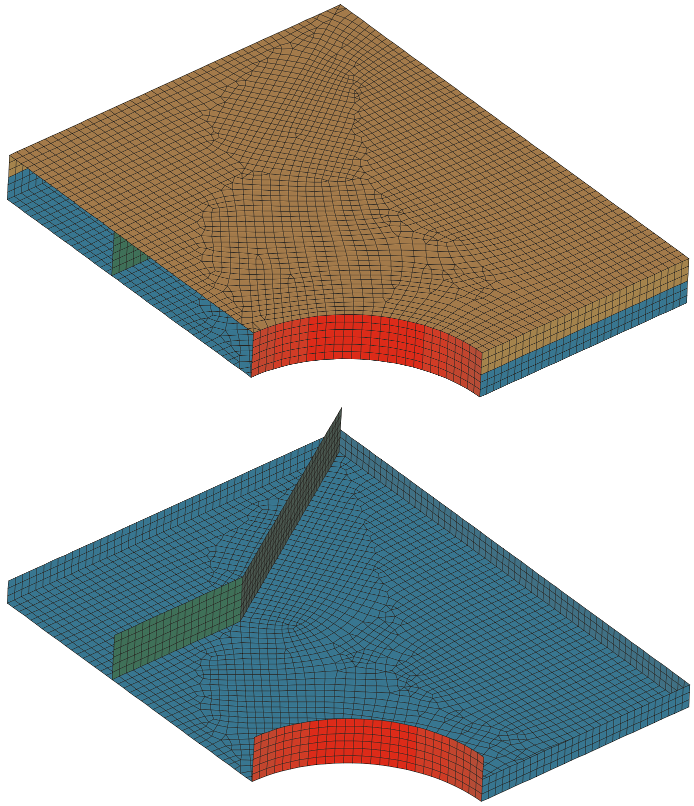

Figure 18: First design of surface model - Total structure (above) and structure without front plate (below)

Table 1 summarizes the result of the parameter optimization.

	before optimization	after optimization
thickness: front plate	1.0 mm	1.31 mm
back plate	1.0 mm	0.88 mm
center rib	1.0 mm	0.50 mm
curvature rib	1.0 mm	1.12 mm
maximum stress	595 N/mm²	500 N/mm²
weight	9.84 kg	11.5 kg

Table 1: Results of first parameter optimization

The original structure violates the stress restrictions. The parameter optimization reduces the stresses to the permitted value, simultaneously increasing the weight to 11.5 kg.

To further reduce weight an additional topology optimization is carried out.

- **Step 3: Topology optimization of the shell model**

Goal of this optimization is to reduce weight to 7 kg or 60%. Figure 19 presents the result of the new geometry and the stress distribution. The stresses are now below 500 N/mm²

