SHAPE OPTIMIZATION FOR LIFE CYCLE INCREASE AND WEIGHT REDUCTION OF ENGINE COMPONENTS USING TOSCA STRUCTURE AND ANSA

[1]Georgi Chakmakov*, [2]Matthias Friedrich
[1]FE-DESIGN Bulgaria OOD, Bulgaria, [2]FE-DESIGN GmbH, Germany

KEYWORDS –
shape optimization, fatigue, plastic strain, mass reduction, exhaust manifold

ABSTRACT –
Attempting to provide more efficient vehicles and because of emission laws, lots of car manufacturers develop engine downsizing. As a result, thermo-mechanical failure of exhaust manifolds and turbocharger housings is getting more and more critical. A good optimization strategy is necessary to significantly increase the number of cycles until cracks occur while simultaneously, the possibility of mass reductions is also pursued. For high temperature applications like gasoline and diesel engines, one main effect that influences the fatigue behavior is the plastic strain amplitude in the material. This effect can be taken into account in a low cycle fatigue analysis and thus be integrated directly into a shape optimization loop with the optimization software TOSCA Structure. Setting up such complex optimization task requires lots of considerations during its pre-processing. The fully integration of the TOSCA Structure optimization task in ANSA allows fast and reliable pre-processing of the optimization setup and has thereby established as standard pre-processing tool for TOSCA optimization tasks. This presentation shows a typical optimization procedure for thermo-mechanical applications on engine components using the performance of ANSA as pre-processor and TOSCA Structure. As a result, the possibility of life cycle increase and mass reduction of an exhaust manifold by means of non-parametric shape optimization is shown.

TECHNICAL PAPER -

1. INTRODUCTION

The desire to increase the power and efficiency of combustion engines has resulted in high thermal and mechanical loads. Exhaust manifolds and turbocharger housings are subjected to heating and cooling cycles, which can induce thermo-mechanical fatigue (TMF). Component performance and durability are critical. Computer-aided design (CAD) and computer-aided engineering (CAE) systems, in conjunction with advanced optimization tools, are needed to balance the many requirements [1].
Using an exhaust manifold component as example, an optimization process workflow is shown, based on the interaction of ANSA as pre- and post-processing tool and the optimization software TOSCA Structure. This workflow enables engineers to provide improvements in the early stages to the design team, resulting in material savings and significantly shorter product development time.

2. SIMULATION OF THERMO-MECHANICAL FATIGUE OF AN EXHAUST MANIFOLD

<u>Thermo-mechanical loading conditions</u>

Simulation of thermo-mechanical fatigue consists of a CFD (Computational Fluid Dynamics) analysis, a sequentially-coupled thermo-structural analysis, and for higher temperature applications, a durability analysis.

Initially, gas temperatures in the manifold inner passages, and the average heat transfer coefficients, are determined for the thermal analysis with an unsteady CFD simulation. The transient temperature field for the entire structure is then calculated by the thermal analysis. Mechanical loads on the manifold occur due to assembly loads (bolt pre-tension), and from stresses induced by thermal expansion. Thermal load cycles are main cause of fatigue damage. Plastic deformations occur during the first load cycles. The thermal expansions are restrained by the bolted connections.
The combination of these conditions leads to high compressive stresses. If the yield point of the material (which is lower at high temperature) is exceeded, the compressive stresses will reverse and become tensile stresses during cooling of the engine. These tensile stresses may locally exceed the material's tensile yield strength. The result is cyclically-recurring strain, a major cause of thermo-mechanical fatigue [1].

