

# Truck Frame Optimization Considering Crashworthiness, NVH and Static Responses

Hong Dong<sup>1</sup>, Juan Pablo Leiva<sup>1</sup>, Brian Watson<sup>1</sup>, Weizhao Gao<sup>2</sup>, Feng Pan<sup>2</sup>

<sup>1</sup>OmniQuest

<sup>2</sup>ShareFEA Engineering Technology, Ltd.

## 1 Abstract

This paper demonstrates how to efficiently perform optimization for vehicle structures, taking into account nonlinear responses from LS-DYNA crash simulation, as well as responses from linear loading conditions such as NVH and Static. The optimization process is based on the Equivalent Static Load (ESL) method and uses an iterative process which utilizes the non-linear structural analysis results from LS-DYNA and the linear structural analysis and optimization capabilities of GENESIS. With this integration, the combined multidiscipline problem can be solved with only a few LS-DYNA simulations (5 to 10). In addition, large-scale optimization techniques, such as topology, topometry, topography and freeform, can easily be employed. The optimization process and results will be demonstrated using two examples: topology optimization of a beam cross-section under impact and static loading and topometry design of a truck frame under crash, normal modes, and static loading conditions simultaneously.

## 2 Introduction

Structural optimization techniques, coupled with linear finite element analysis, are being used to solve large scale design optimization problems efficiently with reduced computational cost. However, when nonlinear finite element analysis is used to analyze the structure, then performing optimization becomes cumbersome and computationally expensive due to the lack of efficient integration between the analysis program and optimization techniques. Optimization methods using the Equivalent Static Loads (ESLs) have been proposed to efficiently perform optimization based on a nonlinear finite element analysis using a linear structural optimization software [1]. The ESL is defined as the static load vector in the linear static analysis that produces the same response field as the nonlinear analysis. A wide variety of nonlinear analysis problems have been optimized using the ESL method [2][3][4][5][6]. A brief introduction about ESL methodology is given in the next section.

## 3 Review of the Equivalent Static Load Method

Optimization methods using Equivalent Static Loads (ESLs) have been proposed in the literature [1] to efficiently perform optimization based on a nonlinear finite element analysis. ESLs are defined as a set of static load vectors, applied on a model to perform linear static analysis that produces the same displacement field as obtained in the nonlinear analysis. A preliminary nonlinear analysis is performed to evaluate the nonlinear displacement field. This displacement field is used to compute the ESLs. Essentially, the displacements from the nonlinear analysis are multiplied with the stiffness matrix of the linear analysis model to compute the ESLs. Optimization is performed using these ESLs as linear static loads and the design is updated in the nonlinear analysis. The procedure is repeated until convergence is achieved.

The methodology of optimization based on ESLs has been described in the literature for different classes of nonlinear analysis problems. A brief overview of the associated theory is presented here. The governing equation of motion for a transient nonlinear analysis is shown below:

$$M\ddot{z}_N(t) + K_N(b, z_N(t))z_N(t) = f(t)$$

where  $z_N(t)$  is the displacement field over time. The time domain can be divided into a finite number of instances. At each instance, ESLs can be computed by taking a product of the linear stiffness matrix  $K_L$  with the nonlinear displacement at that instance, say  $t_a$  as shown in the equation below:

$$f_{ESL}(t_a) = K_L z_N(t_a)$$

Each of these ESLs,  $f_{ESL}(t_a)$  vectors are applied on the structure as a load case. Multiple loading conditions can be efficiently handled by the linear structural optimization without a significant increase in computational cost. The structure is optimized considering all these loading conditions.

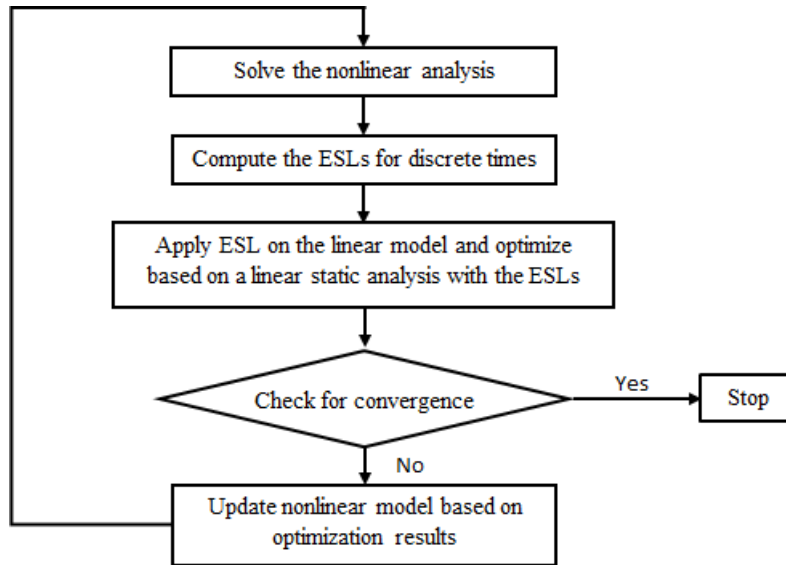


Fig.1: Flowchart for optimization based on ESL Method

As the optimization modifies the design, the ESLs are no longer equivalent to the nonlinear analysis result. The results from the linear optimization are used to update the nonlinear analysis model and a nonlinear analysis is done to compute the new displacement field. This process is repeated until predefined convergence criteria are satisfied. A flowchart illustrating the procedure is given in Figure 1.

