Optimization of a fan shroud by ANSYS/DesignModeler and optiSLang

Markus Egerland , Dirk Roos& Johannes Will

SiemensVDO automotive AG

Electric Motor Drives

Dynardo GmbH

Summary

Object of this paper was a pilot project to test the ability of the software combination optiSLang/ANSYS to handle a complex CAD geometry with a significant number of parameters for parametric optimization. An automotive fan shroud was chosen as an example, target was the reduction of mass with the restrictions of stress, plastic strain and normal deformation (z-direction).To get access to CAD-parameter within a parametric finite element evaluation ANSYS Workbench was used.

Since the attempt to use the Interface between Ansys and the CAD tool ProEngineer failed due to the high amount of reference dependent parameters and the used skeleton construction, the decision was made to generate a FEM applicable CAD Model with the ANSYS DesignModeler. Using that approach an automatic parametric process between the optimizer optiSLang, finite element simulation in workbench using a geometry update in Design modeler was successfully installed.

A further intention was to show the automatic generation of a FE mesh which is significant for a parametric optimization procedure using CAD parameter. The optimization starts with a sensitivity analysis with 93 input and 6 output parameters to find the most significant parameters for an ARSM (adaptive response surface method) optimization, this "pre-optimization" with 9 parameters provides a design which was used for an EA (evolutionary algorithm) optimization in the complete defined parameter range.

To complete the workflow, a robustness analysis around the best design was started and after two modifications a robust design with a mass reduction of ca. 15% was found.

Keywords

Mass reduction, CAD Model, sensitivity, parametric optimization, adaptive response surface method, evolutionary algorithm, robustness, mesh control

1. Target of the Project

During virtual engineering processes automatically optimization becomes more and more an object of interest.

Among topological and topographical optimization parametric optimization can be an instrument to improve the properties of products.

Optimization tools are available within the classic Ansys and in the Workbench environment (DesignXplorer) as well. These tools are recommended for FEA models with a small amount of continuously varying parameters (5...10 Parameter).

With rising computer power the handling of considerably more parameters seems to be possible and eligible.

Dealing with a high number of parameters in CAE-based optimization is one of the targets optiSLang was developed for.

To test the ability of the combination optiSLang/ANSYS to handle a significant number of parameters a fan shroud was chosen like in the fan module shown in Fig.1.

The shroud has to meet several requirements (acoustical, fluid mechanical, stability), but for this project only the reduction of mass under the restrictions of equivalent stress, plastic strain and deformation in axial direction under a statically load was chosen as purpose.

Fig.1: Fan module: fan, shroud and drive

2.   Generation of FEA Model

Base of the model was a CAD geometry in ProE. Since the ANSYS ProE interface exists, an attempt to use the original ProE model with accurate simplifications was obvious.

Within the original model more than 2000 Parameters may be found, some of them in used references or the skeleton construction. At this entire situation geometric regeneration failures occurred. The CAD geometry was not dedicated for FEA with automatic parametric optimization.

The decision was made to generate a new model with the ANSYS DesignModeler.

2.1 Geometry

With the DesignModeler a simplified CAD model was generated and parameterized with the most significant parameters. To avoid regeneration failures mathematical relations were created between some parameters. This is possible with the DesignModeler parameter manager.

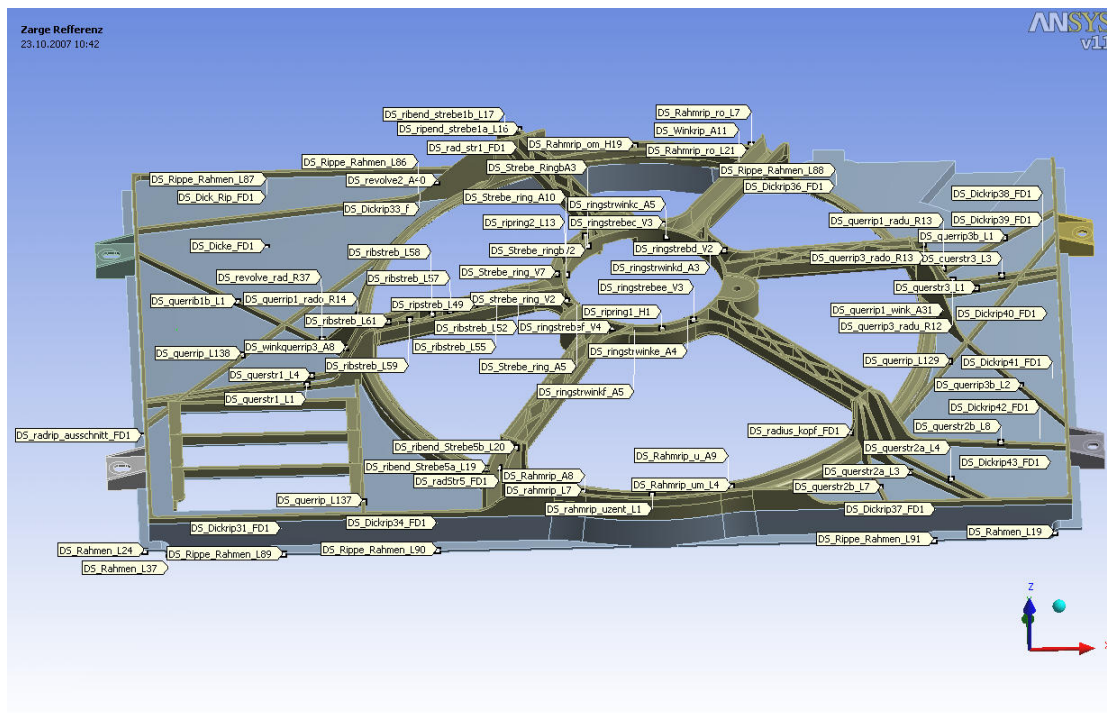The model was divided in different bodies and merged to a single part.