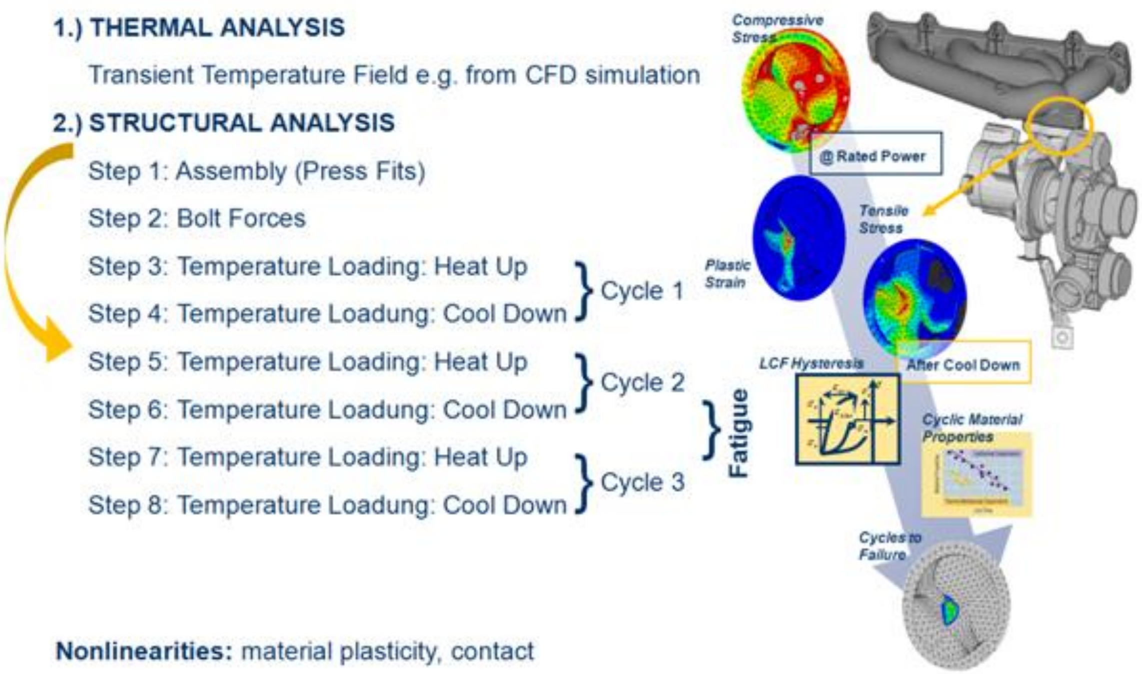

Figure 1 – Analysis sequence including fatigue [1]

For applications with higher temperatures, additional creep effects have to be included into the damage analysis. Depending on the application, different mechanisms may be relevant. At the temperatures normally measured in diesel engines, it is usually sufficient to consider only plastic strains, and their reduction, during optimization [1].

<u>FE-Analysis of an exhaust manifold</u>

The complete generation of the exhaust manifold FE-model was done in ANSA v14.1.0 based on CAD geometry input (*.igs or *.stp). ABAQUS was chosen as FE-solver for the exhaust manifold thermo-mechanical simulation. Since a sequentially-coupled thermo-structural analysis is required, two separate FE-decks have been generated.
First, a thermal analysis is performed to calculate the transient temperature field for the structure. Second, the structural analysis is performed based on the assembly loads and the calculated temperature field.
For the thermal analysis, 2 complete heat-up-cool-down cycles are simulated. Assembly loads and the calculated temperature field are then applied in the subsequent structural analysis to calculate the resulting plastic strains.
Figure 2 shows the results of the sequentially-coupled thermo-structural analysis. The post-processing is done using MetaPost v14.1.0. The figure shows the temperature field after the second calculated heat-up cycle and the resulting plastic strain distribution.

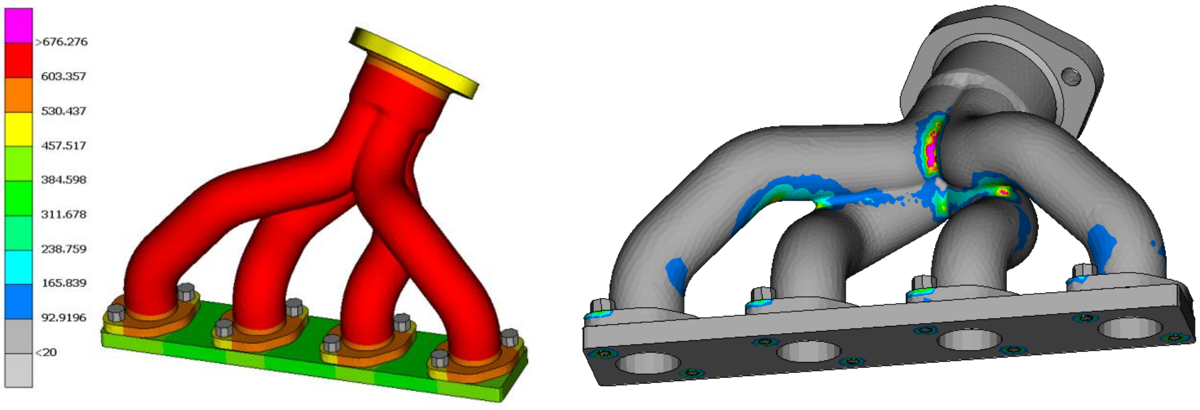

Figure 2 – Temperature field and resulting plastic strain distribution in the exhaust manifold