#### 4 ESLDYNA - ESL Implementation for LS-DYNA

The algorithm and the ideas behind the ESL method are detailed in the previous section. In this section the implementation of the methodology to optimize for responses based on a LS-DYNA analysis is described.

ESLDYNA is an implementation of the ESL methodology to perform optimization based on a LS-DYNA analysis. GENESIS is used as the linear structural optimization package to perform the optimization. GENESIS itself uses the BIGDOT [7] optimizer developed by Vanderplaats [8]. The ESL for LS-DYNA implementation provides the capability to use structural optimization techniques in GENESIS for efficient design. The flowchart of ESLDYNA is shown in Figure 2:

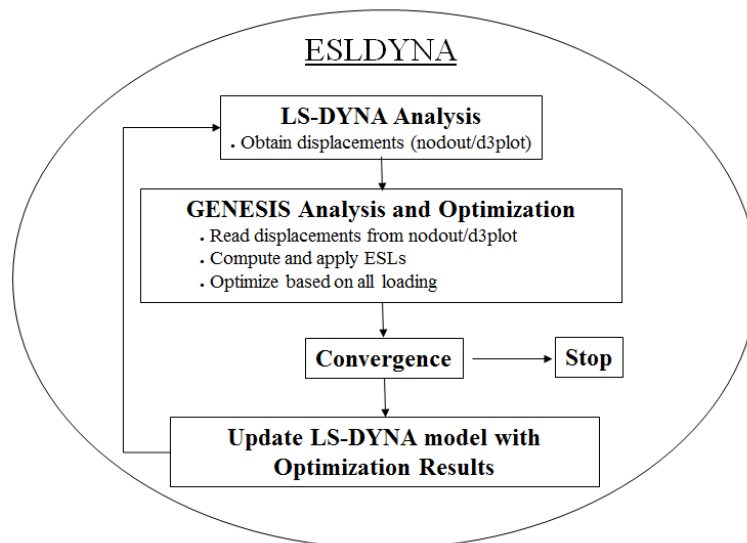


Fig.2: Flowchart of ESLDYNA for LS-DYNA

As a first step, a preliminary LS-DYNA analysis is performed to compute the displacements of LS-DYNA. Then ESLDYNA will read the displacements from LS-DYNA nodout or d3plot files. GENESIS will use these displacements to compute the ESLs and apply them on the structure. Each set of displacements for a given timestep will be treated as one load case in GENESIS for the analysis.

Based on the defined design data for GENESIS, optimization is performed. Once the optimization is done, the LS-DYNA model is updated with the results from the GENESIS optimization. After updating the model, LS-DYNA analysis is carried out to analyze for a new set of displacements. The new set of displacements (LS-DYNA nodout/d3plot files) is used in the subsequent GENESIS optimization run. Subsequent GENESIS simulations are launched using the RESTART option in GENESIS. The last design cycle of GENESIS is used as the starting point. The loop is repeated until the convergence criteria is met.

As discussed earlier, large scale optimization such as topology, topometry, topography, freeform, etc. can be performed for nonlinear analysis with ESLDYNA. The computational cost for the entire optimization typically would take about 5 to 10 LS-DYNA analyses.

ESLDYNA can be used to efficiently design the structure, but it does have some limitations that are inherent to the ESL methodology. As the ESLs are computed based on the responses from the LS-DYNA model, the methodology works well when equivalent responses exist in the linear regime. When equivalent linear responses are not available, then the nonlinear responses need to be described using one or more combinations of linear responses to achieve the same effect.

## 5 Examples

### 5.1 Topology optimization of a beam cross-section under impact and static loading

The beam is under impact loading and static bending, as well as torsional loading as shown in Figure 3.

