Topology and Topometry Optimization of Crash Applications with the Equivalent Static Load Method

Katharina Witowski*, Heiner Müllerschön, Andrea Erhart, Peter Schumacher, Krassen Anakiev

DYNAmore GmbH
Industriestr. 2
70565 Stuttgart
Germany

Abstract

This paper deals with topology and topometry optimization of structures under highly nonlinear dynamic loading such as crash using equivalent static loads (ESL).

It reports about experiences in the application of the ESL methodology on industrial problems from the automotive industry. LS-DYNA® is used for the nonlinear dynamic, for topology and topometry optimization GENESIS from Vanderplaats R&D is applied. The methodical investigations have been performed within a research project, founded by the association BMBF, with several partners from German automotive companies.
On the application of the method on large scale problems numerous problems are encountered. Setting up a fully automated and robust process on an HPC cluster with nested linear and nonlinear finite element analysis and optimization for multiple load cases turned out to be a challenging task.

The general objective of the investigations was to evaluate the suitability of the method for different types of crash and impact problems. The appraisement is with respect to quality and usability of the results and with respect to the numerical costs.

Introduction to the Equivalent Static Load Method (ESL)

The ESL method goes back to publications of Park et al. [1]. Using the ESL method, the original nonlinear dynamic optimization problem is divided into an iterative “linear optimization ↔ nonlinear analysis” process: Within each iteration step a nonlinear dynamic FE-analysis is performed, using LS-DYNA® for instance. From displacement fields $\boldsymbol{u}$ at several selected time steps of this analysis *equivalent static loads* are evaluated. Those respective equivalent static loads $\boldsymbol{F}_{ers}$ would generate the same displacement field under the assumption of linearity, i.e. $\boldsymbol{F}_{ers} = \boldsymbol{K}_{lin}\boldsymbol{u}$ using the linear stiffness matrix $\boldsymbol{K}_{lin}$, see Figure 1. This procedure comprises a kind of time discretization.

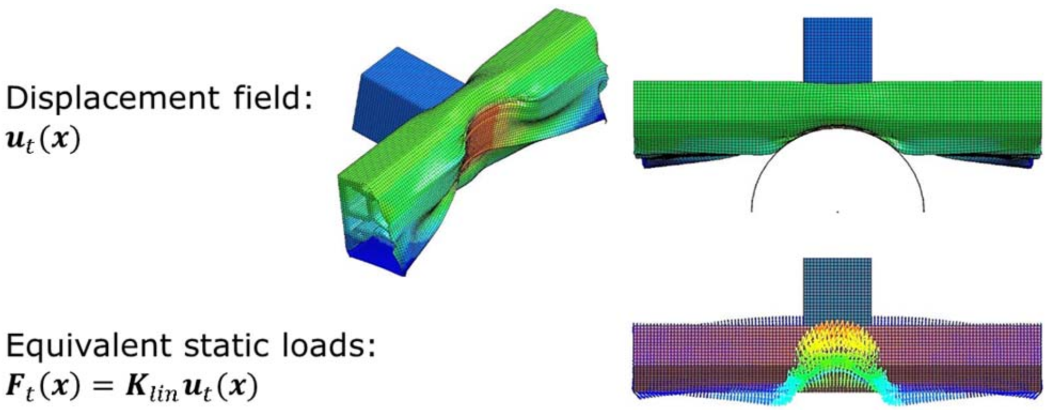

Figure 1: Equivalent Static Loads

A linear multiloading topology optimization of the system for the equivalent static load cases is then performed with e.g. Genesis, leading to an optimal density distribution for the given equivalent loads. A nonlinear dynamic FE-analysis of the system with the new density distribution and associated material parameters is performed, and convergence of the original optimization problem is tested, regarding the objectives and constraints. In the case of convergence the iterative process stops and the final design is found, if not, new equivalent static loads from the deformation fields are evaluated and the next iteration starts. Figure 2 shows the algorithmic realization of the ESL method.

