Deutscher Luft- und Raumfahrtkongress 2011
DocumentID: 241385

SHAPE OPTIMIZATION OF COMPONENTS FOR THE ARIANE 5ME UPPER STAGE

A. Reim, TEB11, Astrium GmbH, Airbusallee 1, 28199 Bremen, Germany
G. Quappen, TEB11, Astrium GmbH, Airbusallee 1, 28199 Bremen, Germany

Abstract

This paper describes the optimization method, which is developed to optimize main structural components of cryogenic upper stages. It utilizes free shape as well as nonlinear shape optimization techniques. Contrary to the optimization of upper stages using storable fuel, the above mentioned structures are subjected not only to high forces and pressures but also to high temperature gradients. Additionally, the production of the structure implies geometrical constraints in the shape of the structure. The goal of this work is therefore to combine these requirements into a comprehensive and as far as possible automated optimization method for use in preliminary structural optimization phase.

1. INTRODUCTION

The reduction of structural weight is a main factor for improving the overall performance of a launcher system. Therefore, innovative optimization methods have to be utilized to enable a lightweight design, which simultaneously fulfills all requirements in the structure. Within the scope of this work, different optimization methods are utilized to optimize the upper and lower Y-ring of the Ariane 5ME bare tank regarding the reduction of bare tank mass. This task is performed considering all essential boundary conditions and constraints. The starting point is a modified bare tank design, which is derived from the ESC-A upper stage. The FE model consists of a 5° segment model and contains different load cases as well as a specific temperature distribution (hot case).

The structural forces, which act on the structure, can be grouped into pressure loads and general loads. The pressure loads consider the internal gas pressure p_{gas}, the corresponding hydrostatic pressure p_{HS} and quasi static hydrodynamic pressure p_{HD} of the LOX and LH2. The hydrostatic pressure is driven by the axial acceleration, whereas the quasi static hydrodynamic pressure is calculated by analyzing the dynamic response of the tank structure and their interaction with the propellant masses. The over all pressure is given by equation (1) and (2).

$$p_{LH2} = p_{gas,\,LH2} + p_{HS,\,LH2} + p_{HD,\,LH2} \tag{1}$$

$$p_{LOX} = p_{gas,\,LOX} + p_{HS,\,LOX} + p_{HD,\,LOX} \tag{2}$$

The general loads comprising inertia and aerodynamic loads are derived from stage specifications to represent the loads that act on the tank over its interfaces. As shown in figure 1, loads are introduced from the two payloads into the upper stage (F_{PL1} and F_{PL2}). Additionally, the engine loads are implemented by consideration of F_{ENG} for the inertia loads during main stage flight and thrust loads during the upper stage flight.

The thermal mapping for the different components is given and will not be modified within this work, cp /1/.

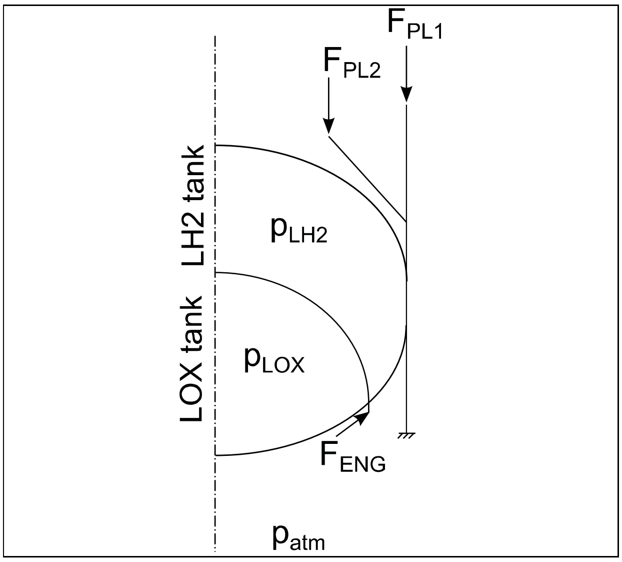

FIG 1. Cross section of A5ME upper stage (schematic).

