Car Body Optimization Considering Crashworthiness, NVH and Static Responses

Phani Adduri, Juan P. Leiva, Gary Quinn, Brian C. Watson
Vanderplaats Research and Development, Inc
41700 Gardenbrook Rd Suite 115 Novi MI 48375
padduri@vrand.com

Abstract

This paper demonstrates a design system to efficiently perform optimization based on responses computed from multiple LS-DYNA® analyses while also taking into consideration the linear loading conditions such as the ones for NVH and Static responses. The proposed design system, ESLDYNA, is based on the Equivalent Static Load (ESL) method, which requires the iterative process of non-linear structural analysis (LS-DYNA) and linear structural analysis and optimization (GENESIS). Unlike general purpose optimization software packages, it does not require a large number of analysis calls even for problems with large numbers of design parameters. Therefore, large-scale optimization techniques, such as topology, topometry and topography, can be easily employed. Several examples using different optimization techniques will be presented. One of the examples will include optimizing the design for frontal crash, normal modes and static loading conditions simultaneously.

Introduction

Automobile manufacturers are required to design their vehicles for safety so that the occupants will survive a variety of crash scenarios. Computer simulated crash analysis has evolved over the years to help augment the crash test programs and to give engineers insight into the crash event. These nonlinear simulations have become commonplace during the design phase to save time and cost. Designing an automobile for compliance with these safety standards along with fuel efficiency standards is hard because of some contradiction between these requirements.

Today, to improve the design, analysts and engineers are using analyses and coupling them with general-purpose optimization packages. With these optimization packages, one can design the structure for different crash events along with other NVH and static requirements. Impact simulations typically require a significant amount of computational time and resources. While using a general-purpose optimization package, the computational cost increases exponentially as the optimization might need to evaluate several designs. As a result, there is a growing interest in using approximate models (also called surrogate models) to deal with optimization processes. An approximate model is constructed from a limited set of simulations and used during the optimization to obtain an improved design. But, in practice, there is a limitation on the number of independent design parameters that can be used because of the accuracy of the approximate models.

In contrast, structural optimization programs where the optimization is coupled with linear finite element analysis are being used to solve optimization problems with a large number of design parameters. There are several large-scale optimization techniques, such as Topology [1],

Topometry [2], Topography [3], and Freeform [4] available to efficiently design the structure for linear loading conditions. Even with the large number of design variables, the computational cost involved is not prohibitive. Performing these same large-scale optimization techniques using general-purpose optimization packages is not practical due to the computational cost involved. When nonlinear finite element analysis, like a crash simulation, is used to analyze the structure, then performing large-scale optimization becomes cumbersome and computationally expensive due to the lack of efficient integration between the analysis programs with optimization techniques. Optimization methods using the Equivalent Static Loads (ESLs) [5] have been proposed to efficiently perform optimization based on a nonlinear finite element analysis using linear structural optimization software. The ESLs are defined as the static loads in the linear static analysis that produce the same response field as the nonlinear analysis. A wide variety of nonlinear analysis problems [6-10] have been optimized using the ESL method. A brief introduction of the ESL methodology is presented in the next section.

Review of the ESL Method

Optimization methods using Equivalent Static Loads (ESLs) have been proposed in the literature to efficiently perform optimization based on a nonlinear finite element analysis. ESLs are defined as a set of static loads, 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.

A brief overview of the associated theory is presented here. The governing equation of motion for a transient nonlinear analysis is shown below.

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

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.