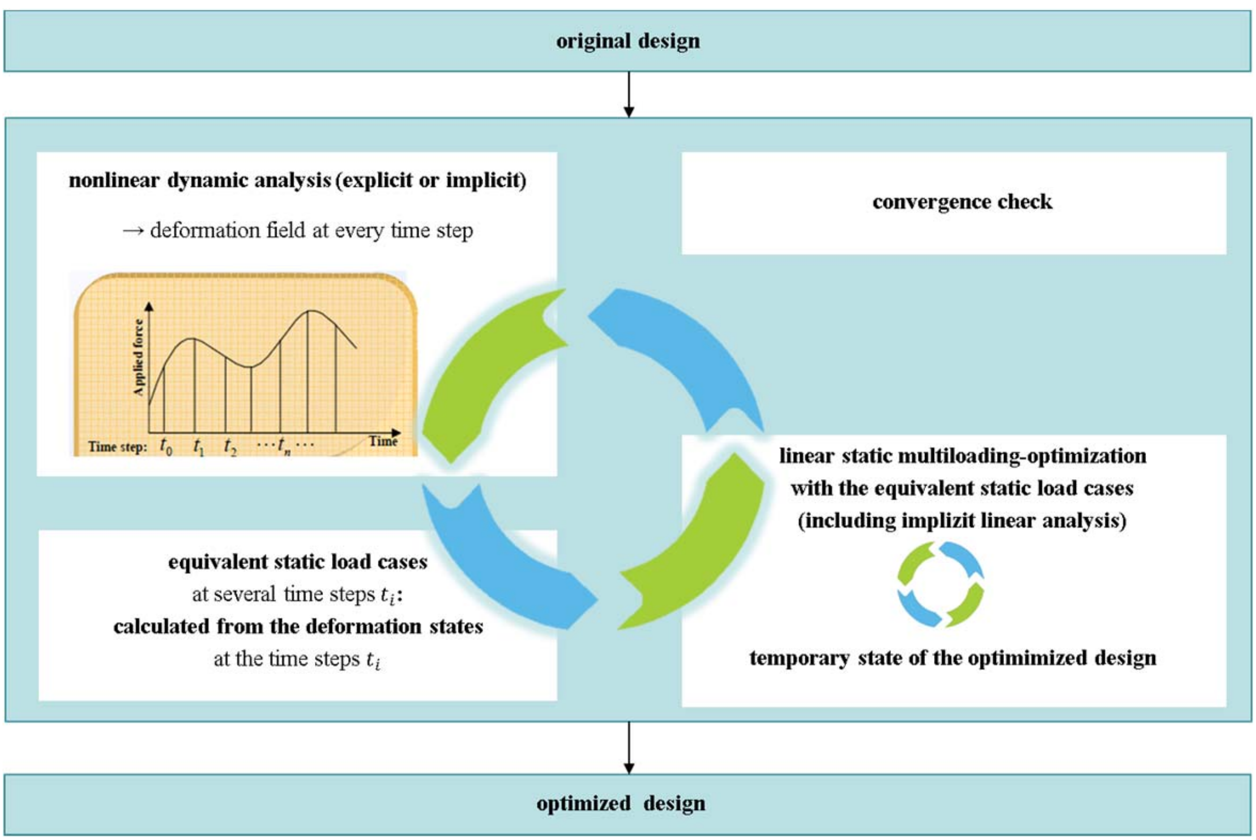

Figure 2: ESL-algorithm

Realization of the ESL method with Genesis® and LS-DYNA®

The ESL method utilizes the possibilities of the well established linear optimizing software Genesis. Several objectives and constraints can be chosen. The ESL method can be applied to different optimization tasks like topology optimization, sizing optimization, or topometry optimization. The data-transfer between the nonlinear analysis software and Genesis as linear optimization tool and the calculation of the equivalent static loads is realized by VR&D with interfaces for various established FE-solvers, for example LS-DYNA.

Within the linear multiloading optimization loop, Genesis performs internally an implicit linear analysis. Therefor the nonlinear LS-DYNA model has to be translated to a linear static Genesis model. Automation of this process is a challenging task. Many keywords and modelling features of LS-DYNA are supported, but not 100% yet. For our applications, the support of the system for this implicit linear input turned out to be eventually problematic. An interface called dyna2nastran had to be implemented for the transfer of missing keywords.

For the optimized design, a new LS-DYNA model for the next nonlinear dynamic LS-DYNA analysis is automatically written. For topology optimization applications it turned out to be necessary to merge areas with similar density into discrete density increments. Otherwise the number of parts and material cards for LS-DYNA would be extremely high.

An automated process with LS-DYNA and Genesis has been set up on an HPC environment process with combination of implicit linear and explicit nonlinear analysis for large models. We recommend to run the implicit linear multiloading optimizations on a machine with huge main memory (depending on the number of degrees of freedom), while running the explicit LS-DYNA analyses parallel on a cluster.

Preliminary investigations for the application to topology optimizations

A) Element size for topology optimization

Before starting a topology optimization, it's useful to investigate, which element size is necessary within the design space, because the whole design space is filled with solid elements, which are in general very intensive in terms of computing time. The mesh should be fine enough, that the analysis results are of sufficient precision.
Furthermore it has to be thought about the element formulation (discretization of displacements within the element and number of integration points).

B) Influence of elements with very low density for topology optimizations

For topology optimizations density distributions with clear zero or one densities are desired, i.e. full or no material. Intermediate density states are not desired, because these are unphyiscal. This is enforced through the so called SIMP-Ansatz for the material parametrization of the material parameters as a function of the relative density. Thereby the question comes up, what shall be done with elements, whose relative density tends to zero. Can theses elements remain in the analysis model or do they have a negative adulterant influence on the results. To answer this question, following exemplary model with a topology optimized density distribution is regarded, see figure 3.