2. DEVELOPMENT METHODS

Within the development of upper stage structures, different methods for calculating the stresses can be used , cp /2/:

- Handbook calculation (Level 1)
- Simplified analytic methods (Level 2)
- Finite Element analyses (Level 3a/b)

These methods for stress calculation can be combined with different optimization strategies in order to optimize the design regarding an objective, e.g. the structural mass. Common strategies are, cp. /3/:

- Topology optimization
- Shape optimization
- Sizing

Since the depth of the considered details within the methods for stress calculation varies over the different levels, only level 3 methods can be used for all of these three optimization strategies. Furthermore, also the status of the project influences the freedom to perform design changes. In early design stages, a topology optimization can generate information about main subcomponents and their location within the planned structure, cp. figure 2.

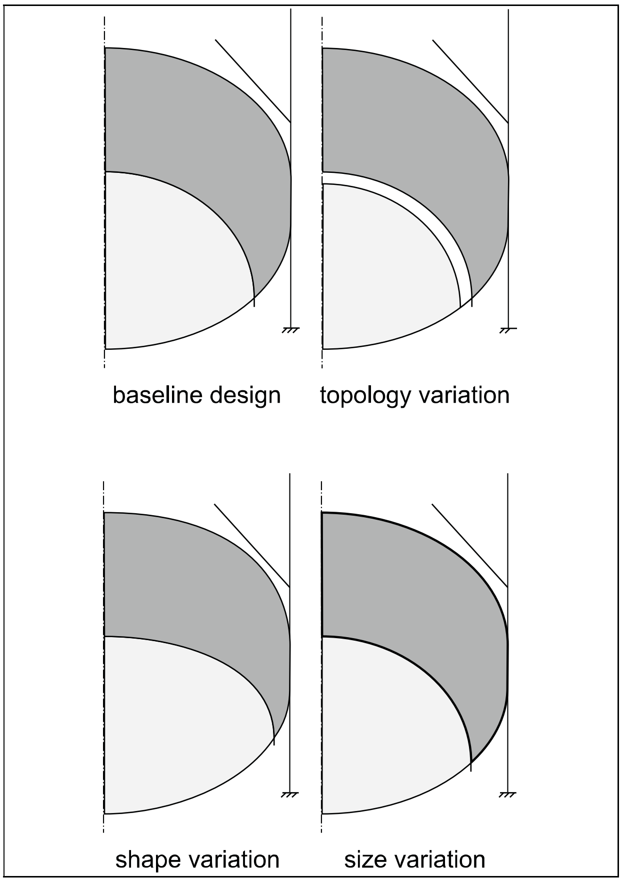

FIG 2. Optimization methods.

If the topology of the structures is already fixed, a shape optimization can improve the performance of the structure without changing the topology. Therefore, only the location and the dimension of the subcomponents are modified to meet all requirements and reduce structural weight. The latter optimization method will be used to reduce the structural weight of the bare tank, since the topology is already fixed.

2.1. Optimization problem

In order to perform an optimization of the structure, the optimization problem has to be defined. It contains the aspects *system parameters*, the *optimization objective*, *optimization constraints* as well as the *system function* /4/.

System parameters are given by the geometric definition of the structure. They can be thickness information, radius values or any other geometrical parameter.

The optimization objective of cause is mass reduction. Thus, the mass has to be minimized by simultaneously keeping the design in the constraints limits. Different constraints are integrated into the analyses:

– Maximum von Mises stress

The maximum von Mises stress depends on the material as well as on the temperature at the analyzed location and the loading condition (yield vs. ultimate stress). For the aluminum alloys under investigation, the maximum allowable von Mises stress typically rises for lower temperatures.

– Maximum Shear stress

The maximum shear criterion is used for analyzing the weld seams, which connect the structural parts. Like for the von Mises stress, it also depends on the material, the temperature and the loading condition.