Fig.2: DesignModeler Geometry with parameters

2.2 Meshing

The meshing strategy for the optimization is not exactly the same as for a single FEA . For a single FEA two kinds of meshes would be possible: A model consisting of mid-surfaces with shell elements or a volumetric model with a tetraeder mesh. As an alternative ANSYS 11.0 provides a combination of a tetraeder and a solid-shell (190) mesh. This alternative was not used since Ansys 10.0 was in use when the project starts, it should be tested later.

A mid-surface model seems to be not stable enough for some parameter combinations and not accurate for all sorts of wall thickness. Therefore the decision was reached to use tetraeder elements. The mesh for the optimization task has to meet four different requirements:

1. Automated meshing must be possible for changing edges, angles and surfaces.
2. The mesh quality must be comparable for every parameter combination
3. Accurate results for the changing geometry, in view of the thin structures.
4. In light of the expected number of calculations the calculation time should not be to long .

For a single FEA it would be possible to locate problematic regions and parts with high stress gradients and to refine the mesh at these specific regions.

But concerning the second requirement this is not possible for the optimization model because critical regions as well as stress maxima can change the location due to parameter variation.

Further ANSYS provides the use of virtual surfaces for areas made of multiple faces, which can be critical in this example at the inner motor mounting ring.

But for the optimization this does not work since the amount of faces is not the same for every combination.

At the end the mesh was generated as follows:

A sphere (Fig.3) was defined at the motor mounting ring with an element size of 1,72mm which was validated for the sharpest existent angles.

A general element size for the rest of the model was found by calculating the stress for different element sizes.

The relevance for the model was set to the highest level (100).

This mesh (Fig.4) with about 190000 elements is an accurate mesh for the reference model, but it has to be also checked for a lower wall thickness.
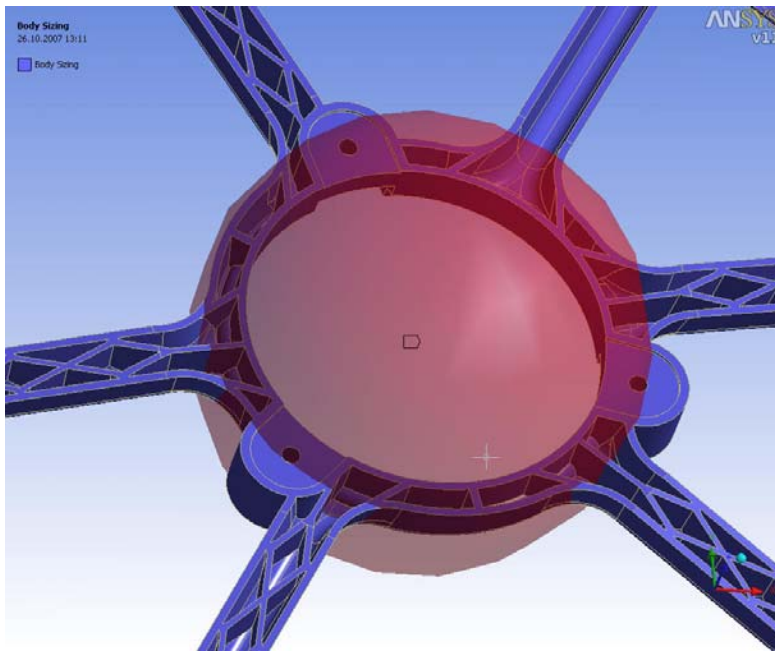

Fig. 3
Sphere of influence for motor mounting ring.
Sharp angles occur between rips.

At this point a first sensitivity analysis with optiSLang was started to get more information about the model.

Are there any regeneration problems?
Can the ANSYS Workbench mesher always find a mesh?
Where are the stress maxima located?

After ca. 100 calculations the analysis was stopped and the results were evaluated only regarding this questions. To solve the occurring events moderate changing in the geometry and in the variation range was made.
The best design in respect of mass out of the solutions, which had a significant lower wall thickness as the reference model was used to validate and refine the mesh as follows:

- Comparing nodal solution with element solution to get an idea of result quality.
- Executing an adaptive mesh solution to see "how ANSYS Workbench would refine".
- Checking again with the adaptive mesh comparing nodal and element solution to get a reference result for equivalent stress.