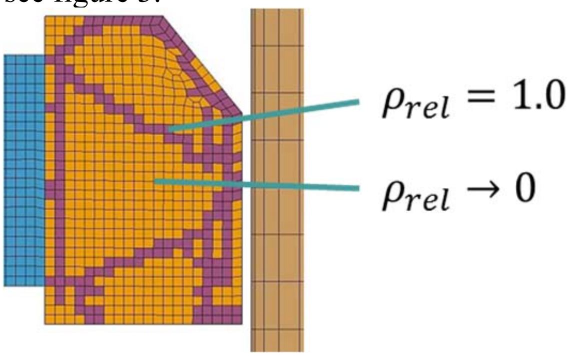

Figure 3: Example of a typical optimized density distribution

It's the optimized topology of an extrusion profile under pole impact with the objective of maximum stiffness. The topology is a kind of box girder with two cross beams. The intermediate elements have a relative density of almost zero.
This model is being analyzed for the load case pole impact, on the one hand with the elements of "almost-zero" density and on the other hand without those elements, i.e. these "almost-zero density" elements are deleted, see Figure 4.

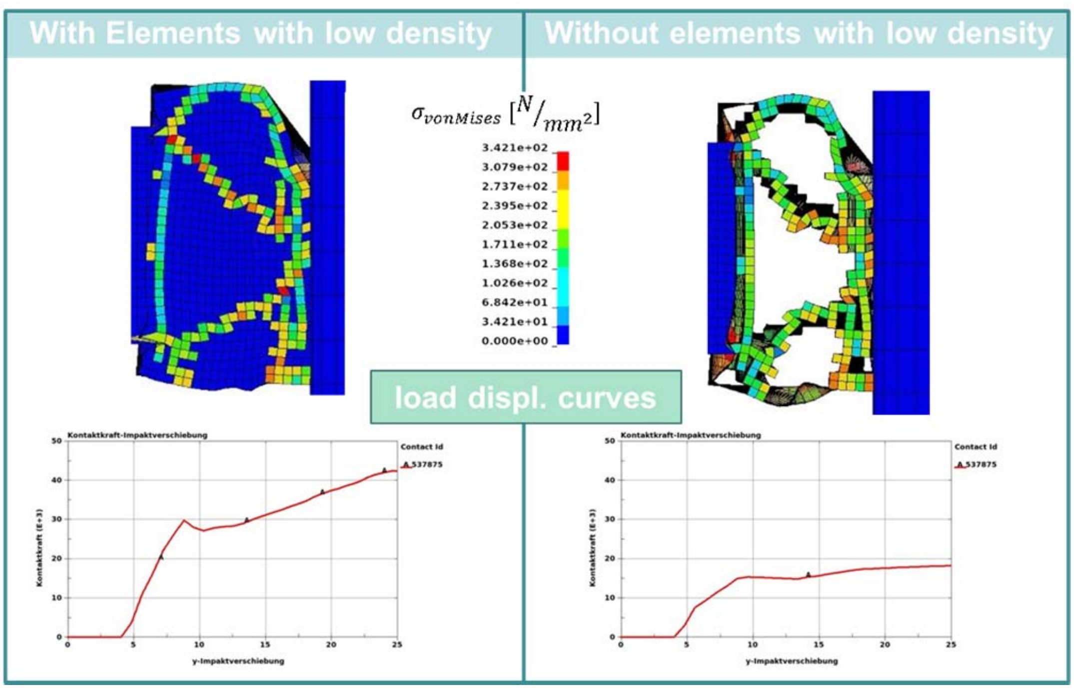

Figure 4: Influence of elements with "almost zero density" on the analysis results

From the displacements under pole impact as well as from the contact forces over intrusion diagram it can be seen, that the elements with "almost-zero" density have distinct stiffening influence on the results.
In the model, where these elements are removed, the cross beams buckle under the loading and the maximum contact force is 19 kN. In the other model, where those elements are not deleted, the buckling of the cross beams is precluded and the contact force grows up to 43 kN.
The reason for this is, that the zero-density-material is enclosed within a kind of chamber, and that the volume is conservated for von Mises plasticity plastic flow. I.e. this enclosed material

without density and therefor without stiffness can have a distinct stiffening influence on the results due to the volume conservation.
In addition, these “almost-zero-density” elements can cause numerical problems.

For the implicit LU-decomposition, as a part of the linear gradient based optimizations with Genesis, these “almost-zero-density” elements should remain within the model however; otherwise ruptured topologies would cause singular stiffness matrices.