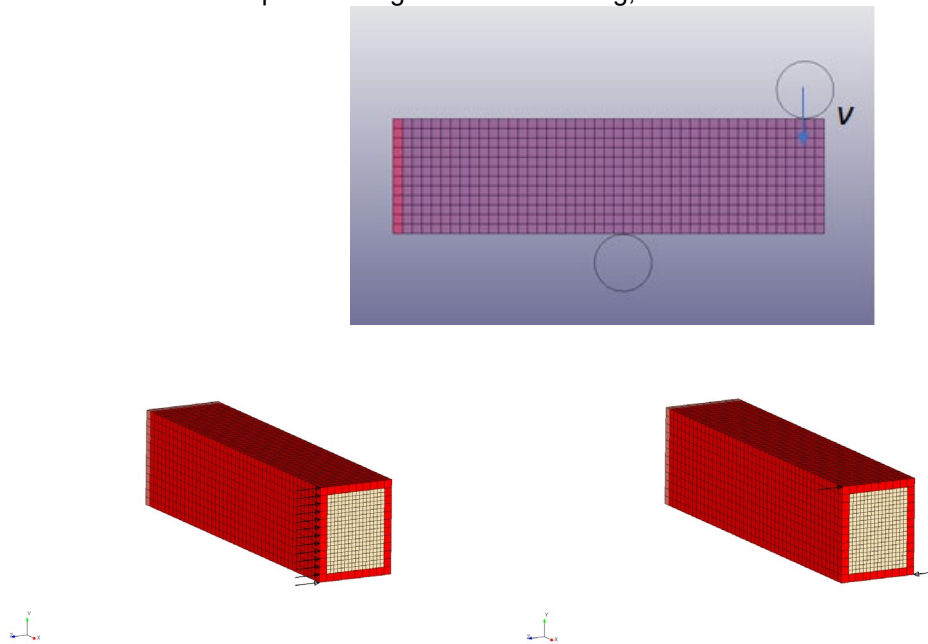


Fig.3: Loading conditions: impact, bending, and torsional

We would like to use topology optimization to design the beam cross-section such that it has certain stiffness but can also absorb energy during impact loading. The topology optimization problem is formulated as min. displacements and max. strain energy simultaneously for the ESL load case, which is to simulate the scenario of maximizing the internal energy for achieving energy absorption effect. Meanwhile, min. strain energy objective is used for static bending and torsional load cases. 30% of the mass will be kept for topology design region. The topology optimization result is shown as in Figure 4. Here extrusion and symmetry fabrication constraints are used for the beam cross-section topology. The material for the beam is piecewise linear plasticity material in LS-DYNA.

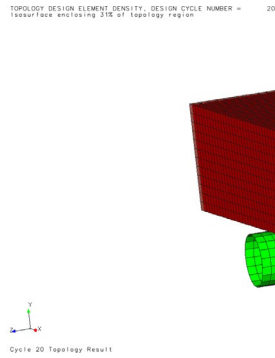


Fig.4: Topology optimization result of the beam cross-section

For this topology optimization, we used weighting factor as 1.0 for the conflicting objectives: min. displacements and max. strain energy. By varying the weighting factor, we can get a stiffer or softer beam according to the purpose of the design (Figure 5).

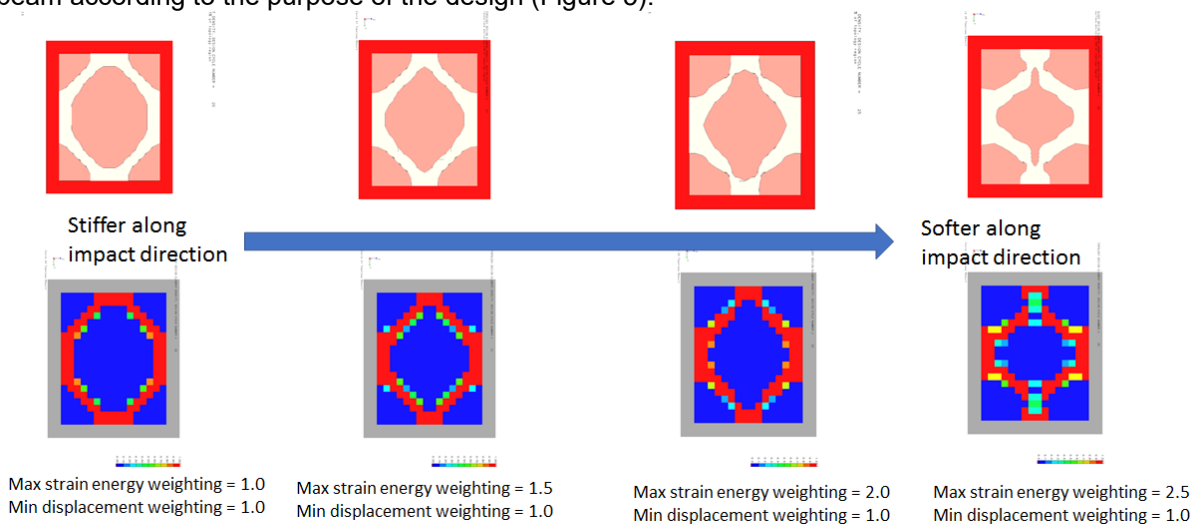


Fig.5: Topology optimization result of the beam cross-section with different weighting factor for stiffness vs. energy absorption

For the extreme cases, if we remove the maximize strain energy objective, we will get the stiffest structure along the impact direction, and if we remove the min. displacement objective, we will get the softest structure along the impact direction (Figure 6).