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

Each of these ESLs, $f_{ESL}(t_a)$ are applied on the structure as a loading condition in the linear analysis. Multiple loading conditions can be efficiently handled by the linear structural optimization without significant increase in computational cost. The structure is optimized considering all these loading conditions. As the optimization modifies the design, the ESLs are no longer equivalent as 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 the convergence criteria are satisfied. A flowchart illustrating the procedure is given below
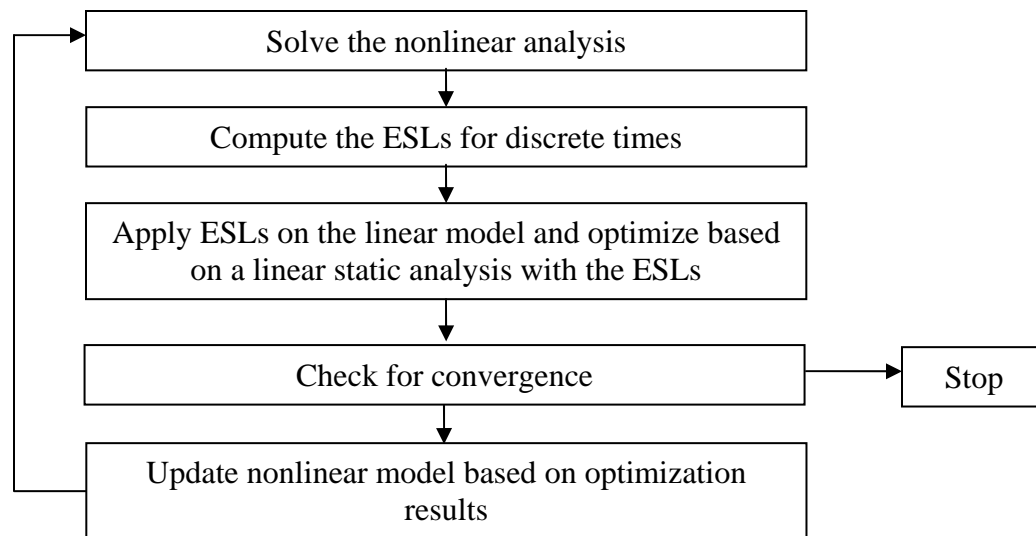
```
                    ┌──────────────────────────────────────┐
              ┌────▶│       Solve the nonlinear analysis     │
              │     └──────────────────────────────────────┘
              │                      │
              │                      ▼
              │     ┌──────────────────────────────────────┐
              │     │    Compute the ESLs for discrete times │
              │     └──────────────────────────────────────┘
              │                      │
              │                      ▼
              │     ┌──────────────────────────────────────┐
              │     │ Apply ESLs on the linear model and optimize based
              │     │   on a linear static analysis with the ESLs │
              │     └──────────────────────────────────────┘
              │                      │
              │                      ▼
              │     ┌──────────────────────────────────────┐        ┌──────┐
              │     │          Check for convergence         │──────▶│ Stop │
              │     └──────────────────────────────────────┘        └──────┘
              │                      │
              │                      ▼
              │     ┌──────────────────────────────────────┐
              └─────│   Update nonlinear model based on optimization
                    │                results                  │
                    └──────────────────────────────────────┘
```