3. INTEGRATION OF THE TOSCA STRUCTURE TASK ANSA

The full integration of the TOSCA Structure task in the ANSA task manager significantly simplifies the definition of the optimization setup. In ANSA, templates are offered for the complete setup, execution and post-processing of a topology, shape and bead optimization with TOSCA Structure. The setup of the optimization task is function based. The user has continuous interaction with the FE-model during pre- and post-processing of the optimization task. Further, a built-in consistency check assists the user while defining the optimization setup.

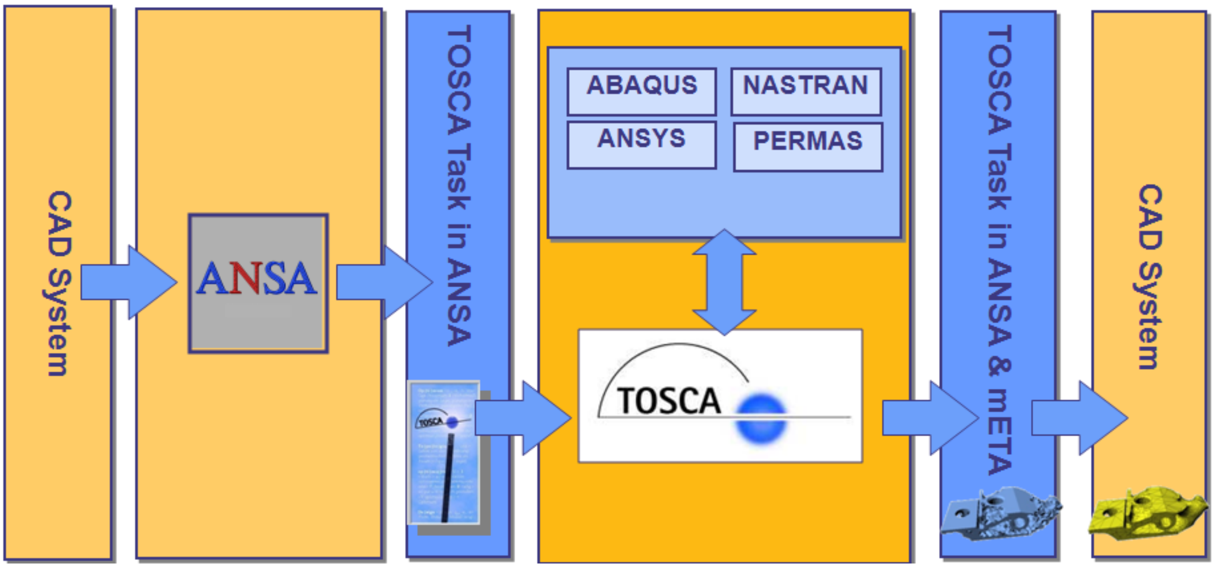

Figure 3 – Support of Non Parametric Optimization with the TOSCA Task in ANSA

2. SHAPE OPTIMIZATION

Shape optimization is a process in which the surface of a part is modified to minimize or maximize a certain objective function, for example stresses or frequencies. In contrast to a topology optimization, new geometry is not created. Rather, the original geometry is modified.

<u>Shape optimization using TOSCA Structure</u>

Non-parametric shape optimization, as employed by TOSCA Structure, is performed directly on the finite element model. The portions of the part surface which may be changed ("the design area") are defined using node groups. Selected nodes are moved normal to the part

surface, and adjacent interior nodes are also moved, to avoid distortion of the mesh (mesh smoothing). A high level of design flexibility is possible because each node can be moved independently.
The algorithm for shape optimization within TOSCA Structure is based on optimality criteria and provides very fast convergence behavior. After only a few (usually 5-10) iterations, considerable improvement in the stress distribution and fatigue life may be achieved. The resulting free-form surface is described by the modified surface mesh [1].