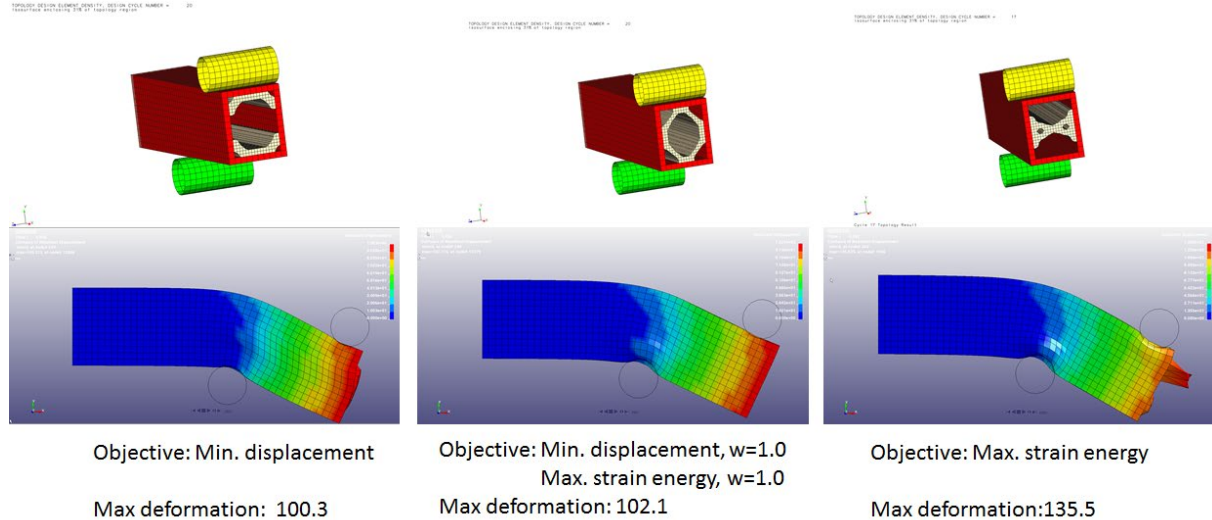


Fig.6: Topology optimization result of the beam cross-section with and without consideration of energy absorption

From the topology results, we can see the optimization tries to increase/decrease the length of the vertical member for the more/less energy absorption effect.

## 5.2 Topometry optimization of a truck frame under crash, normal modes, and static loading conditions

This is a simplified truck frame model based on the model at National Highway Traffic Safety Administration website:

<https://www.nhtsa.gov/crash-simulation-vehicle-models>

The truck is under 3-crash loading conditions: pole crash, frontal crash, and rear crash, as shown in the images below.