The system function is represented by the finite element analysis, which can be calculated for a discrete set of system parameters. The result of the FE analyses depends on several parameters. Besides the mesh definition, which is used for analyzing the structure, the analyses type (linear vs. nonlinear calculation) has an influence on the accuracy of the calculations. Within this work, a nonlinear optimization will be performed in order to enhance the optimization accuracy.

3. OPTIMIZATION PROCESS

On the basis of preparatory multidisciplinary analyses, the topology for the integration of the LOX compartment into the stage, which is new for the Ariane 5ME structure, was fixed. Thus, the topology of the upper stage is given and shall not be modified.

The initial design of the Ariane 5ME upper stage is derived from the ESC-A version, which is already in use for GTO missions. The geometry of the structure was adapted to the new functional needs. Since this new predesign was not optimized for the loads of the Ariane 5ME, the margins of safeties within the structure did not meet the mass target within the required margin philosophy.

On the bases of this information, Level 3 investigations were performed to improve the structural design regarding mass. For this task, two different optimization strategies, the Free Shape Optimization (FSO) and the Shape Optimization (SO) were used iteratively, cp. figure 3.

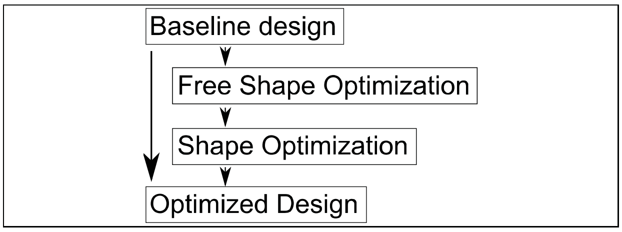

FIG 3. Established optimization procedure.

Both, the FSO and SO are implemented in the Altair software suite HyperWorks, cp. /5/. The FSO can only be performed with the proprietary OptiStruct optimization code, which includes FE capabilities. This FE solver can optimize linear FE models.

The SO can be performed using OptiStruct or HyperStudy. The latter one gives the user the advantage to integrate other FE solvers like MSC.NASTRAN. The advantage of using NASTRAN is that all design criteria, which are defined for the structure, can be integrated into a nonlinear FE analysis. This analysis does not only contain geometrical but also physical nonlinearities. This leads to precise calculations of stress and deformation.

After these two optimization steps are performed, the optimized design is obtained.

3.1. Free shape optimization (FSO)

The goal of the free shape optimization in this framework is, to get engineering information about the structure. Since it is not possible yet to couple the FSO with a nonlinear FE solver, only linear calculations can be performed.

3.1.1. Preprocessing procedure

The FSO is performed using Altair OptiStruct by several different steps:

1) Import and prepare the NASTRAN *.bdf File

In the frame of this project, the weakness of OptiStruct is that only linear calculations are possible. This leads to a imprecise mass estimation for the investigated. Thus, the optimization process should not be finished right after the free shape optimization.

In order to establish the FE model in HyperMesh, some preprocessing has to be performed. Firstly, after the BDF-file import, all nonlinear material data cards (MATT1) have to be deleted. The MATT1 cards are not supported within OptiStruct Version 10. Within the parameter card, the SCREEN parameter should be activated to get an output in the f06 file containing information regarding errors.

To get the stress levels at the components surface, shell elements have to be generated. These elements are added to the original BDF model and will be kept throughout the OptiStruct analyses.

2) Integration of the optimization constraints

Before the preprocessing is started, an in house software tool is filled with the general information about the model: The material data and the allowable stress are essential input within this step. Furthermore, the working directory and the temperature intervals for stress post processing have to be set.

In collaboration with Altair Germany, different scripts were provided to allow the integration of the constraints into the optimization model:

- Generate temperature based element sets
- Derive material properties for different temperatures within the material
- Connect the element sets to the corresponding material properties
- Set the maximum von Mises stress considering the temperature in the element sets and set it as a constraint
- Set the maximum shear stress considering the temperature in the element sets and set it as a constraint

The scripts allow the modification of the imported NASTRAN model in order to perform linear analysis with OptiStruct.