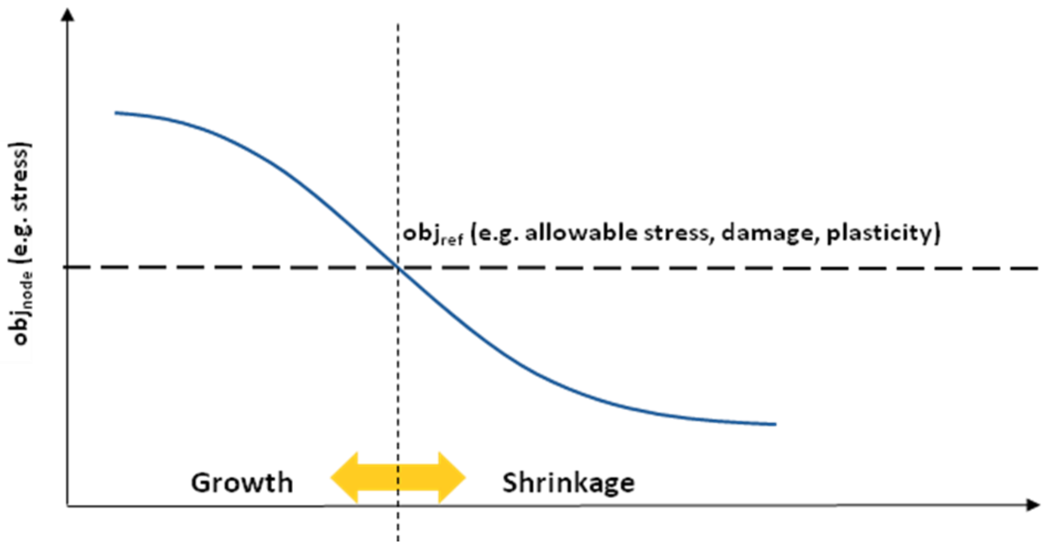

Figure 4 – Exemplary distribution of loading (objective) for an arbitrary part surface

Figure 3 shows the main principle of the controller algorithm, embedded in the TOSCA Structure shape optimizer. Generally, an area with higher objective than the reference value, will grow (material is added), while an area with lower objective than the reference value, will shrink in. With the shape optimization controller algorithm, the objective will be homogenized around a target value, thus allowing low-stressed regions to shrink and material volume of the optimized part to be reduced. At the same time, critical values around local hot-spots are improved by locally adding material.

Setup of the shape optimization using ANSA

The goal of the optimization is to reduce the weight of the manifold by 15% and at the same time to reduce the plastic strains in the critical regions as far as possible.
There are 5 critical areas with high plastic strains identified after the structural analysis. Since a weight reduction is pursued, a more global optimization strategy is chosen where the design area is most of the outer surface of the manifold. However, the identified critical hot-spots are also brought out in the optimization setup and the plastic strain values in those regions will be monitored during the optimization. The definition of the target nodal groups around the critical hot-spots is easily done in ANSA after the node IDs with the highest plastic strain from each region are determined in MetaPost.
The 15% target weight reduction is considered as a volume constraint. Further, nodal fixations are applied to all functionality surfaces to exclude them from the optimization. To keep a certain minimum required material thickness, a minimum member-size restriction is also specified.
The shape optimization is executed under consideration of a temperature field update in each design cycle. Hence, the heat transfer and the structural analysis will be sequentially executed in every single optimization iteration.