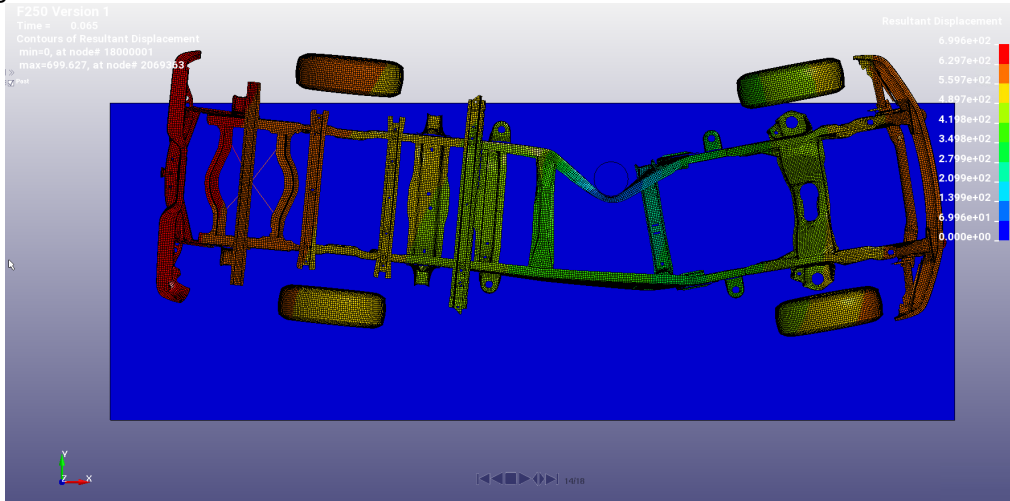


Fig.7: Pole crash

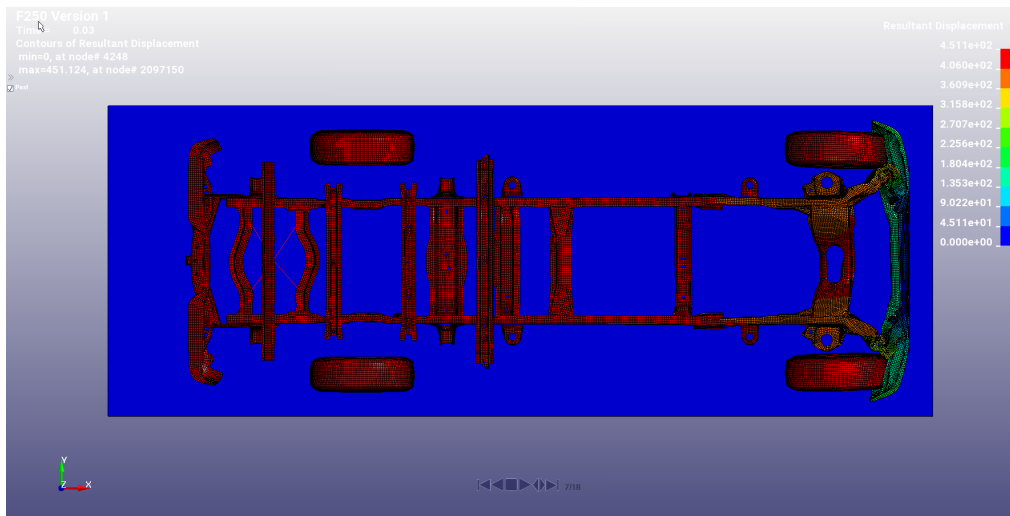


Fig.8: Frontal crash

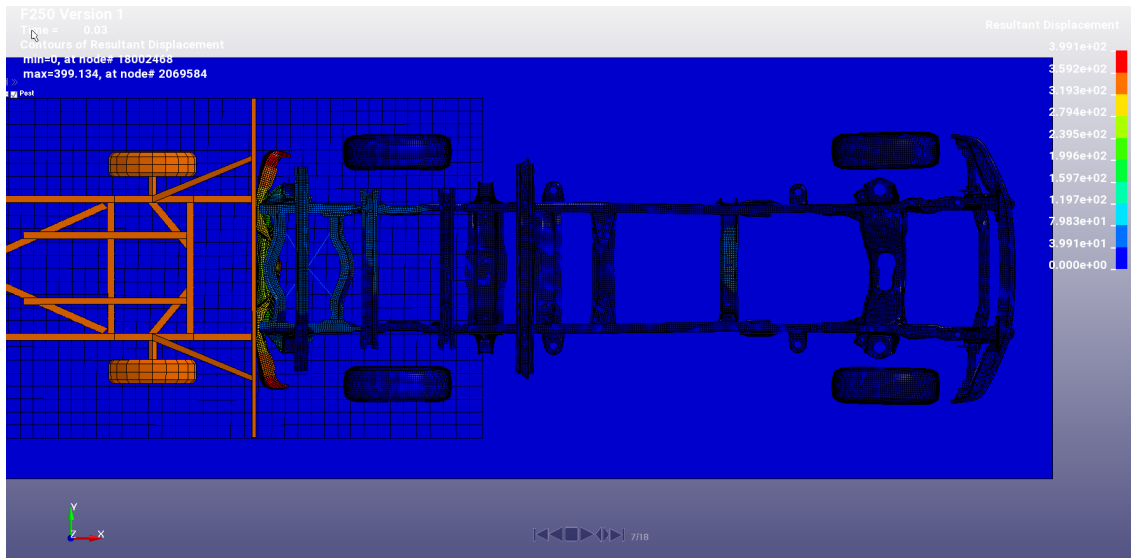


Fig.9: Rear crash

In addition to the crash loading, a static torsional loading is applied at the front shock tower location, and a normal modes loading is also applied for the optimization problem.