3) Integration of the optimization objective

For the current investigations, the mass is set as the optimization objective.

4) Start the optimization

3.1.2. Generated output

The FSO uses the integrated morphing tool to generate advantageous shapes. This is done without any input needed from the specific mechanical engineer leading to a common understanding about how the structure can be modified to lower the needed mass.

The free shape optimization provides an optimal shape for linear FEM as shown schematically in figure 4.

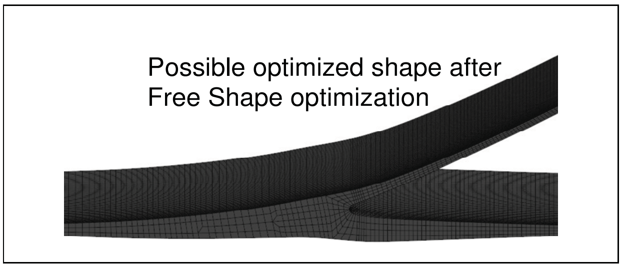

FIG 4. Possible optimized shapes calculated by FSO.

Additionally the main stress peaks can be visualized as shown in figure 5. The data provides information about the shapes, which can improve the structural performance. These shapes can be simplified and checked regarding production constraints. Afterwards they can be integrated into the FE model for further optimization.

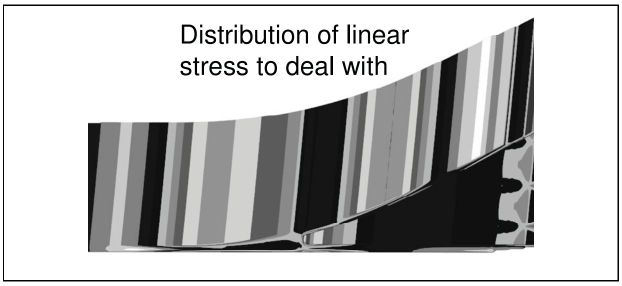

FIG 5. Stress levels at the beginning of the FSO.

The FSO is directly implemented into the OptiStruct FE solver and can therefore consider information of every single finite element. This information is then used to modify all elements within the next calculation step. The convergence rate is therefore quite high.

3.2. Shape optimization (SO)

Depending on the predefined shapes, the shape optimization will search a combination leading to a mass reduction. The different shapes have to be user defined and can be superimposed by the optimization algorithm.

The predefined shapes can also be used to integrate production based constraints into the model at this point, which leads to more realistic designs. Nevertheless, the generated weight gain may in some cases be reduced in comparison to the free shape optimization since more constraints are considered.

3.2.1. Preprocessing procedure

The preprocessing of the shape optimization model for the user optimization software HyperStudy is based on the original NASTRAN bdf file. Thus, the bdf file has to be imported into HyperMesh. Afterwards, the desired shapes can be modeled using the HyperMorph tool, cp. figure 6. For every shape, also a design variable is generated.

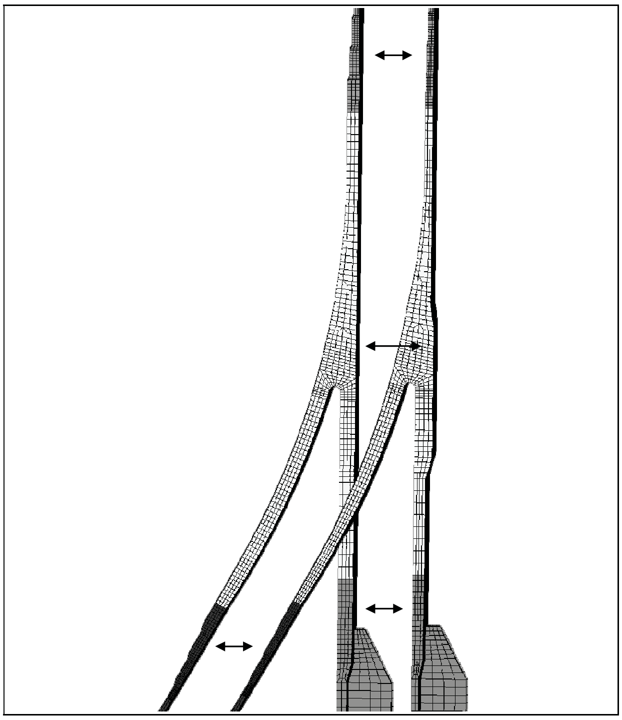

FIG 6. Predefined shapes for SO.