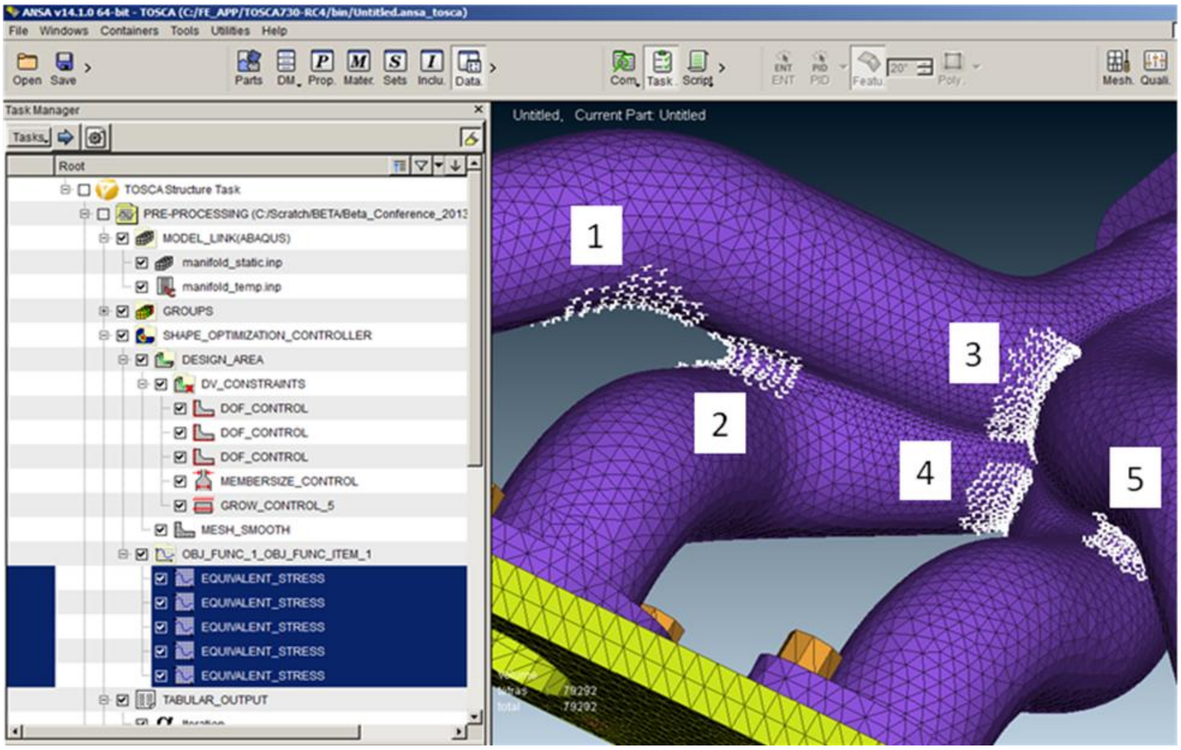

Figure 5 –TOSCA Structure shape optimization task in ANSA with highlighted hot-spots

<u>Shape optimization run</u>

During the first 3 optimization iterations, the outer surface shrinks globally to achieve the target volume constraint. At the same time, also slight local improvement of the plastic strain distribution is performed. After the volume constraint is reached, the plastic strain homogenization starts and the regions around the hot-spots begin to grow.
The initial design investigation has shown that the nodal growth after iteration 3 has positive influence on all hot-spots except hot-spot 5 where the nodal growth causes a fast increase of the plastic strain. Thus, the plastic strain improvement of 30%, gained during the first 3 iterations, is reduced to its original plastic strain levels after 8 iterations. To consider plastic strain reduction on all hot-spots, an advanced optimization strategy is adopted which uses the positive influence on hot-spot 5 from the global shrinkage in the first 3 iterations.

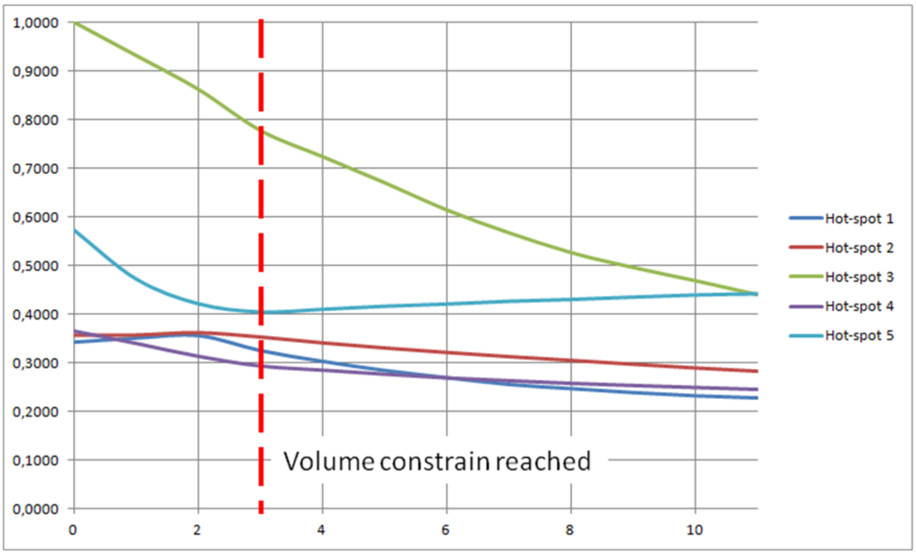

Figure 6 – Normalized plastic strain plots in the 5 critical regions over the iterations for a new combined optimization strategy