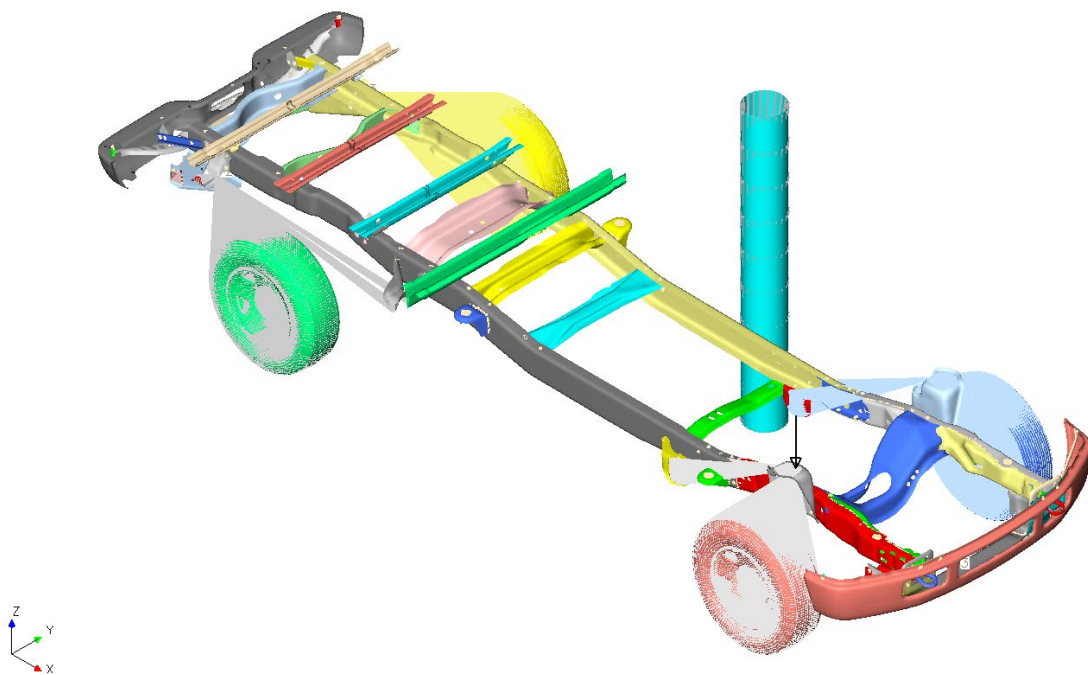


Fig.10: Static torsional loading

The goal of this design is to use topometry optimization to reinforce the truck frame such that the intrusion at impact location is reduced, as well as optimizing the static stiffness and the frequency.

The optimization formulation is as follows:

Objective:

- Min. intrusion for pole crash (time = 0.065)
- Min. intrusion for frontal crash (time = 0.01, 0.015, 0.02, 0.025, 0.03)
- Min. intrusion for rear crash (time = 0.01, 0.015, 0.02, 0.025, 0.03)
- Min. strain energy for static torsional loading

Constraints:

- Frequency of the first non-rigid body mode > 1.0\*Initial



Mass < 1.05\*Initial

Design region and design variable:

Thickness of each shell element. The thickness range is [Initial thickness, 2.0\* Initial thickness]

The design region is shown as in Figure 11. There are total 56005 shell elements in the design region. Symmetry and Coarsening fabrication constraints are applied. Coarsening is a special feature in GENESIS topometry design. It can help in reducing the number of design variables and in obtaining a more manufacture-ready design by enforcing multiple adjacent elements to have same thickness. With symmetry and coarsening applied, the number of independent design variables is reduced to 2541.

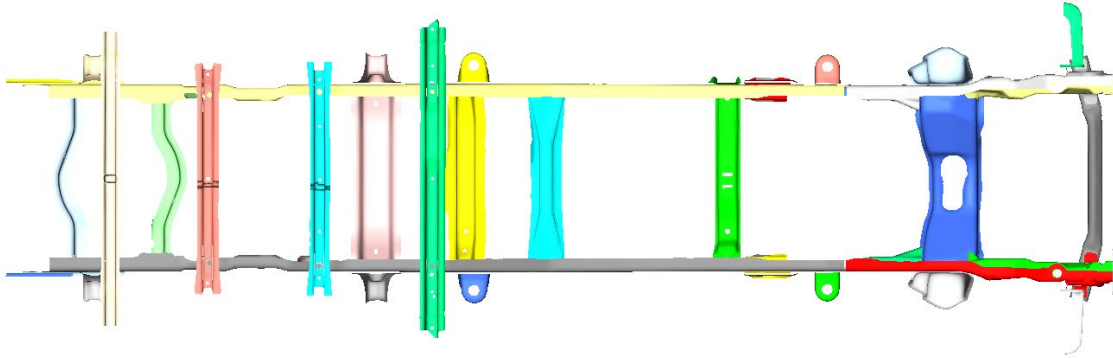


Fig.11: Topometry design region

For this optimization problem, there are 11 ESL load cases, 1 static load case, and 1 normal modes load case defined. The optimization process will search for the weak locations and use topometry design to reinforce them. In this particular problem set up, the thicknesses are not allowed to be reduced but the mass is allowed to be increased up to 5%. Alternatively, one might allow the shell thickness to be reduced or increased, while the total mass be restricted to be less or equal to the initial mass, which will redistribute the mass to the most critical locations for reinforcement while not increasing the total mass.

The thickness change from topometry optimization is shown in Figure 12, 13, 14, and 15. The blue color means no change, and the red color is the largest thickness change. The thickness change result shows that the reinforcement locations are close to the side, front, and rear impact locations. We can also see that multiple connection locations are reinforced, and the connection bars are reinforced, as well.