Since the adaptive mesh provides only a locally refined mesh it is not usable for the optimization process.
But according to the reference result received from the previous step now the mesh can be refined again by approximating the mesh size.
The here found mesh (Fig.5) had about 350000 elements and was used for the next optimization cycle.
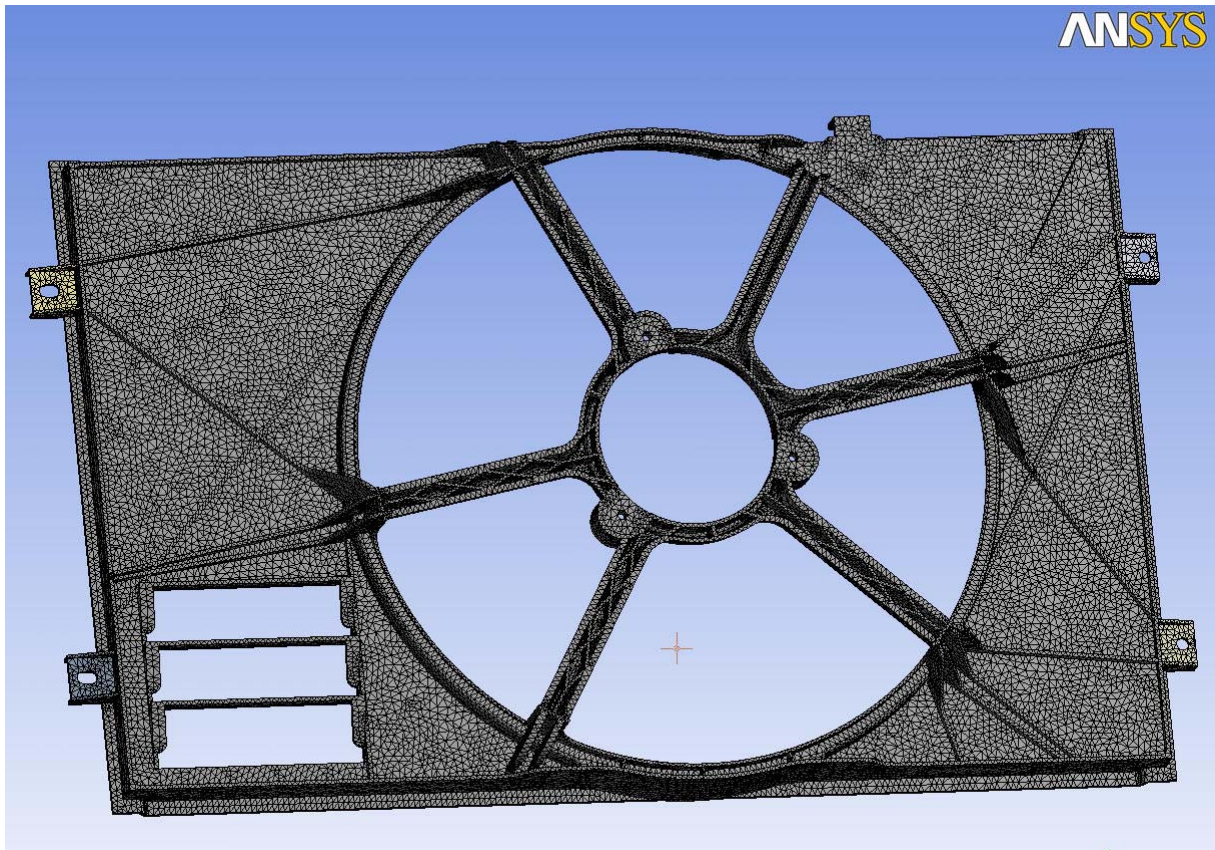
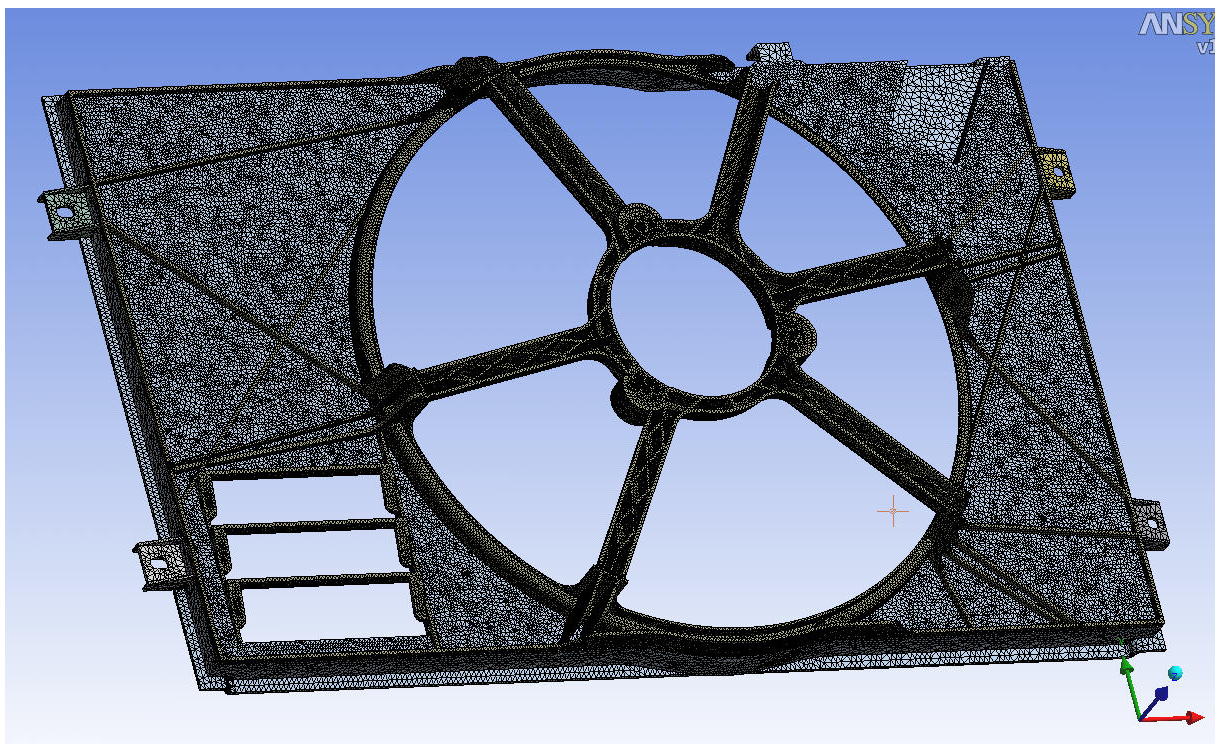Fig.4: First mesh for reference model with 190000 elements

Fig.5: Final mesh with 350000 elements

2.3 Material and boundary conditions

Since the maximum plastic strain is a restriction for the optimization a multilinear isotropic material model with plastic option was used, which is the default within Workbench after version 9.0.
The Model is loaded with a remote force of 200N on the motor mounting ring.
Remote displacements are used to model the supports at the bolt-holes.