Figure 1: ESL Methodology

ESLDYNA – Implementation of ESL Method 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 into a design package is described.
ESLDYNA [11] is an implementation of the ESL method for coupling a nonlinear LS-DYNA [12] analysis with linear optimization software, GENESIS® [13], for designing the nonlinear model. A preliminary LS-DYNA analysis is done to compute the displacements of LS-DYNA. These displacements are used to compute ESLs. These ESLs are used to perform the optimization on the GENESIS model. Reading the LS-DYNA displacements, computing the ESLs, and optimizing the model are done within GENESIS. The design changes from the optimization are used to update the LS-DYNA model and an analysis is preformed to compute new displacements. This process is repeated until design convergence. In case of a transient nonlinear analysis, the time domain is discretized and applied as multiple load cases on the linear model. The flowchart in Figure 2 illustrates the ESLDYNA process along with the tasks performed by LS-DYNA and GENESIS.
ESLDYNA uses GENESIS as the linear structural optimization package to perform the optimization. GENESIS has the capability of performing several different types of optimization including large scale optimizations like topology, topometry, freeform, etc. The implementation of ESLDYNA provides the capability to use all the different optimization techniques that are available in GENESIS for efficient design.
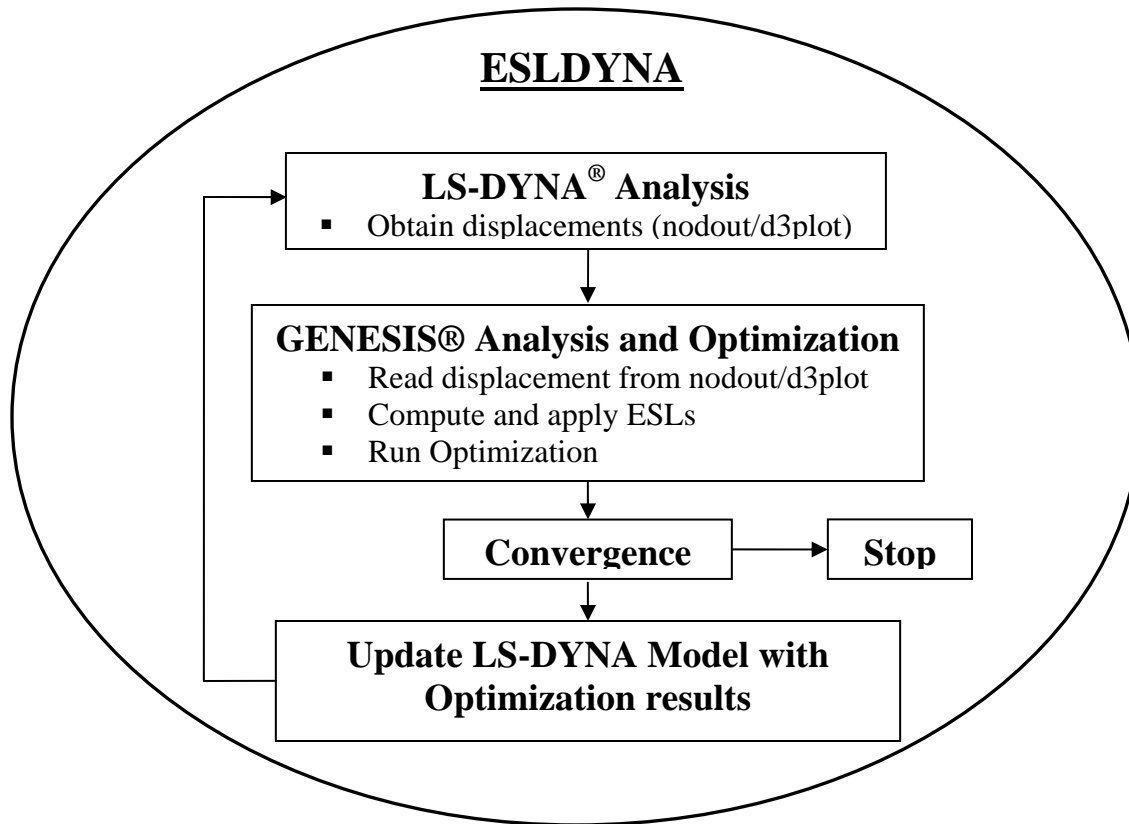
Figure 2: ESLDYNA Process Flow

Two finite element models, one for GENESIS and one for LS-DYNA, are used by ESLDYNA to perform the optimization. GENESIS has the capability to read the displacements from LS-DYNA and convert them into ESLs. These ESLs are used during the optimization. The updated design from GENESIS is used to modify the LS-DYNA model and a nonlinear analysis is performed. Data is seamlessly transferred between the LS-DYNA and the GENESIS models with no programming needed by the user. ESLDYNA controls the entire process without any user intervention. With the inbuilt technology to map the results even if the finite element meshes between the linear and the nonlinear model are different, transfer of data between the models can be accomplished easily.

As mentioned earlier, optimization can be performed with a large number of independent design parameters i.e. large-scale optimization like topology, topometry, topography, freeform etc. Even with these large-scale optimizations, the computational cost involved for the entire optimization would typically be about the time taken to run 10-15 LS-DYNA analyses. ESLDYNA supports the use of multiple LS-DYNA models, with different loading conditions from the nonlinear analysis, to optimize the structure. Design changes that are often not possible using LS-DYNA with standalone optimizers can be performed with ease.