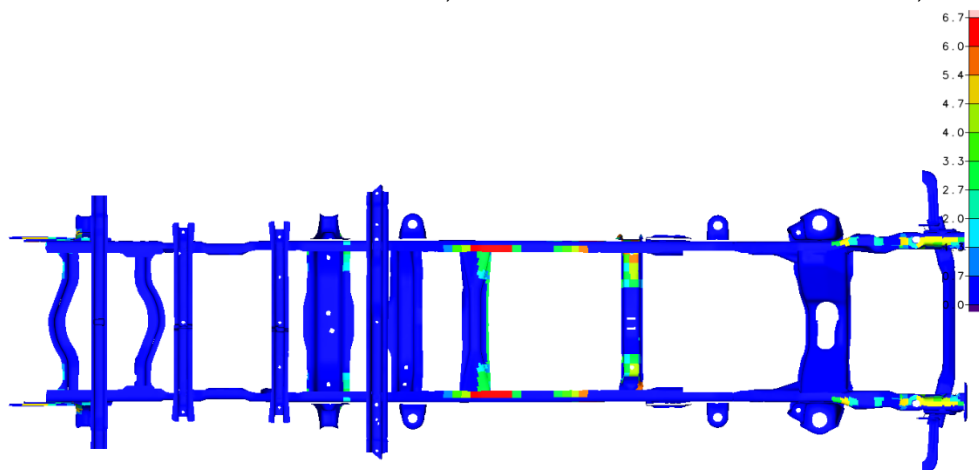


Fig.12: Thickness change – top view

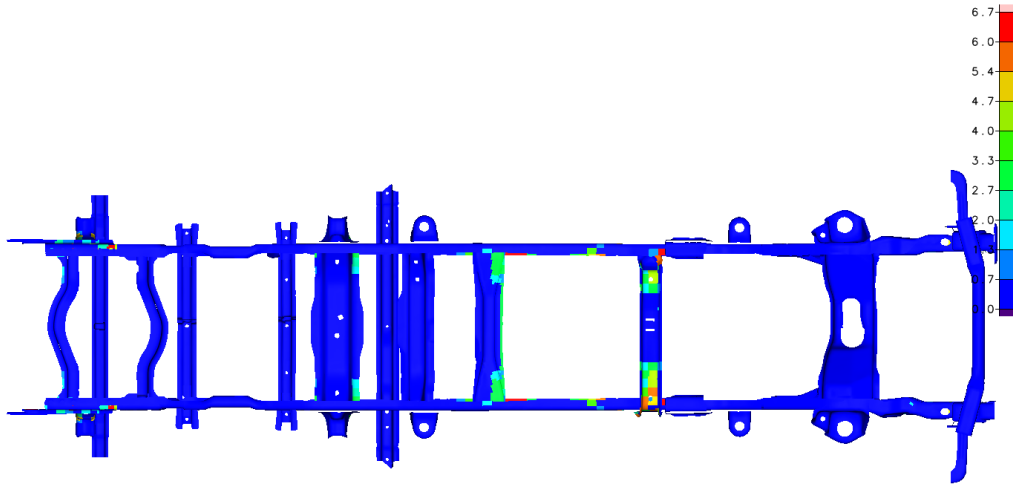


Fig.13: Thickness change – bottom view

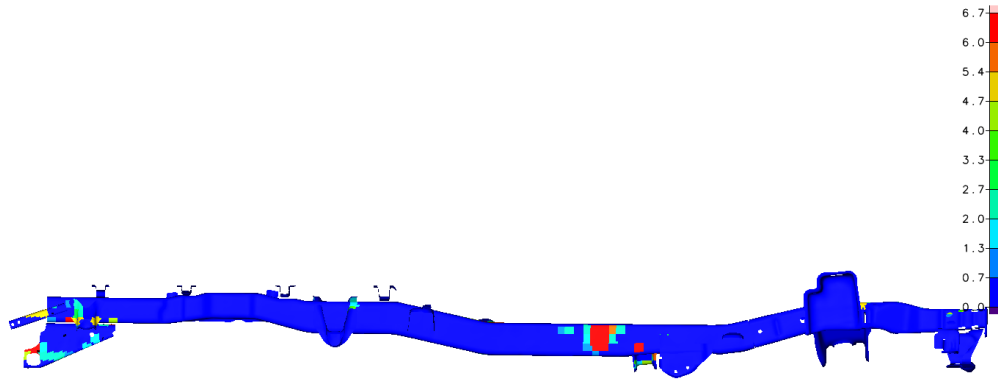


Fig.14: Thickness change – side view

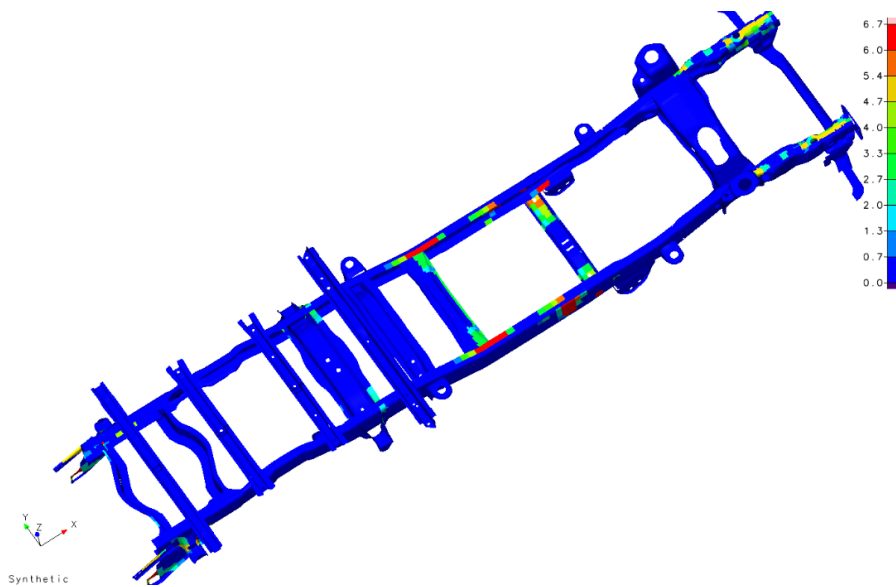


Fig.15: Thickness change – iso view



The objective changes are shown in the table below. The table shows the intrusion is reduced and the torsional stiffness is improved, as well.

Response/Load case Id	Loading step	Design Cycle 0	Design Cycle 25	Change
1 / 1 / RELDISP	Pole, time = 0.065	2.502555E+02	2.264023E+02	-9.5%
2 / 2 / RELDISP	Frontal, time = 0.01	6.508553E+00	4.823852E+00	-25.9%
3 / 3 / RELDISP	Frontal, time = 0.015	1.557543E+02	1.227816E+02	-21.2%
4 / 4 / RELDISP	Frontal, time = 0.02	1.557543E+02	1.227816E+02	-21.2%
5 / 5 / RELDISP	Frontal, time = 0.025	2.290658E+02	1.764109E+02	-23.0%
6 / 6 / RELDISP	Frontal, time = 0.03	3.013910E+02	2.240677E+02	-25.7%
7 / 7 / SENERGY	Torsional	1.948100E+04	1.396990E+04	-28.3%
8 / 9 / RELDISP	Rear, time = 0.01	5.207580E+00	3.261001E+00	-37.4%
9 / 10 / RELDISP	Rear, time = 0.015	1.207791E+01	9.128539E+00	-24.4%
10 / 11 / RELDISP	Rear, time = 0.02	5.813605E+01	5.775914E+01	-0.65%
11 / 12 / RELDISP	Rear, time = 0.025	8.992345E+01	8.678464E+01	-3.5%
12 / 13 / RELDISP	Rear, time = 0.03	1.032378E+02	1.079651E+02	4.5%

Table 1. Summary of objective changes

The entire optimization process is finished with 5 ESL cycles, i.e. 6 LS-DYNA analyses and 30 GENESIS optimization cycles. Table 2 shows the elapse time in each of the programs for all design iterations. Although GENESIS iterates within the linear structural optimization process, the run time is much shorter than LS-DYNA analysis time.

	LS-DYNA	GENESIS	Total
Elapse time (secs)	45721	1902	47648
Number of analyses	6	30	

Table 2. Elapse time and number of analyses