C) critical time step size

For topology optimizations one could ask the question, whether the critical time step for the dynamic explicit analyses of the model goes down due to the decreasing density in individual elements. That would lead to increasing analyses times in the course of the optimization loops.
The critical time step of an element is $\Delta t_{krit} \sim \frac{L\sqrt{\rho}}{\sqrt{E}}$ with L being the relevant element’s edge length, density ρ and E-modulus E. Following the SIMP-Ansatz for the material parametrization as a function of the relative density, the current E-modulus is given by $E(\rho_{rel}) = (\rho_{rel})^p E_0$, with the initial value E_0 and p>1.0. Inserting this relation, for the critical time step follows:
$\Delta t_{krit} \sim \frac{L\sqrt{\rho_{rel}\rho_0}}{\sqrt{E_0(\rho_{rel})^p}} = \frac{L\sqrt{\rho_0}}{\sqrt{E_0}} \cdot \frac{\sqrt{\rho_{rel}}}{\sqrt{(\rho_{rel})^p}}$. The first term $\frac{L\sqrt{\rho_0}}{\sqrt{E_0}}$ is the critical time step for the initial “full” material. The second term is $\frac{\sqrt{\rho_{rel}}}{\sqrt{(\rho_{rel})^p}} > 1.0$, because exponent $p > 1.0$.
So we can conclude that the critical time step will not go down due to the density decrease within several elements, i.e. the analysis time will not increase in the course of optimization loops.
But of course due to distortion of the elements, the relevant element’s edge length can decrease leading to a decreasing critical time step. But this holds in general for explicit dynamic analysis.

Applications of the ESL method

A) Topology optimization of a front bumper with the ESL method

An aluminum bumper (600mm x 100mm x 160mm) shall be optimized for the three load cases pole impact with 29 km/h, static bending and static torsion, see Figure 5. Constraint for the load case crash is a contact force < 40 kN while the intrusion should be smaller than 70mm. For the load case bending, the maximum deflection has to be < 0,3867 mm and for the load case torsion, the wrinkling has to be < 3.554*10-3 rad. The bumper shall be an extrusion profile with web thickness ranging from 1.6 up to 3.5 mm. This ESL topology optimization was realized with the objective „maximize internal energy“ for the load cases bending and torsion. As an additional constraint a relative mass of 30% was enforced. The dynamic load case pole impact was discretized in time into 12 load cases for the calculation of the equivalent static loads. All load cases were weighted with 1.0. The design space was discretized with fully integrated hexaeder solid elements with 2mm edge length.

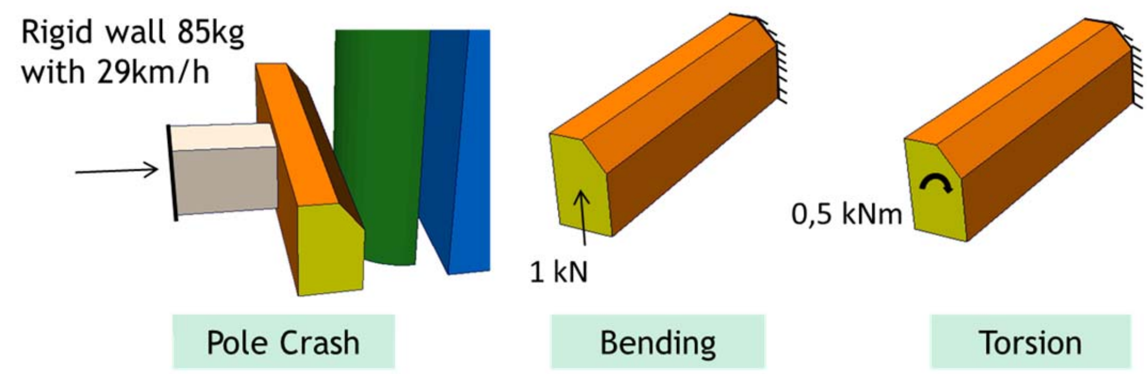

Figure 5: Load cases of the front bumper topology optimization

The optimized topology is depicted in Figure 6. It is a box girder (the outer shells are not depicted) with inner webs. It can be used as a starting point for a subsequent shape or size optimization e.g. with LS-OPT®.