As GENESIS can solve the linear finite element analysis, linear loading conditions can also be analyzed and optimized simultaneously along with the loading conditions from LS-DYNA. So, with ESLDYNA one can design the structure by taking into account the performance from all the crash events along with any static and NVH requirements.

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 are to be described using one or more combinations of linear responses to obtain the same effect.

Examples

Topology Optimization of a Bumper Beam

In this example, topology optimization is performed to design a bumper beam. Two different loading conditions are considered for the LS-DYNA analysis. So in this example, displacements are extracted from two different LS-DYNA simulations and used simultaneously to compute the equivalent static loads and perform the optimization.

A curved bumper beam, as shown in the figure below, is modeled using solid elements. The beam is constrained at the two ends on its back side. In one loading condition, the beam is impacted by a rigid block at the center with initial velocity in y-direction and in the other loading condition it is impacted at an off-center location as shown in the figure below. An elastic piecewise linearly plastic material model (MAT024 from LS-DYNA) is used for the beam while a rigid material model is used for the impactor. The beam is topologically designed to minimize the penetration/intrusion of the rigid block into the beam subject to a mass constraint.
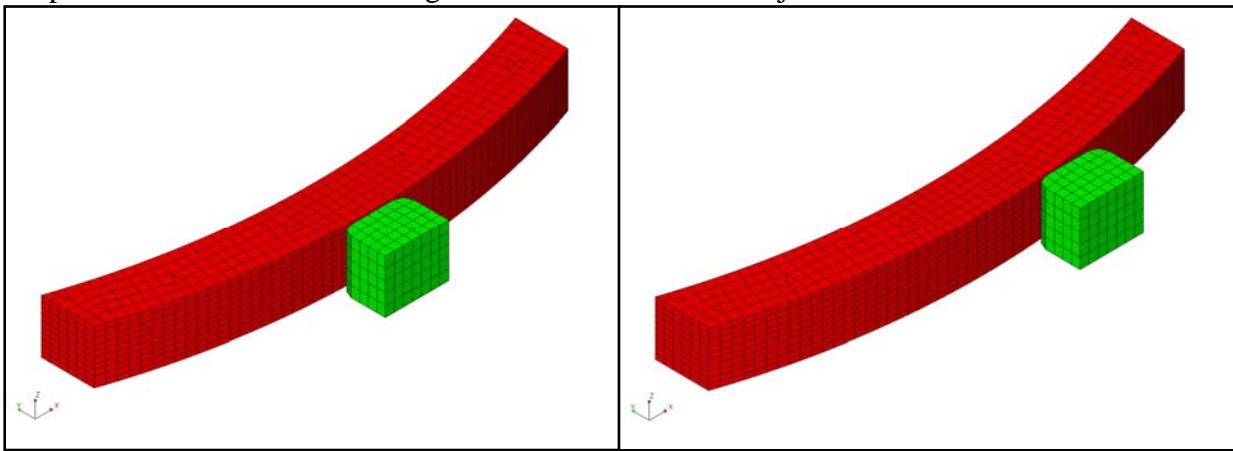

Figure 3: Bumper Beam Model (a) Center loading (b) Off-center Loading

Topology optimization is preformed on the bumper beam to reduce the displacements at the point of contact in both the loading conditions by using only 40% of the initial mass. So the objective for the optimization was to minimize the displacements subject to a mass constraint of only 40% of the initial mass of the beam.