## 6 Summary

ESLDYNA utilizes the ESL method to couple LS-DYNA nonlinear structural analysis with the GENESIS structural optimization software. ESLDYNA allows its users to perform optimization based on both linear and nonlinear loading conditions. This implementation typically achieves an optimum quickly with few LS-DYNA simulations. Total computational cost is low, even for a large number of design variables. Two examples, one with topology optimization and another with topometry optimization, were presented in this paper. These examples demonstrate the optimization process and capability of ESLDYNA.

## 7 Literature

- [1] G. J. Park, Technical Overview of The Equivalent Static Loads Method for Non-Linear Static Response Structural Optimization. *Structural and Multidisciplinary Optimization*, 43(3), 319-337, 2011.
- [2] P. Adduri, J. P. Leiva, G. Quinn, and B. C. Watson, Car Body Optimization Considering Crashworthiness, NVH and Static Responses. 13th International LS-DYNA Users' Conference, June 8-10, 2014.
- [3] I. Kosaka, Improvement of Energy Absorption for the Side Member Using Topography Optimization. 11th International LS-DYNA Users' Conference, Jun 6-8, 2010.
- [4] S. I. Yi, J. Y. Lee, G. J. Park, Crashworthiness Design Optimization Using Equivalent Static Loads. *Journal of Automobile Engineering*, 226(1), 23-38, 2012.
- [5] ESLDYNA User's Manual, Version 19.0, Vanderplaats R&D, Inc.
- [6] K. Witowski, H. Mullerschön, A. Erhart, P. Schumacher, K. Anakeiv, Topology and Topometry Optimization of Crash Applications with Equivalent Static Load Method. 13th International LS-DYNA Users' Conference, June 8-10, 2014.
- [7] BIGDOT User's Manual, Version 2.0, Vanderplaats R&D, Inc.

[8] Vanderplaats, G. N., "Very Large-Scale Optimization," Proc. AIAA/USAF/NASA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Long Beach, CA, Sept. 6-1, 2000, AIAA Paper Number 2000-4809.