3. Optimization workflow

After generating an accurate FEA model the task is now to define a strategy for the optimization workflow.
Target of the optimization was to reach a mass reduction of the fan shroud by more than 10%.
The restrictions are:
1. Plastic strain less than 0,1%
2. Maximum equivalent stress less than 25 MPa
3. Directional deformation less than 6mm.
The output parameters "maximum stress2" and "deformation_max" are used for checking the model behavior .
In this example the optimization workflow was as follows:
Sensitivity Analysis
- To examine the parameter range and identify the most sensitive parameters.

Pre-optimization with adaptive response surface method
- With a number of less than 15 parameters the ARSM is an effective method.

Final Optimization with evolutionary algorithm
- To examine the whole parameter range to use further optimization potential.

Robustness analysis
- To check the best design in view of defined restrictions.

Corrective action and further robustness analyses.
- To get finally a robust design.
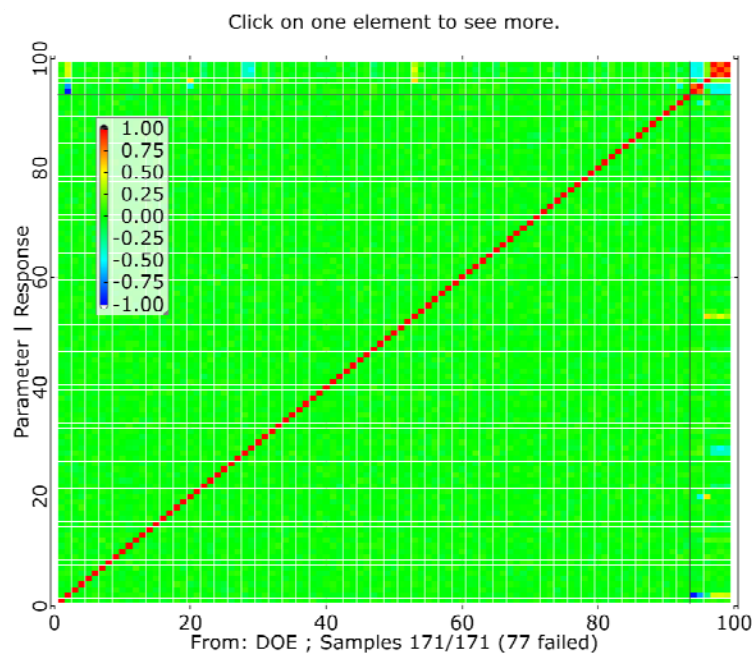
4. Results

4.1 Sensitivity

Fig.6: Parameter range and response after   Sensitivity analysis.

To investigate the parameter range of initial 92 input and 6 output parameters a sensitivity analysis was performed.

If it is possible to reduce the amount of parameters to less than 15, an adaptive response surface optimization within the reduced parameter range is the consequent next action.

Viewing the results of the sensitivity analysis the amount of 77 failed designs seems to be high.

It is advisable to check the failed designs:
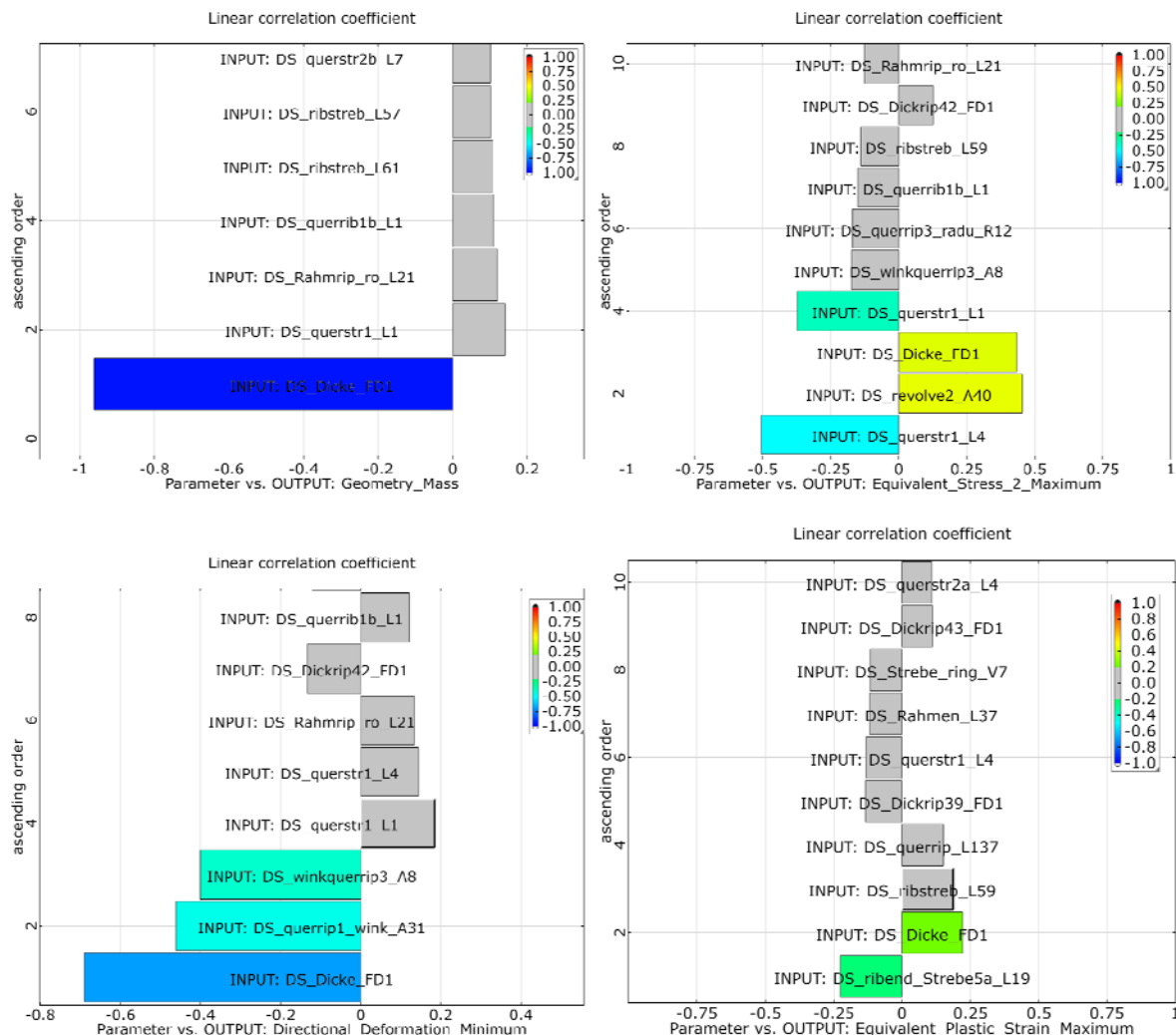
Are there meshing problems?