After all shapes are defined, they are exported to a finite element definition file, which contains the shapes for application within HyperStudy. At this point, special attention has to be put on the coordinate system, in which the nodes of the shapes are defined. HyperMesh (Version 10) does not export the coordinate system of the nodes at this step.

Since the weight has to be assessed, the NASTRAN weight generator can be activated using the parameter "GRDPNT". This will output the weight information for the whole structure in the f06 output file generated by NASTRAN, cp. /6/.

Within HyperMesh also shells elements can be integrated on the surface of the components, if needed for post processing. This is recommended for sections characterized by bending stress states in order to assess the magnitude of the outer fiber stress correctly. The procedure is the same as within the free shape optimization.

After all preprocessing is performed, the NASTRAN BDF-File is exported. On this basis, the template for the optimization within HyperStudy can be generated. Therefore the bdf has to be imported using the Template-Editor. It is used to replace the static node definition cards with the dynamic shape definition.

Finally, the constraints have to be included into HyperStudy. Therefore, element sets of the same temperature region are to be derived and exported to HyperView, which is used to data postprocessing. The op2 result file of the NASTRAN calculation is loaded into HyperView and the stress plots for the corresponding element sets are generated. This stress data is then printed into text files in a way that for every element in the element set the maximum stress is given.

Within HyperStudy, these files will be imported. They can be integrated as a response into HyperStudy and processed using the "max(..)" function. Finally, the maximum stress for every temperature region is derived as a response. For each of those responses, constraints can be integrated into HyperMesh to consider the temperature dependent material allowables.

3.2.2. Generated output

The main outcome of the final shape optimization is the structural weight and the optimized shape considering manufacturing aspects. At this point conclusiveness checks are performed and the final shape should be recalculated using the traditional NASTRAN deck. If in these checks the margins of safety are correct, the resulting shapes have to be checked in the design and manufacturing department regarding the geometrical requirements. Additional checks have to be performed regarding the changed characteristics in dynamics and fatigue, since the structure is significantly lighter and higher loaded after the optimization.

4. OPTIMIZATION OF THE Y-RINGS FOR ARIANE 5ME UPPER STAGE

Based on the optimization process, which has been described in chapter 3, the Y-rings of the Ariane 5 ME bare tank were optimized. The results will be outlined in the following sections, cp figure 7.

The shape optimization is performed once with the boundary conditions used for the initial free shape optimization.

For all optimization runs, the margin policy of 20% on yield and ultimate stress was used. The material allowables for yield and maximum stress as well as for the maximum strain criterion for the welds are considered. The general loads are derived from stage specifications to represent the loads that act on the tank over its interfaces. Furthermore, specific forces are integrated to consider additional loads on the lower Y-ring, which are correlated to inertia forces of the structure itself and the beared propellant masses.

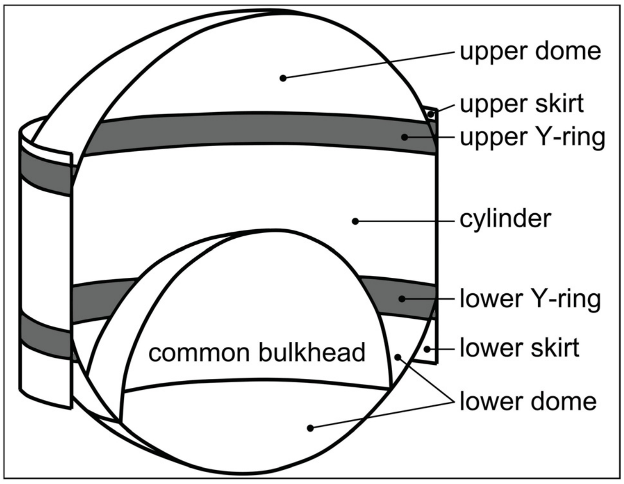

FIG 7. location of the Y-rings in the A5ME upper stage.