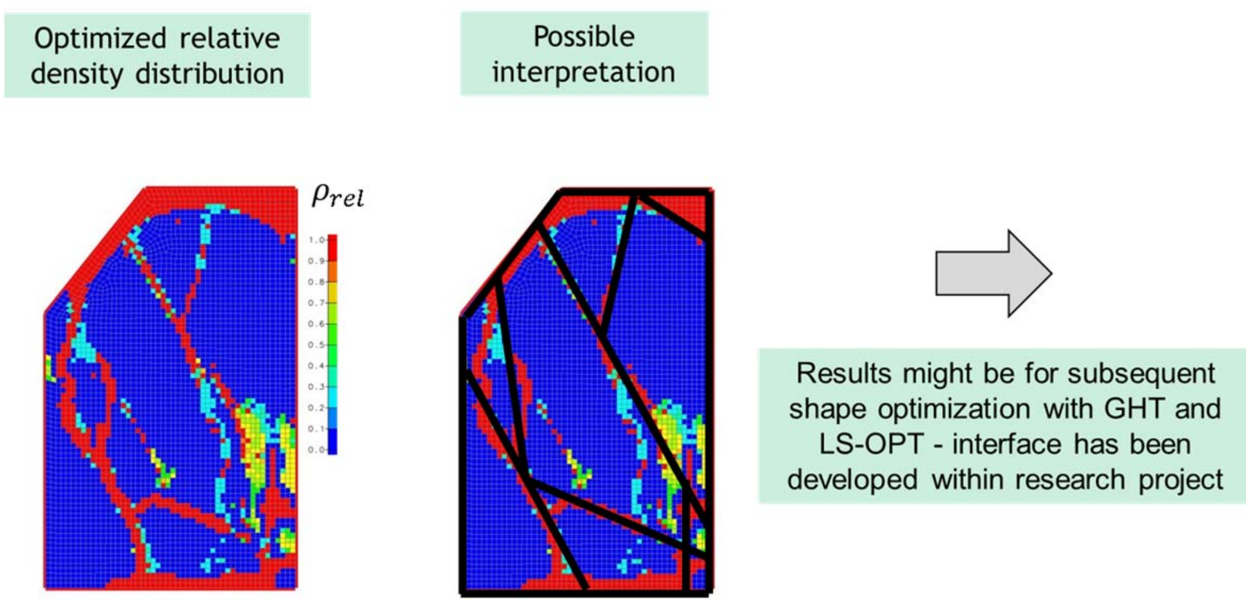

Figure 6: Optimized density distribution within the cross section of the bumper and possible interpretation

The deformations and plastic strains as well as the contact force as a function of time of this topology are given in Figure 7. The maximum intrusion of the load case impact is 67.1 mm, i.e. this constraint is fulfilled and the maximum contact force is 41.1 kN, i.e. very close to the requested limit of 40 kN.

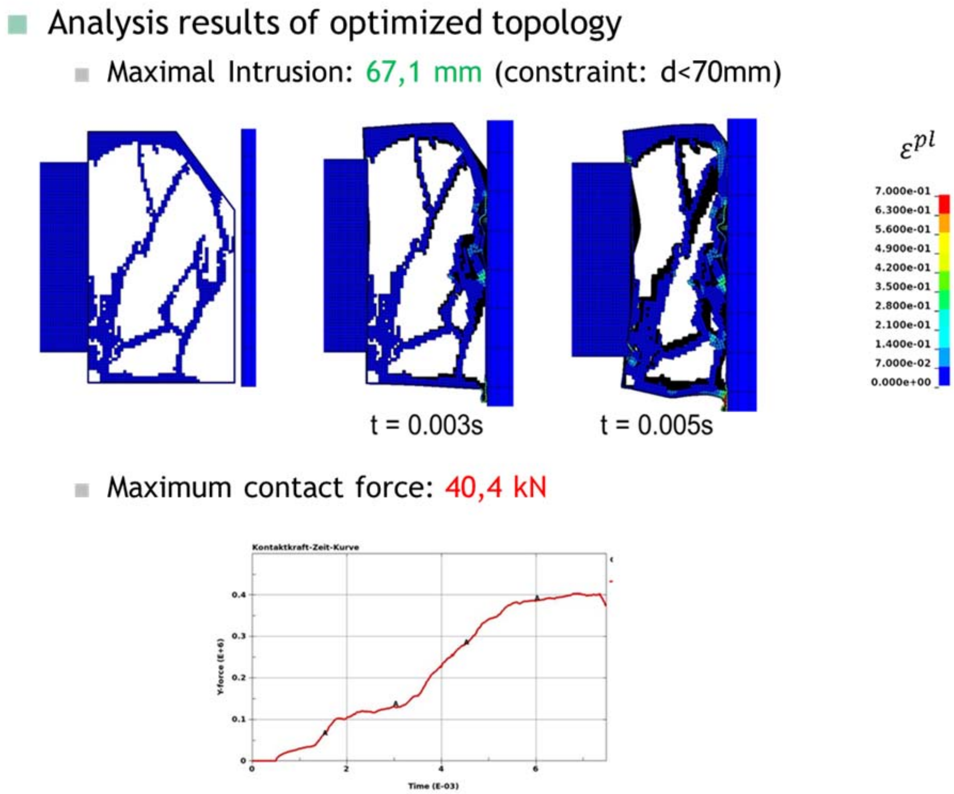

Figure 7: Analysis results of the optimized bumper

B) Topometry optimization of a inner hood panel with the ESL method

The following application is a joint project between MAGNA STEYR Engineering AG & Co KG and DYNAmore GmbH. The motivation was to develop a standardized method to design an inner hood panel, see Figure 8. This method should be able to take into account different package and geometry conditions. The main load cases are head impact (pedestrian safety) and stiffness. Expected result is an optimized design of the inner hood panel with an optimal HIC value for head impact and stiffness values for static load cases. The outer hood has a constant shell thickness t=0.6mm and material H220. The inner hood is a duplicate of the outer hood with coincident elements but separate properties with material DX 56D.
The design variables for the optimization are the thicknesses of every single element (Topometry Optimization). The thickness varies between 0.1mm and 5.0mm. To reduce the number of design variables 4 neighboring elements are clustered to have the same thickness variable during the optimization.