During the optimization, ESLDYNA converges after six iterations which mean that six LS-DYNA simulations were carried out for each loading condition. The displacements actually increase as the optimization is trying to remove 60% of the mass. With the available 40% of the mass, the optimization tries to build the structure so that the displacements are minimized. The figures below show the optimal topology design of the beam.
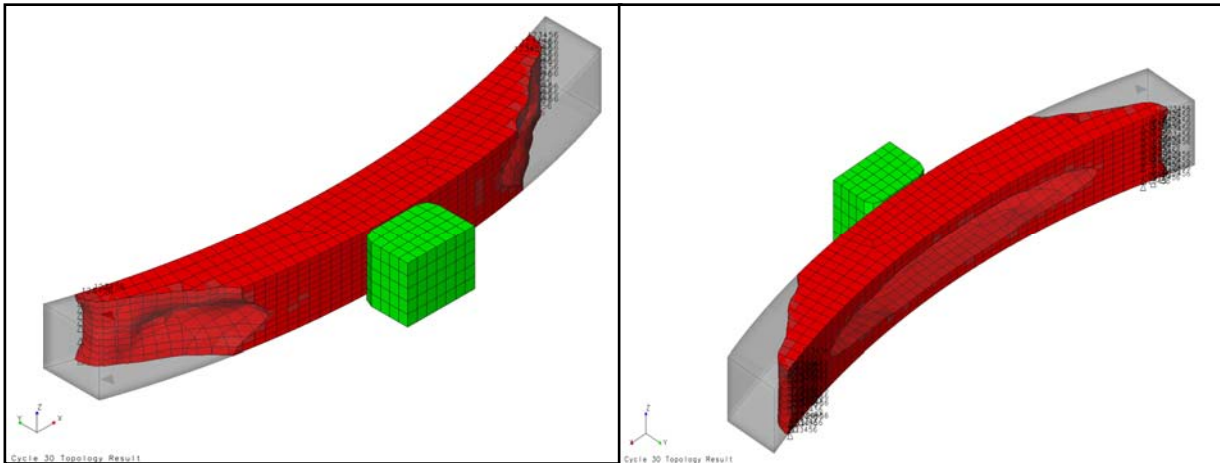
Figure 4: Bumper Beam Optimized Topology

Topometry Optimization for frontal crash, Static Stiffness and Normal Modes
In this example, topometry optimization was performed on a full vehicle model. LS-DYNA analysis was used to simulate a frontal crash event. Along with the frontal crash, the first natural frequency and the static torsional stiffness of the vehicle were also considered during the optimization.

A full vehicle model obtained from NCAC [14], as shown in the figure below, was used for this study. The figure below shows both the GENESIS model as well as the LS-DYNA model.
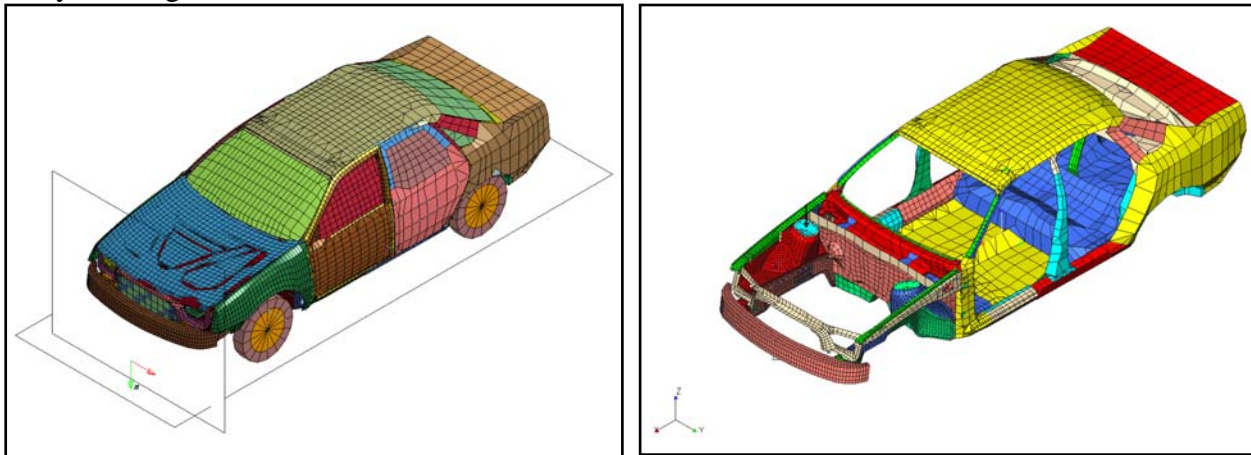

Figure 5: Full Vehicle Model (a) LS-DYNA Model (b) GENESIS Model

For the crash analysis, the passenger compartment intrusion at four locations was considered during the optimization. The objective of the optimization was set to minimize the relative displacements for these four locations so as to reduce the intrusion in the passenger compartment.