Are there geometric regeneration problems?

Fig.7a-d: a) Linear correlation coefficient according to mass
b) Linear correlation coefficient according to equivalent stress
c) Linear correlation coefficient according to Deformation
d) Linear correlation coefficient according to plastic strain

If corrective action is necessary the user has to decide, if the sensitivity analysis has to be repeated or if the workflow can be continued with the next step. A repetition is only reasonable, if the correction could affect the results.

In this task 8 sensitive parameters were identified (Fig.7a-d).

Further the result check showed a parameter, which was not variable due to a dependency to another parameter. This dependency was eliminated and a new parameter was generated.

Since it was not reasonable to repeat the sensitivity analysis because of this new parameter, the decision was made to use the parameter in addition to the previously found 8 parameters for the ARSM.

4.2 Adaptive Response Surface Optimization

With the 9 optimization parameters found as described before a pre optimization using the adaptive response surface method was performed.

After 49 calculations a best design with a mass of 930 g was found. The most critical response was the plastic strain which reached the limiting value of 0,1%. Furthermore some optimization parameters met their boundary.
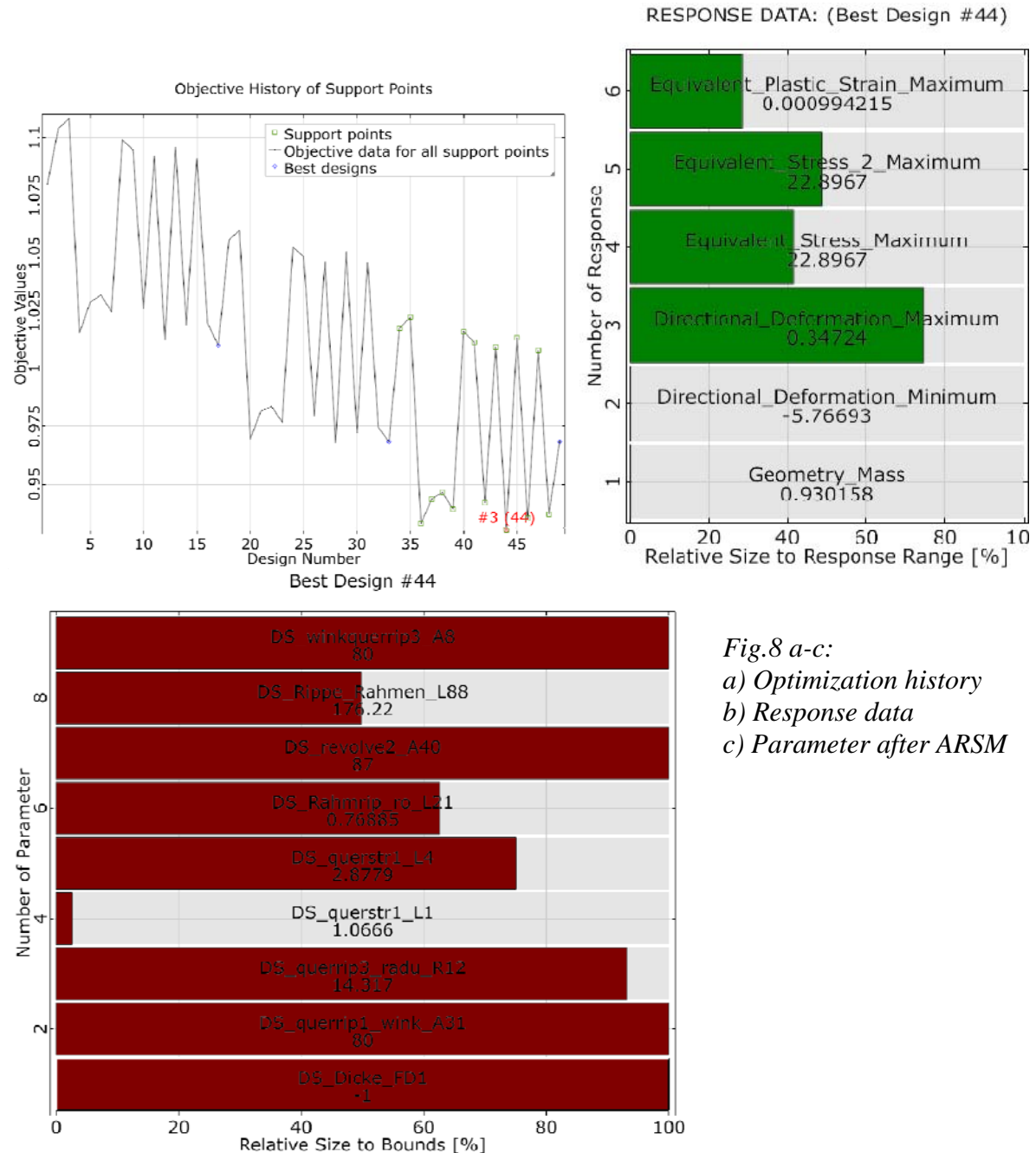
With a mass reduction of 13,7% the target of 10% was already beaten after the ARSM.