Figure 6 displays the result of a new optimization strategy, consisting of combination of two sequential shape optimizations. The initial optimization is stopped after iteration 3 at the time the volume constraint is reached. These 3 iterations bring already a 30% plastic strain decrease at hot-spot 5. Therefore, at this point, the nodal growth in the hot-spot 5 region is prohibited, because of the great negative influence, and the optimization is restarted with this setup change. It can be determined from figure 6 that this fixation still has a negative influence on the plastic strain reduction at hot-spot 5. However, the negative effect after the fixation is much lower and at the end of the optimization, there is a 23% plastic strain reduction at hot-spot 5. The biggest plastic strain reduction, achieved by the shape optimization, is at hot-spot 3, where the overall maximum plastic strain is located in the original model. The plastic strain reduction at hot-spot 3, which represents also the overall maximum plastic strain reduction, accounts to 56%.

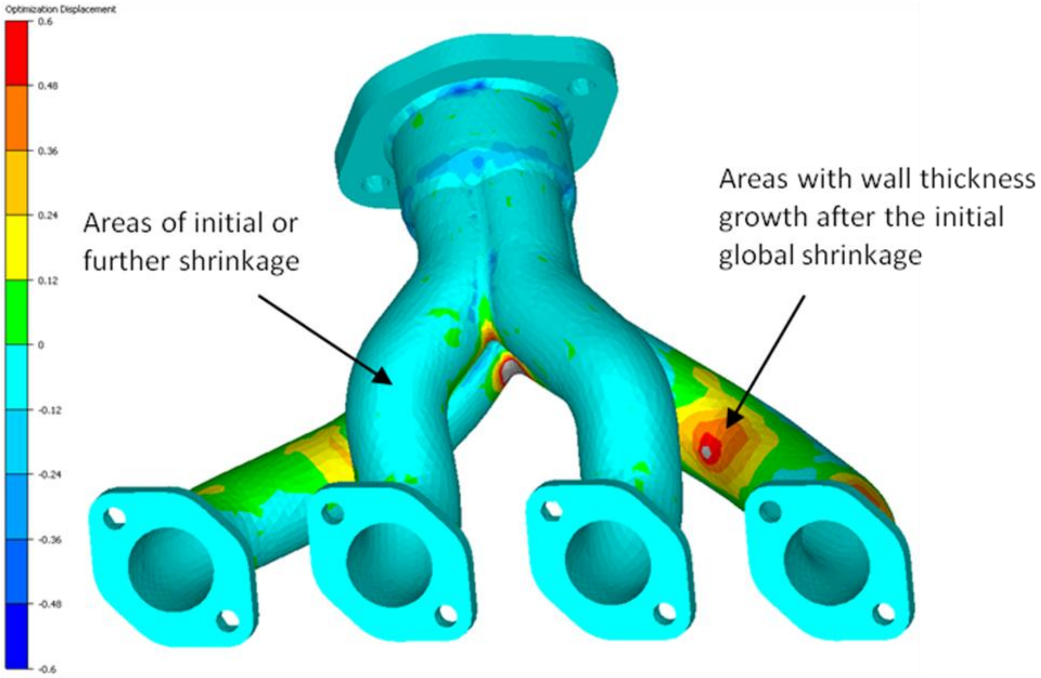

Figure 7 – Wall thickness change after the initial global shrinkage (iteration 4 to 11)

On figure 7, the change in the wall thickness is shown for the iterations after the volume constraint has been reached. Figure 8 shows the change in the contour geometry at the critical hot-spot 3 before and after the optimization with the corresponding level of plastic strain.