Along with this crash loading condition, two other loading conditions were also considered during the optimization. One loading condition was a normal modes calculation to compute the first natural frequency of the structure. Another was a static torsional load for computing the static torsional stiffness. During the optimization, the first natural frequency and the static torsional stiffness were constrained. The first natural frequency was constrained such that the frequency value was at least the same as the initial model. The static torsional stiffness was

constrained to be more than 75% of that of the initial model. Along with these, one more constraint was enforced on the mass of the design to be less than the mass of the initial model.
So the optimization problem was to minimize the intrusion from the frontal crash such that the first natural frequency was not degraded, and also the torsional stiffness was improved without any mass increase. The figures below show the thickness distribution of the initial and the final design. The colors represent the element thickness distribution of the model.
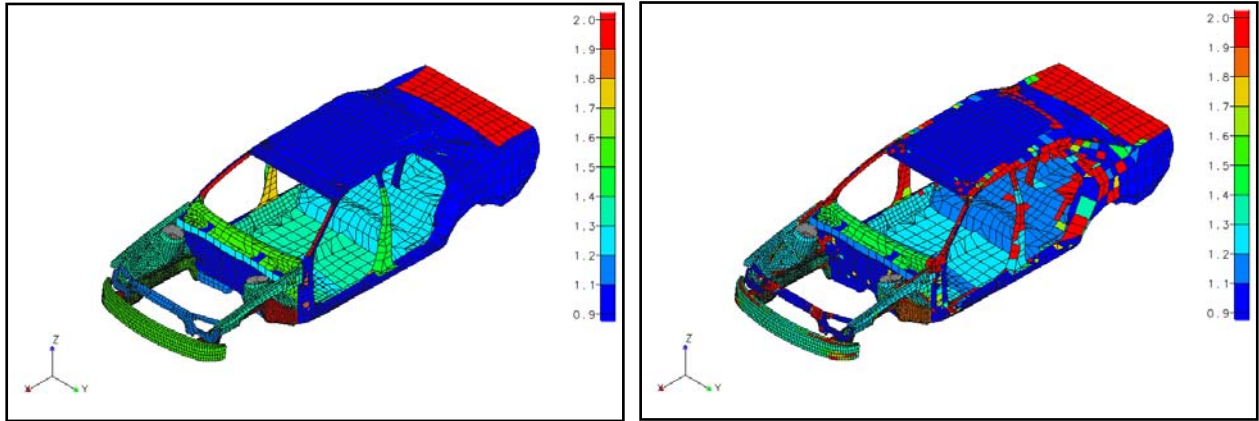

Figure 6: Full Vehicle Optimization Results (a) Initial Thicknesses (b) Optimal Thicknesses

ESLDYNA found the solution in 15 iterations (which means 15 LS-DYNA analyses). The desired 75% increase in the static torsional stiffness was achieved without increasing the mass of the model or degrading the first natural frequency. Also, the intrusion at the four locations was reduced by about 25% of the initial design. The figure below shows a comparison of intrusion values between the initial and optimal designs at two locations.
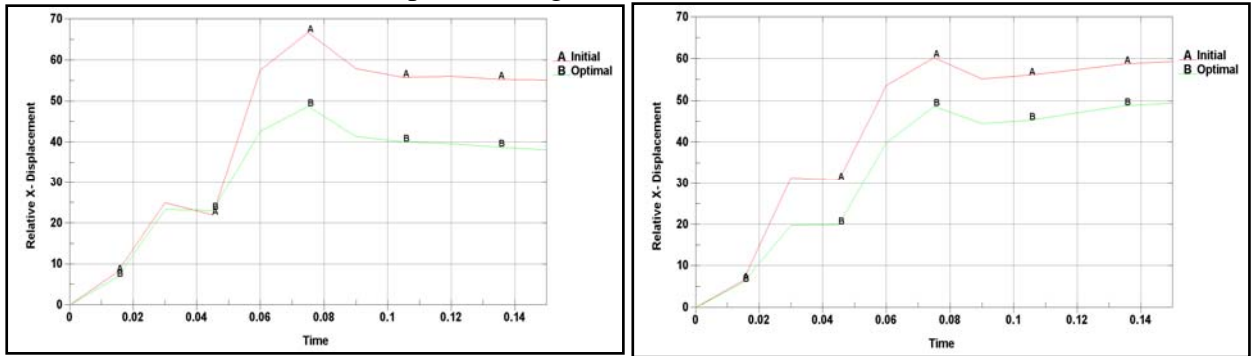