Fig.8 a-c:
a) Optimization history
b) Response data
c) Parameter after ARSM

Although the target was reached the optimization was continued with an evolutionary algorithm in the whole parameter range of 93 parameters to investigate the potential of optimization in the full design space. To allow further improvements in addition the boundaries of the optimization parameters which met their limits were expanded slightly.

4.3 Evolutionary algorithm

Using the best design of the pre optimization as starting value an optiSLang design improvement using Evolutionary algorithms was started.
For this design#44 of the ARSM was chosen as the start design.
By default the optiSLang design improvement uses 1 start design, 5 new (mutated) designs per generation, an archive strategy as well as adaptive mutation procedure.
Using parameter free constraint handling ensured the usage of the best design without violation of a constraint to be the parent of the next generation.
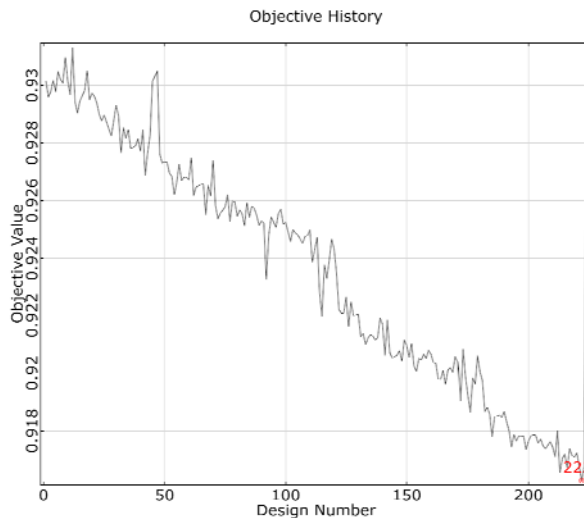
Fig.9: Objective history of evolutionary optimization   Fig.10: Response data evolutionary optimization
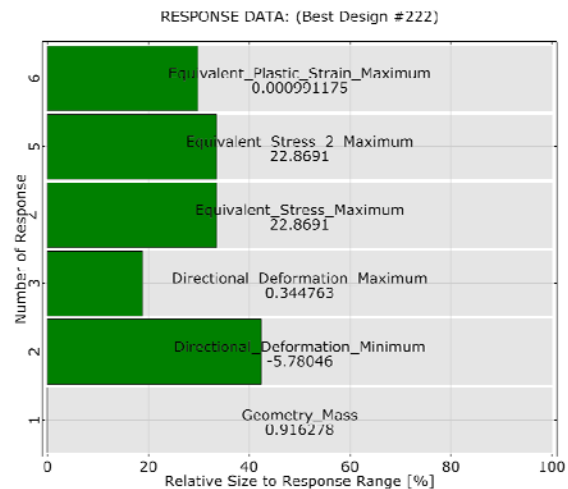
After 224 calculations the mass of the shroud was reduced to 916,3g, an improvement of 15,7% compared to the reference design and 1,5% compared to the best ARSM design.
Viewing Fig.9 a further improvement of the design seems to be possible, but the optimization process by using an evolutionary algorithm is very slow compared to the ARSM.

4.4 Robustness 1

Last step of the optimization workflow is the robustness analysis. For this not the exactly best design was used, but a modified version with CAD applicable parameter data. Here just the parameter values of the optimal design are set to values which could be produced.
This modified model was checked by FEA if it meets the defined constraints (Fig. 11).
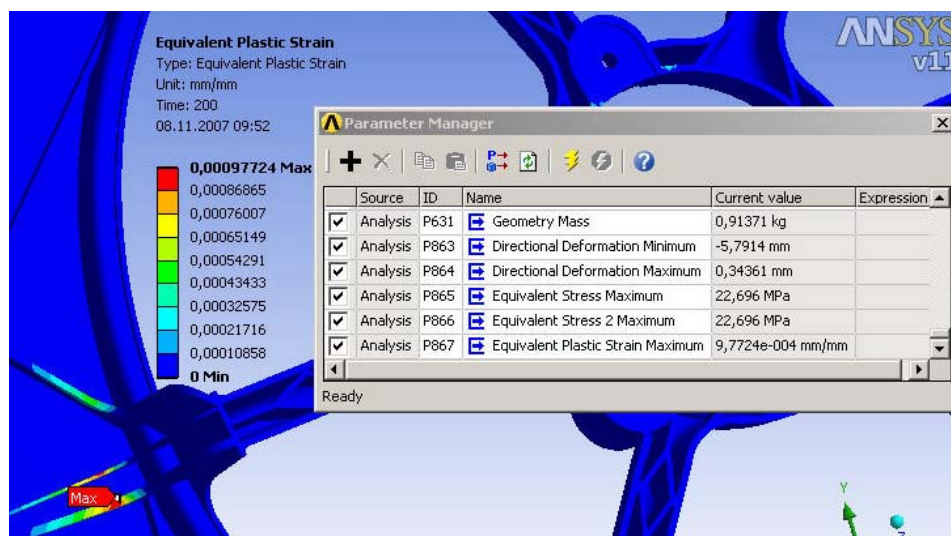

Fig.11
Base design for robustness.