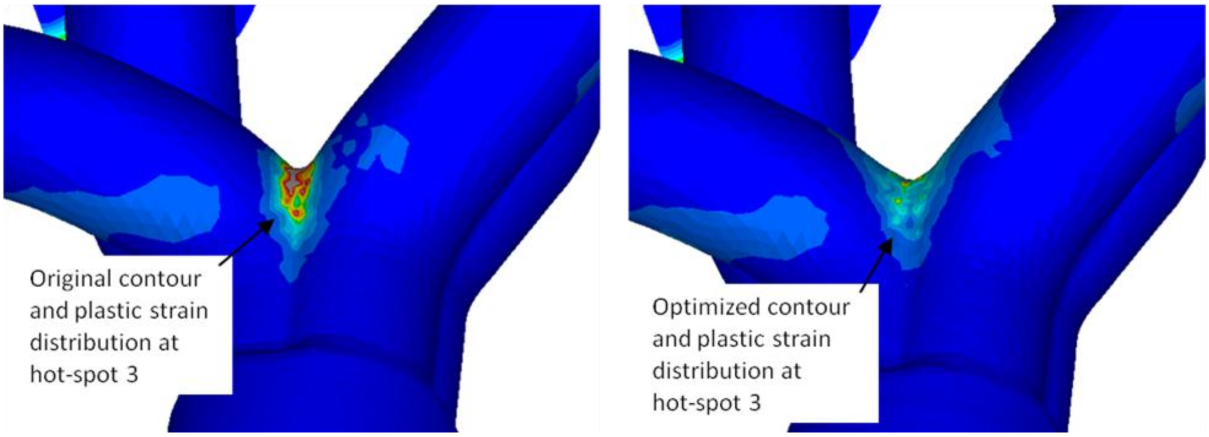

Figure 8 – Optimization of the geometry at maximum plastic strain location

3. OPTIMIZATION RESULTS TRANSFER

As stated before, the non-parametric shape optimization modifies the outer geometry of the optimized part. The modifications are performed nodal based. Thus an optimum free-form contour and a non-uniform wall thickness is the optimized result.
To support the design team in the process of interpretation and remodelling the optimized geometry, a simplified geometry model of the optimized state is needed as output. TOSCA Structure offers the possibility to output the optimized surface in CAD-readable formats such as *.stl and *.igs. However, the generated surface data is based on the surface elements of the FE-model, where each facet represents a separate face. This often leads to a significantly increased size of the output data.
ANSA offers an alternative way to generate a simplified CAD-readable output of the shape optimization results. Using the ANSA morphing function DEFORM MAP, the original CAD geometry can be mapped onto the FE-mesh of the optimized state. Here, as an input data for the transformation, the shape vectors can be used. The vector information can be generated in ASCII format from the surface nodal coordinates and the nodal optimization displacements, which are provided as output in TOSCA.

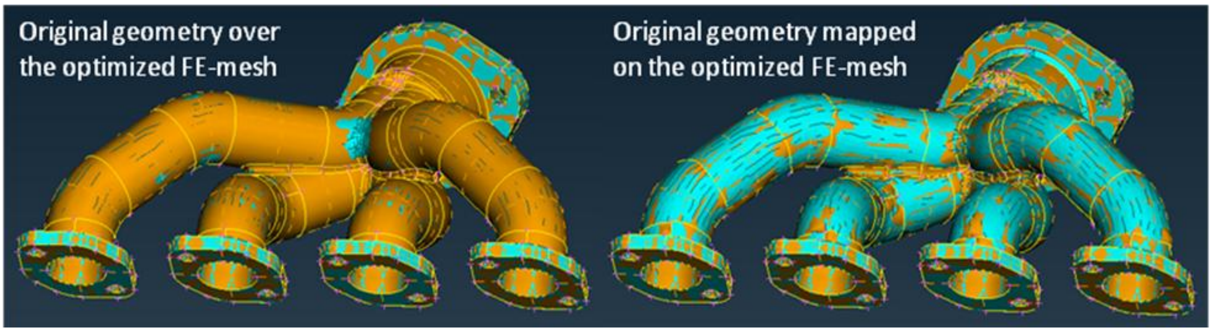

Figure 9 – Generating optimized geometry by mapping the original geometry to the FE-mesh of the optimized model

4. CONCLUSIONS

Non-parametric shape optimization proves to be an efficient strategy to optimize thermo-mechanically loaded engine components such as exhaust manifolds. Using the controller based shape optimization strategy with TOSCA Structure, a successful optimization was performed on an exhaust manifold by achieving a weight reduction of 15% and simultaneously reducing its maximum plastic strain by 56%. This leads to a significant improve in the durability performance and increasing the life cycle of the component.
Alongside with the optimization software TOSCA Structure, ANSA and MetaPost are very important part of the whole optimization process, starting with the FE-modelling of the exhaust manifold and evaluation of the simulation results to identify the critical areas for shape optimization. Further, the fully integrated TOSCA task in the ANSA task manager allows the user to easily create a complex setup for optimization. Finally, using the powerful morphing capabilities of ANSA, a geometry output of the optimization result can be created by mapping the original geometry onto the FE-mesh of the optimized model. This way, the optimization result transfer process becomes simplified and more efficient.

REFERENCES

(1) Friedrich, M., Biel J., Bangert C., Optimierung von thermomechanisch beanspruchten Bauteilen, ATZextra, May 2012