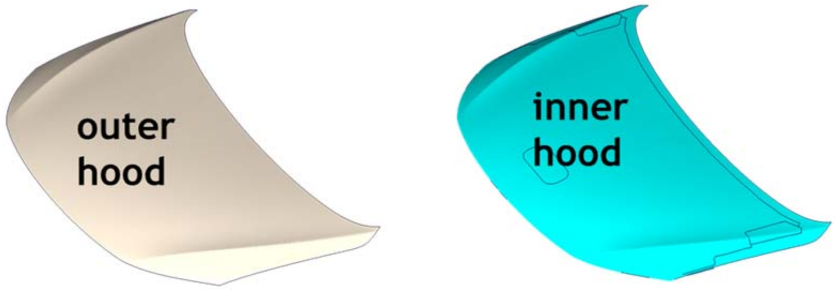

Figure 8: Application to a supporting structure of an engine hood

The LS-DYNA model for the nonlinear impact simulation is a reduced car model with blocking package elements in the engine compartment, see Figure 9. For the optimization with the ESL method the Genesis model is reduced to the hood with hinges and lock. The hood is supported with SPCs on the hinges and the lock. The preceding LS-DYNA simulation has been discretized with 9 equivalent static load cases (Δt=2 ms).

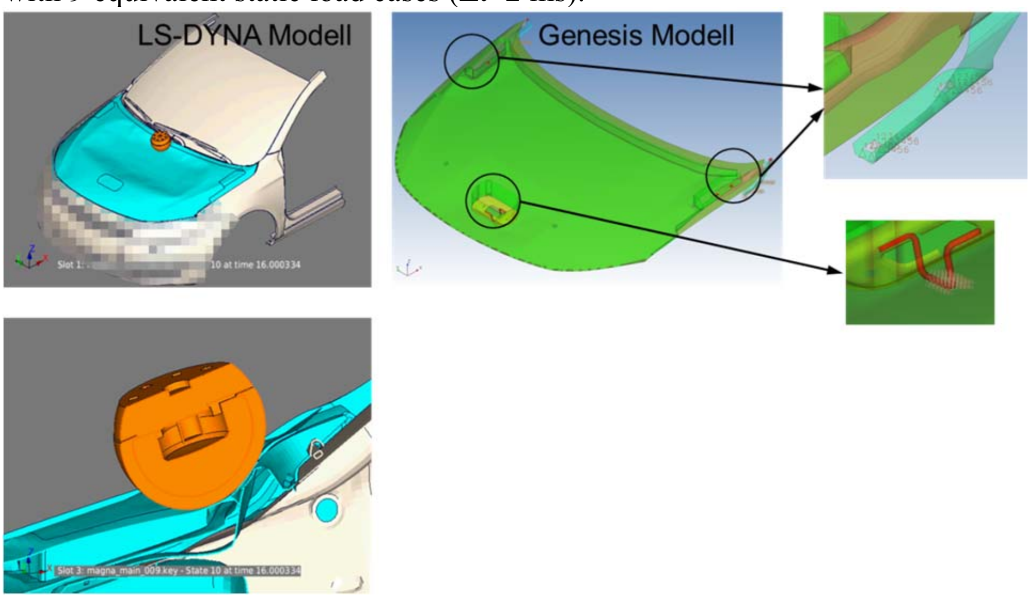

Figure 9: LS-DYNA model for nonlinear analysis and GENESIS model for linear optimization

As load cases head impacts at 11 points, and the static load cases corner bending, torsion, bending cross member and bending longitudinal member are taken into account for the optimization. 12 time steps are taken into account for the evaluation of the equivalent static loads from the dynamic load cases head impact.

As the HIC value can not be used as an objective in the linear inner topology optimization loop the optimization problem is reformulated with the objectives:

Maximize the deformation of the hood by avoiding contact with the stiff (rigid) underlying structure. And the strain energy should be maximized for the head impact load cases. Limit displacements in z-direction for about 80 points with maximum feasible deformation for the ESL load cases with large deformation are introduced as constraints to the optimization.

The thus optimized topometry results for the supporting structure of the engine hood and the consequently created CAD interpretation are given in Figure 10.

The HIC values for each LS-DYNA simulation are additionally exploited.