At last a model with a mass of 913,7g was found, which had even less plastic strain as the former found optimum.

For the robustness analysis tolerances for injection molding parts, DIN 16 901 were used. The parameter range was reduced to 65 parameters and Gaussian distribution with a coefficient of variation 0,01 was adjusted. Using a Latin Hypercube Sampling to generate a certain number of possible design realizations the scatter of important response criteria was calculated.
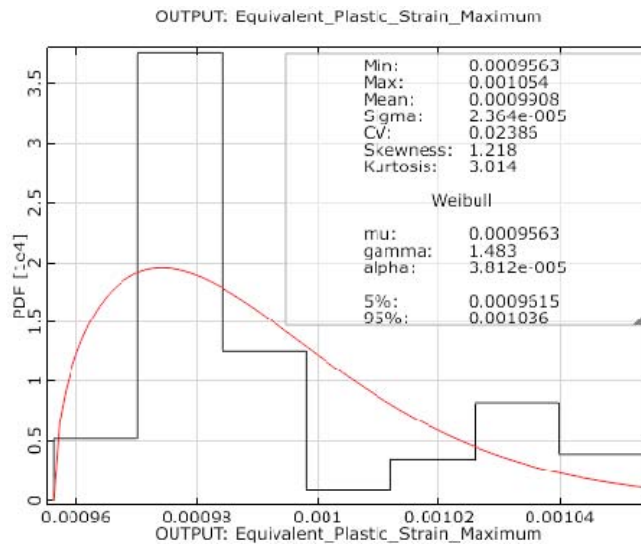

Fig.12
Distribution of plastic strain after 166 calculated designs.

After 166 calculated designs the criterion of plastic strain less than 0,1% was hurt in 35 designs (Fig.12).

The anthill plot (Fig.13) of the parameter with the highest linear correlation coefficient shows two different clusters. This explains the low correlation of the parameters (Fig.14).
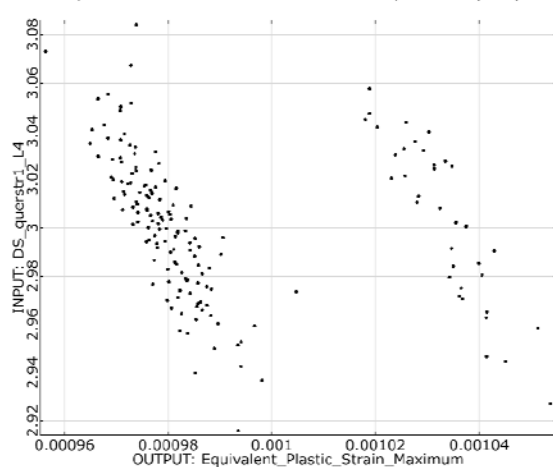

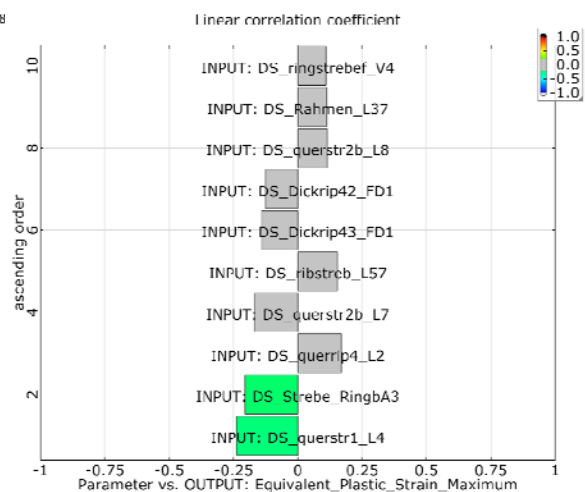Fig. 13: Inp. rib thickness vs. outp. plas. strain   Fig. 14: Coefficients of correlation

The cluster on the left contains the designs which are below the criterion, on the right side the plastic strain is higher than 0,1% without exception. To examine this phenomenon optiSLang provides the possibility to eliminate some designs and consider only the failed designs (Fig.15).

Now the coefficients of linear correlation are higher and in addition to the so far sensitive parameter "DS_querstr1_L4" other parameters show an influence (Fig.16).

Therefore two different actions seem to be suited to decrease violation of plastic strain:

1. Increase thickness of rib "DS_querstr1_L4"- because of the already thick structure less practicable.

2. Increase thickness of rib "DS_Rahmrip_om_H19" or thickness "DS_querstr1_L1"

At this point the user has to interpret the significance of the correlation coefficients: After removing the cluster1 only 35 designs are used for the calculation of linear correlation coefficients.

The influence of parameter "DS_querstr1_L1" is much more plausible as "DS_Rahmrip_om_H19", this was also confirmed by the sensitivity analysis.