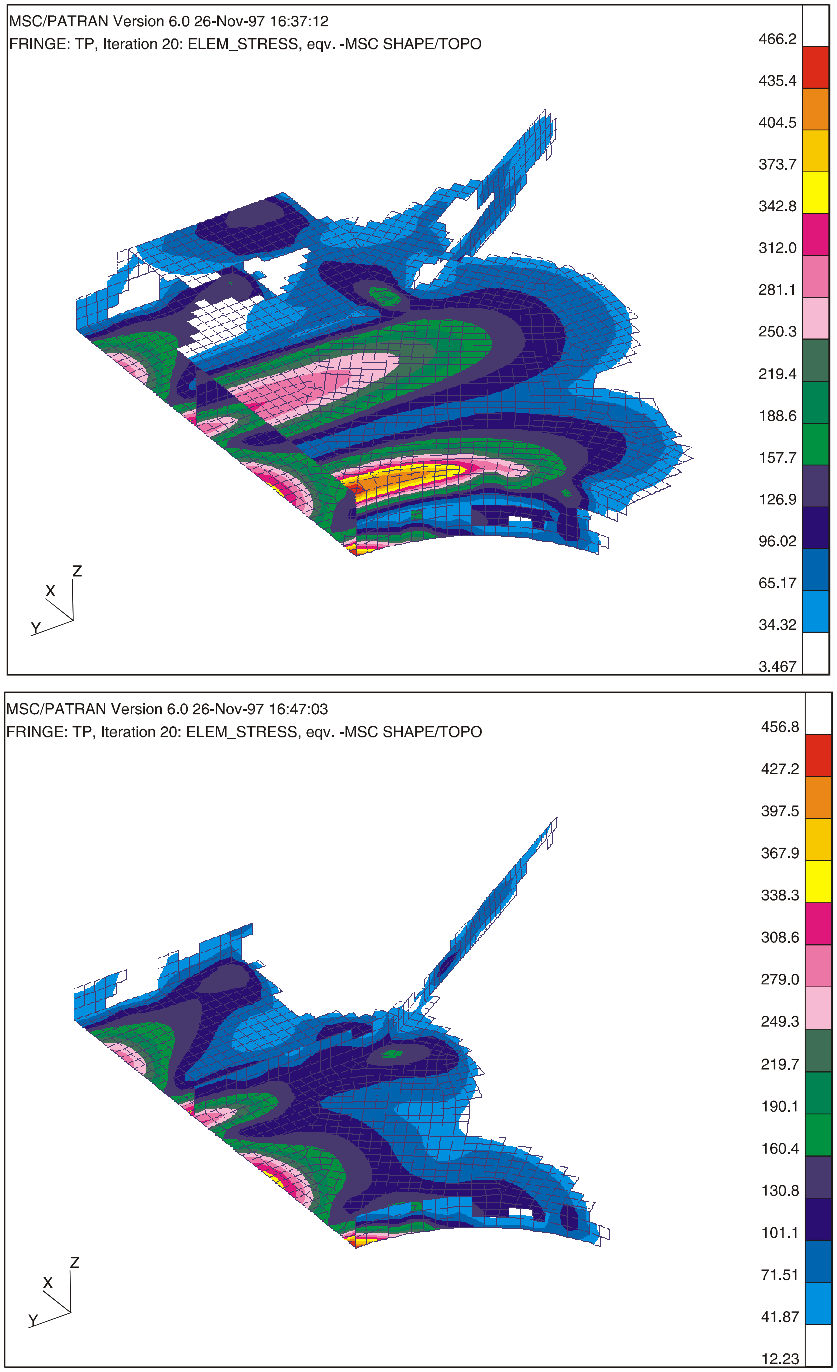

Figure 19: Result of second topology optimization (geometry and stress distribution)

The last topology results are the basis for the end design that is shown in figure20. Again the design does not represent exactly the topology result, but takes into account manufacturing aspects (straight lines).

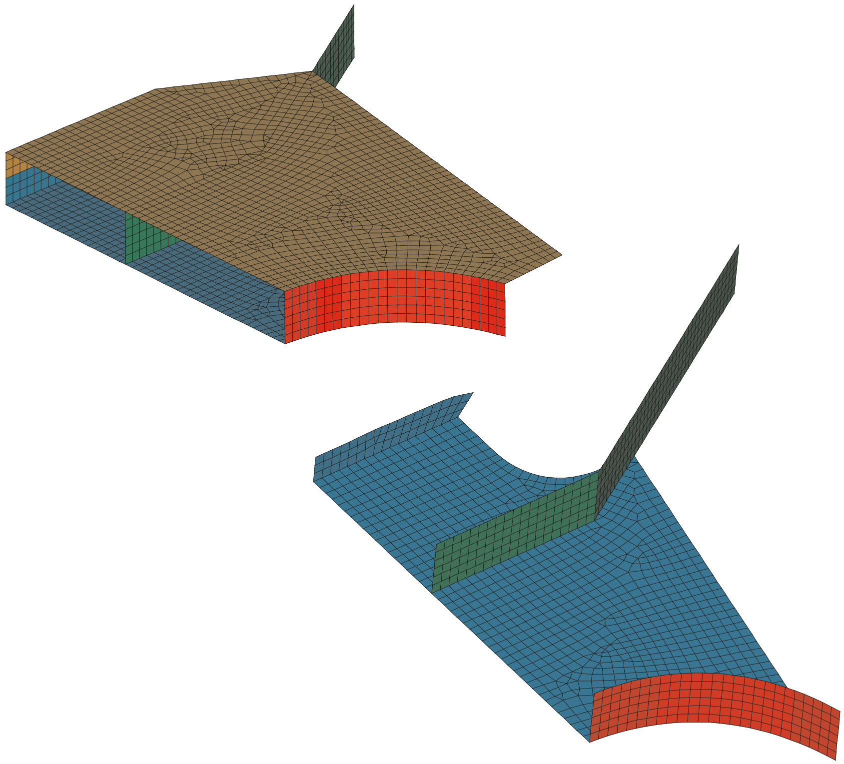

Figure 20: End design - total structure (above) and structure without front plate (below)

- **Step 4: Parameter optimization of sheet metal thicknesses**

In the final step a parameter optimization is carried out again. The same design variables are used as in the first one. The results are shown in table 2.

	before optimization	after optimization
Thickness: front plate	1.31 mm	1.47 mm
back plate	0.88 mm	1.38 mm
center rib	0.50 mm	0.50 mm
curvature rib	1.12 mm	1.12 mm
maximum stress	680 N/mm²	500 N/mm²
weight	4.94 kg	6.1 kg

Table 2: Results of second parameter optimization

Now the structure fulfils all design criteria. The weight is below 8 kg, the maximum stress does not exceed 500 N/mm², the sheet metal thickness is not smaller then 0.5 mm and the manufacturability is granted.

4 Conclusion

The topology optimization has the highest importance in the developing process of all optimization methods as it interferes in the very early phase of the design process. Once the originally design, the manufacturing process and the material are defined it is only possible to achieve improvements with a great effort with respect to time and costs.

With version 2.5 of MSC/CONSTRUCT it was only possible to use SOL 101 of MSC/NASTRAN. With version 3 it is now possible to use SOL 103 for dynamic analyses. In the future it would be important to provide more objective functions, like minimum weight under a maximum allowed stress. From the manufacturing point of view it is of great significance to take into account more manufacturing restrictions like to forbid holes in the topology result.

At this place it is pointed out that optimization can not overcome physics. Especially in the case of topology optimization the stiffness of the design space is always reduced through the elimination of elements and the stresses of the design space are always increased by the optimization. This means that if the design space does not fulfil the requirement already then a structure with less mass will also violate the requirements.

5 References

[1] *MSC/SHAPE*, *MSC/TOPOLOGY User´s Guide* V2 Beta, MacNeal-Schwendler GmbH, München, FE-Design GmbH, Karlsruhe, 1996

[2] M. Wenk, *Topologie*-, *Gestalt- und Parameteroptimierung von Bauteilen unter Einsatz der Finite-Elemente-Methode*, Diploma Thesis, IABG, 1997

[3] MSC/NASTRAN, *Design Sensitivity and Optimization User´s Guide*, MacNeal-Schwendler Corporation, 1992