4.1. Free shape optimization (linear FEM)

For the optimization of the structure one dimensioning load case was found. Within the free shape optimization, only the Y-rings were optimized. Thus, the adjacent structures were not in focus of the variation. The resulting shapes are depicted in figure 8 and figure 9. For the upper Y-ring a reduction of the thickness in the area of the welds can be observed. Additionally a shoulder is integrated on the outer side of the tank at the intersection point of dome and upper skirt. This shoulder is characteristic for all Y-rings, which were optimized. The computational time for the free shape optimization (10 iterations) was 15h.

Undisturbed cylinders and spherical shells show only membrane stresses under internal pressure. But in this transition area the change in curvature and theoretical gap in displacements of cylinder and sphere leads to disturbing bending moments, cp. /7/. To compensate these additional bending moments such an eccentric shoulder is beneficial.

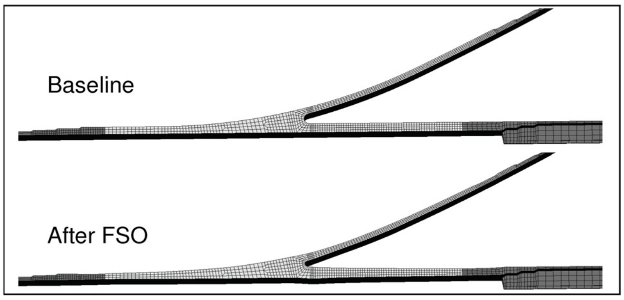

FIG 8. Free shape optimized upper Y-Ring.

For the lower Y-ring similar observations were made, even if the shoulder at the outer side of the ring is not that dominating in this case. Additionally, the small thickness of the weld located at the interface to the cylindrical part of the LH2 tank shows, that the internal pressure and the general loads act contrary leading to lower loads in the pressurized cylinder than in the skirts.

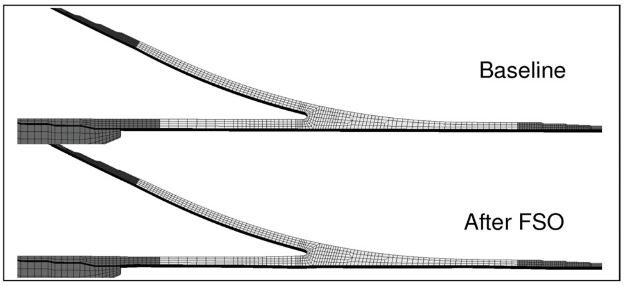

FIG 9. free shape optimized lower Y-Ring.

In total, 6% mass reduction was found performing the free shape optimization of the upper and lower Y-ring. Since the margin of safety is now feasible within the linear analyses coming from -20%, this is a very good tendency. Based on the shapes coming from this analysis, a shape optimization was performed which is described in the section shape optimization.

4.2. Shape optimization (nonlinear FEM)

The optimization of the bare tank structure has to consider the nonlinear effects of temperature gradients and pressure loads. In general, the stress is expected to lower if these nonlinear effects are taken into account. Therefore, a shape optimization using nonlinear FEM was performed to quantify the performance of the optimized structure. The loads were kept unchanged.

The shapes, which have been defined for the shape optimization, are derived for the original design of the rings under consideration of the initial free shape optimization. For each Ring seven different shapes are defined:

- Thickness for each of the welds (3 for each ring)
- Thickness of shoulder region (1 for each ring)
- Radius / Location of the small radius at the intersection point (3 for each ring)

The translation of the radius influences the stress level in the Y-ring significantly. The corresponding design variables are shown in figure 10 and figure 11.