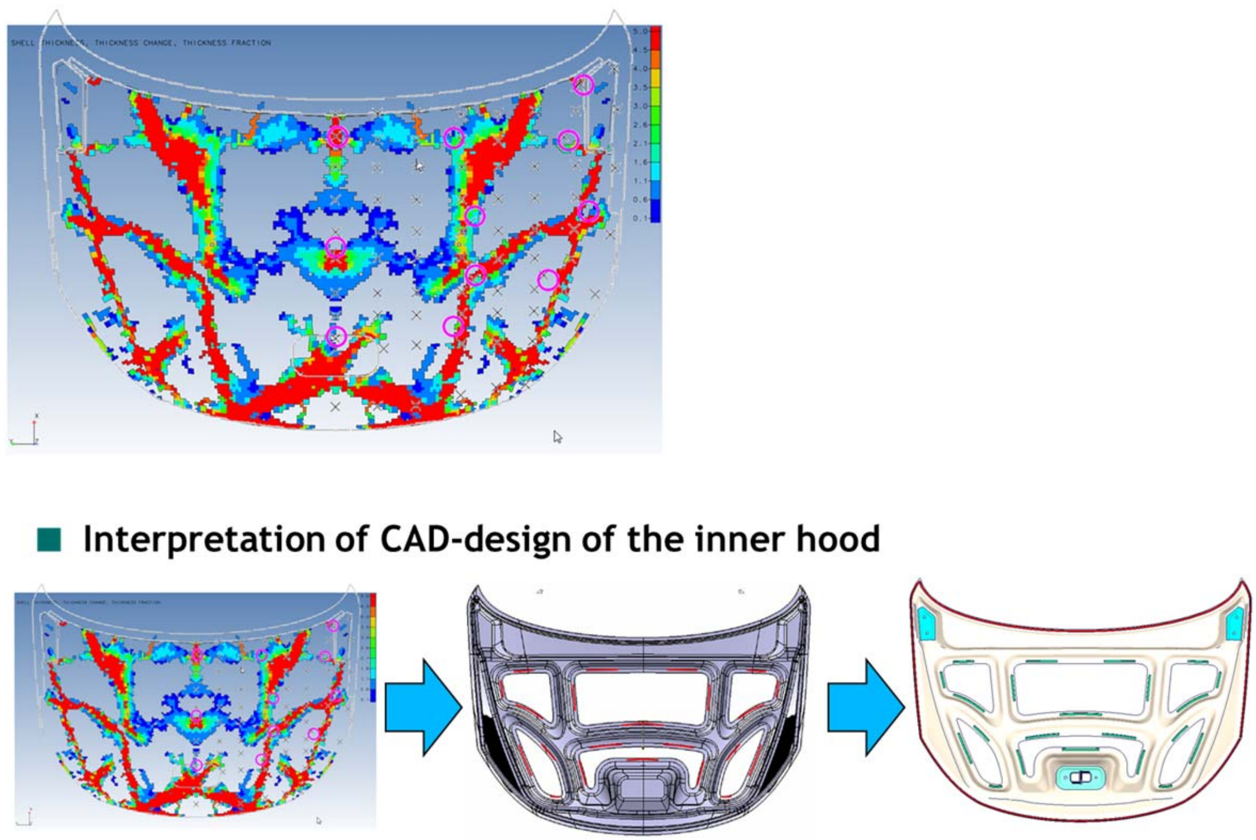

Figure 10: Optimized topometry and CAD interpretation of the inner hood

In a next step a nonlinear parameter optimization with LS-OPT will be performed on the basis of the preliminary CAD design to refine functional requirements. Parameters for the optimization with LS-OPT might be gauge thickness, properties of glue lines, geometric shapes based on morphing, etc.

Conclusions and Summary

The ESL method turned out to be promising for nonlinear applications with rather moderate deformations or with more spread deformations, for any contact problems, etc. as for example roof crash tests, pedestrian safety load cases, pendulum impact, drop tests or gear wheels, see Figure 11. The ESL method enables topology and topometry optimization for nonlinear problems and size or shape (parametric) optimizations with fewer nonlinear solver calls.