Corrective action: The parameter "DS_querstr1_L1" was increased to 1,5mm, further the parameter "DS_Dicke_FD1" which is most significant for mass and deformation was set to 1mm.
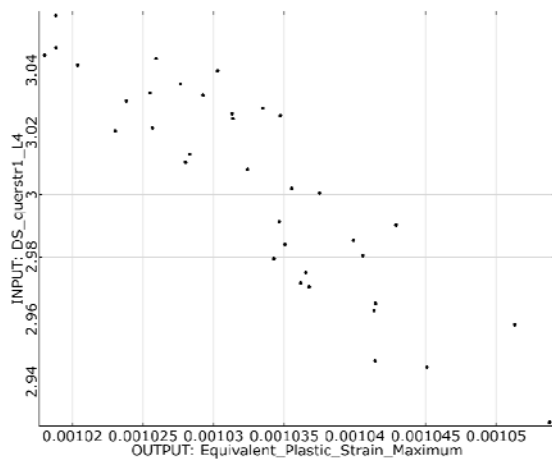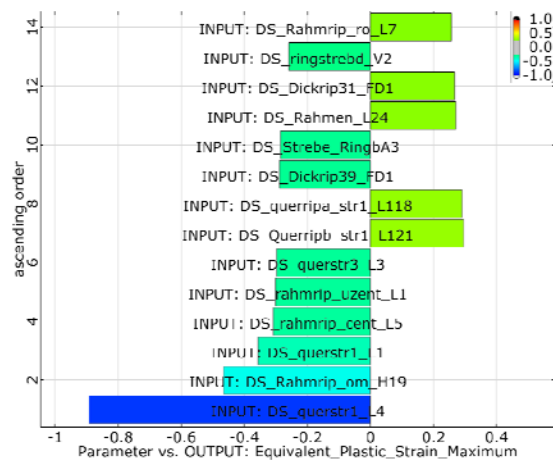

Fig. 15: Inp. rib thickness vs. outp. plas. strain

Fig. 16: Coefficients of correlation

4.5 Robustness 2

With the described changing a second robustness analysis was made. After 223 calculated designs the robustness evaluation was stopped and there was not any design which exceeds the criterion of 0,1% plastic strain.

The mean of the mass was now 923g (14,4%), with a coefficient of variation 0,0019.

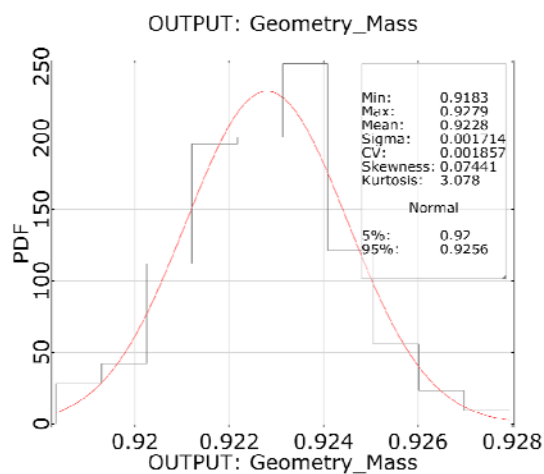Fig.19 shows, the cluster effect now is significant lower as before.
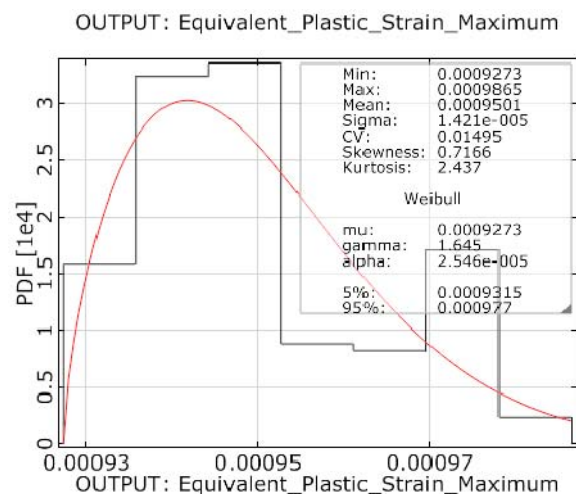

Fig. 17: Distribution of mass

Fig. 18: Distribution of plastic strain

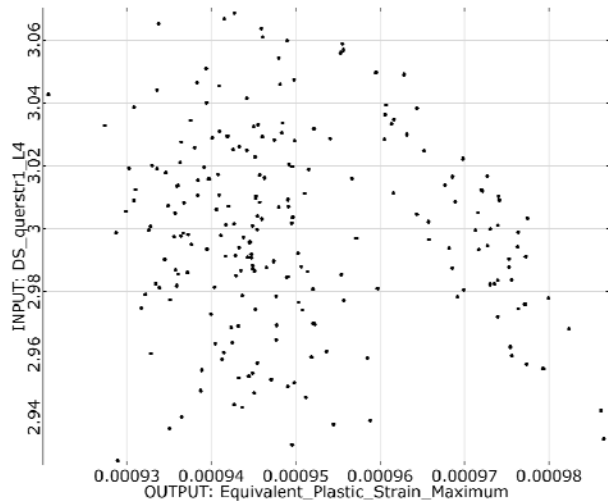OUTPUT: Equivalent_Plastic_Strain_Maximum vs. INPUT: DS_querstr1_L4, (linear) r = -0.130

Fig. 19: Distribution of plastic strain

5. Conclusions

- With the software combination optiSLang/ANSYS Workbench a mass reduction of 14,4% was reached, subject to the verification of producibility.
- By limitation of time and effort a mass reduction of 13,7% is possible, if the amount of parameters can be restricted to less than 15.
- The FE mesh has to be checked for geometrical differences and corrected where required.
- The mesh-tools of ANSYS Workbench cannot be used in the same manner as for a single FEA.
- The use of CAD Models depends on the complexity of the geometry. At least a close communication with the CAD designer is necessary.

6. References

[1]   Release 10.0 Documentation for ANSYS Workbench
[2]   Release 11.0 Documentation for ANSYS Workbench
[3]   optiSLang documentation version 2.1