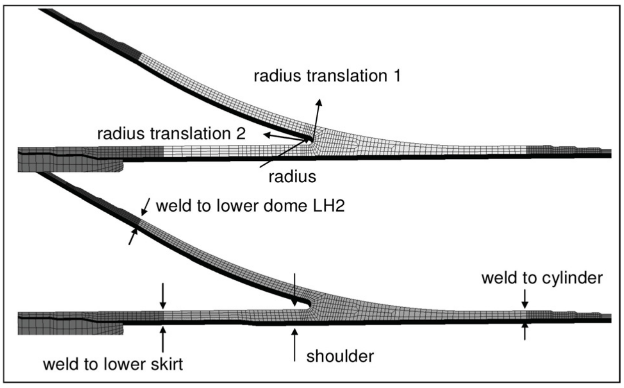

FIG 10. Basic design variable for lower Y-ring.

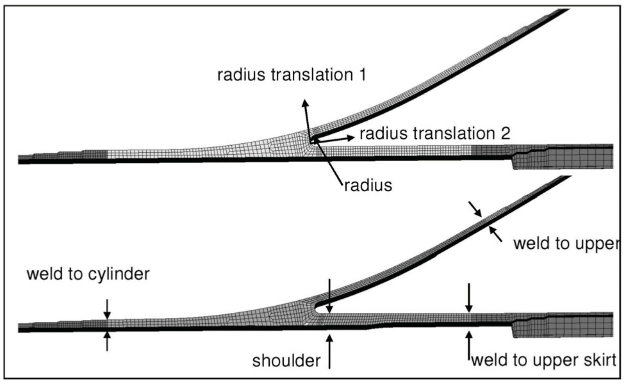

FIG 11. Basic design variables for upper Y-ring.

For both rings this leads to a total number of 14 design variables to reduce the mass. Each run to determine the stresses for all 4 load cases (Min/Max and Yield/Ultimate) in the considered flight phase takes about 12hrs, leading to an initial calculation time of ~7 days for the identification of the first search direction. The overall optimization took about 12 days.

The length of the shoulder is 36 mm in direction of the pressurized cylinder and 51 mm to the upper/lower skirt. The values are measured from the radius as shown in figure 12.

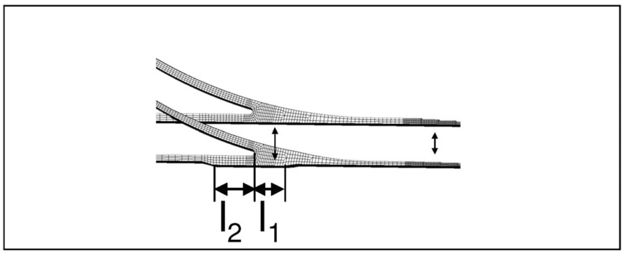

FIG 12. Dimensions of the integrated shoulders.

The resulting shape of the upper Y-ring is shown in figure 13. Also for this optimization run, the shoulder at the intersection point is beneficial for the mass budget.

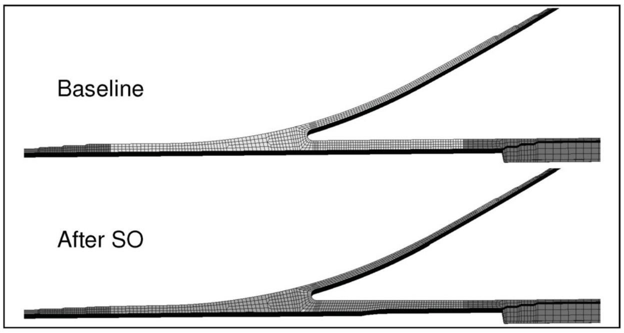

FIG 13. Shape optimized upper Y-Ring.