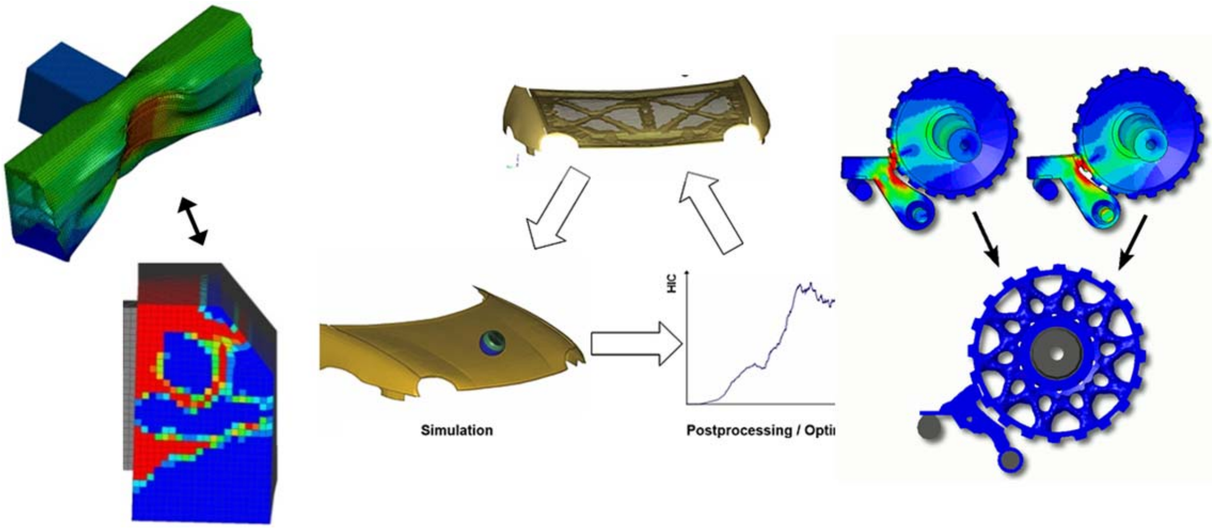

Figure 10: Typical application of the ESL method

Limitations of the method are due to the linearization of the optimization problem and the decomposition of the dynamic process into discrete static load cases. How far these simplifications bear for highly nonlinear structural behavior, like buckling in combination with plastic coating, needs to be further investigated.
Very local buckling, as depicted in Figure 11, is clearly outside the application area of the ESL method.

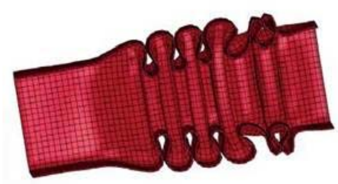

Figure 11: Buckling of a box girder under axial loading

Here, plastic hinges occur after exceeding the yield stress. The necessary force or the necessary moment respectively to introduce a large buckling deformation becomes very small than. For the evaluation of the linear equivalent static loads however, which are introduced as load cases for the linear optimization with Genesis, the linear stiffness is utilized: $\boldsymbol{F_{ers}} = \boldsymbol{K}_{lin}\boldsymbol{u}$. To enforce the same large buckling deformation under the assumption of linear material behavior, and extremely high force or moment resp. would be necessary, however. I.e. for applications with local buckling, the equivalent static loads of the ESL optimization are disproportionate high.

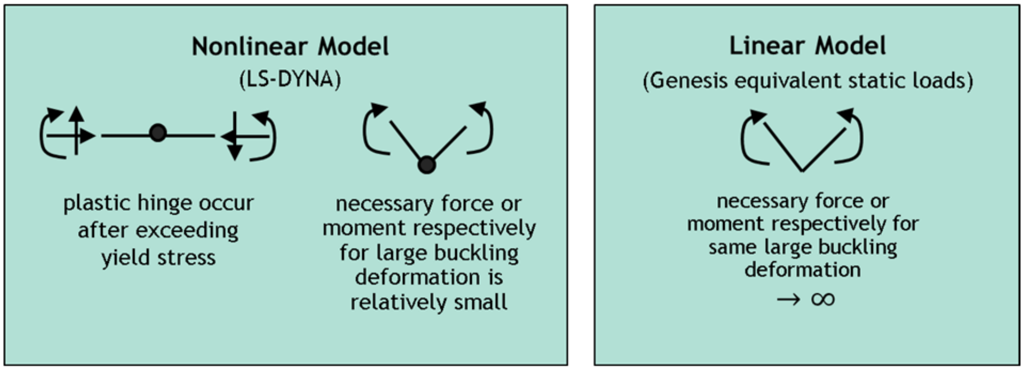

Figure 11: Limitation of the ESL method

The linear gradient based optimization program Genesis offers a large number of objectives and constraints and is very flexible in its application. It can be used for topology, topometry and sizing optimizations.

But – as it is a linear optimization program – objectives and constraints can be defined only for linear entities. This means, consideration of nonlinear responses is not directly possible, for example the minimization of a HIC value for head impact or of acceleration values is not possible as an objective, alternative criteria have to be established.

Constraints have to be defined for linear optimization as well. Consideration of constraints based on nonlinear responses is not possible as well and constraints are satisfied for the linear replacement problem. This might lead to violating the constraints for the real nonlinear problem.

References

Shin MK, Park KJ, Park GJ: Optimization of structures with nonlinear behavior using equivalent load. Comp. Meth. Appl. Math.,196, p.1154-1167, 2007

Park GJ (2011) Technical Overview of the Equivalent Static Loads Method for Non-Linear Static Response Structural Optimization. Struct Multidisc Optim 43: 319-337

Witowski K., Erhart A., Schumacher P., Müllerschön H. (2012) Topology Optimization for Crash. Conference Proceedings of the 12th international LS-DYNA User Conference, 3.-5. Juni 1012, Detroit.

Erhart A., Schumacher P., Müllerschön H. (2012) Topologie Optimierung mit LS-TaSC™ und GENESIS/ESL® für Crash-Lastfälle . Beitrag auf dem 11. LS-DYNA Forum, 9.-10. Oktober 1012, Ulm.