The resulting shape of the lower Y-ring is shown in figure 14. The thickness of the shoulder is smaller than for the upper ring. This is in accordance to the free shape optimization results.

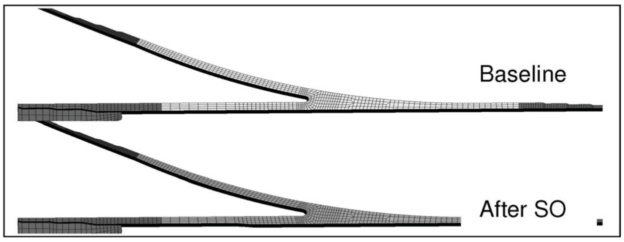

FIG 14. Shape optimized lower Y-Ring.

A total weight reduction of 5% was verified with this optimization.

5. CONCLUSION

Even if the total mass difference between the optimization runs are very similar, the results differ in details.

Thickness of the welds can be reduced as anticipated by the performed optimizations. The mass reduction of the Y-rings is comparable to the one found in the initial Free shape optimization. This is due to two contrary tendencies:

1. Positive influence of nonlinear FEM

The peak stresses within the linear FEM results overestimate the more realistic nonlinear stresses. Structures, which are optimized using linear FEM contain an extra margin of safety in the areas of peak stresses. By reduction of this margin using nonlinear FEM the mass of the structure could be further reduced.

2. Negative influence of predefined shapes used for optimization

The shape optimization is limited to a number of predefined geometric shapes. These can be scaled and superimposed in order to optimize the overall shape. The definition of pre-defined shapes restricts the possible shapes of the Y-rings and with it the mass reduction. The predefined shapes are chosen in a way that manufacturing aspects are considered:

– Constant radius at intersection point
– Parallel surfaces in the area of the weld seams for clamping
– No complex curves surfaces but stepwise constant thicknesses

Due to the initial model geometry, which did not accurately comply to the restrictions, and the morphing algorithms, which evoke small variations in the geometry, small deviations from the production restrictions could not be avoided. Thus, the final shape has to be de-fined in collaboration with the design office. Detailed dimensions of the Y-Rings can be found in Annex 3.

The implemented optimization method efficiently improves the mass budget of the given structures. For the components under consideration, a 9% mass reduction seems to be achievable, if the influence of the design change is also considered in the adjacent structures.

Shape optimization using nonlinear FEM is a huge effort in terms of computational time. The additional value of these optimization runs regarding the mass estimation is relatively small. The two contrary effects that act on the nonlinear analysis nearly cancel each other within the analysis of the Y-Rings under the given loads. The main new information that can be taken from the nonlinear calculation is that geometric details of the structure can be designed in coherence with production boundary conditions. Thus, in the frame of the performed preliminary design of Y-Rings, the optimization using linear FEM leads to a faster mass estimation.

6. REFERENCES

/1/ Frey, B. (2010):
Thermalkonzept der Kryogenen Oberstufe von ARIANE 5 Midlife Evolution, 60. Deutscher Luft- und Raumfahrtkongress 2011, Bremen

/2/ ESA Requirements and Standards Division ESTEC (2010):
Space engineering - Buckling of structures, Noordwijk

/3/ Schumacher, A. (2005):
Optimierung mechanischer Strukturen: Grundlagen und industrielle Anwendungen, Berlin, Heidelberg: Springer

/4/ Lawden, D, F. (2006):
Analytical Methods of Optimization, Mineola: Dover Publications

/5/ Thue, R. B. et al (2010):
Application of Structural Optimization Technologies and Methods to Reduce Design Time and Improve Structural Robustness, in proceedings of AIAA SPACE 2010 Conference & Exposition, AIAA

/6/ MSC Software (2007):
MD R2 Nastran™ Quick Reference Guide, Volume 1, MSC. Software Corporation, Santa Ana

/7/ Öry, H. (1991):
Structural Design of Aerospace Vehicles I, SPACE COURSE, Institut für Leichtbau, RWTH Aachen, Proceedings, 1991