Figure 7: Intrusion Comparison at two locations

The table below shows the elapsed time in each of the programs for all design iterations. Although GENESIS iterates within the linear structural optimization process, runtime is much shorter than the LS-DYNA analysis time.

	LS-DYNA	GENESIS	Total
Elapsed Time (sec.)	50099	2111	52227

Table 1: Time spent during the optimization

Summary

A design package, ESLDYNA, to perform optimization based on both linear and nonlinear loading conditions is presented. ESLDYNA is based on the ESL methodology which tries to do optimization based on a nonlinear analysis, LS-DYNA, using a linear structural optimization package, GENESIS. This method can achieve an optimum design quickly with fewer LS-DYNA simulations and GENESIS design iterations. Total computational cost is almost the same even for a large number of design variables.

Two examples, one with topology optimization and another with topometry optimization are presented in this paper. The first example was used to illustrate that the optimization can be successfully performed for multiple LS-DYNA loading conditions. The second example was used to show that within a given design problem one can use both linear and nonlinear loading conditions simultaneously and perform optimization of both these types of analyses. Even though only topology and topometry optimization problems are shown in the examples, all the different types of optimization that are available in GENESIS have been implemented and successfully tested.

References

[1] JP Leiva, BC Watson and I Kosaka, "Modern Structural Optimization Concepts Applied to Topology Optimization," 40th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Material Conference. St. Louis, MO, pp. 1589-1596, Apr. 12-15, 1999

[2] JP Leiva, "Topometry Optimization: A New Capability to Perform Element by Element Sizing Optimization of Structures," 10th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Albany, NY, Aug. 30 - Sep. 1, 2004

[3] JP Leiva, "Method for Generating Perturbation Vectors for Topography Optimization of Structure," 5th World Congress of Structural and Multidisciplinary Optimization, Venice, Italy, May 19-23, 2003

[4] JP Leiva, "Freeform Optimization: A New Capability to Perform Grid by Grid Shape Optimization of Structures," 6th China-Japan-Korea Joint Symposium on Optimization of Structural and Mechanical Systems, Kyoto, Japan, June 22-25, 2010.

[5] GJ Park, "Technical Overview of the Equivalent Static Loads Method for Non-linear Static Response Structural Optimization," Structural and Multidisciplinary Optimization, March 2011, Vol. 43, Issue 3, pp. 319-337

[6] NS Green, RA Canfield, E Swenson, "Structural Optimization of Joined-Wing Beam Model with Bend/Twist Coupling using ESL," 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Material Conference, Palm Springs, CA, AIAA-2009-2644, May 4-7, 2009.

[7] BS Kang, GJ Park, JS Arora, "Optimization of Flexible Multibody Dynamic Systems Using the Equivalent Static Load Method," AIAA Journal, April 2005, Volume 43, No. 4, pp. 846-852

[8] SI Yi, JY Lee, GJ Park, "Crashworthiness design optimization using equivalent static loads," Journal of Automobile Engineering, January 2012, Volume 226, pp. 23-38

[9] WS Choi, GJ Park, "Structural Optimization using Equivalent Static Loads at all Time Interval," Comput. Methods Appl. Mech. Engr, 191 (2002), 2077-2094

[10] I Kosaka, "Improvement of Energy Absorption for the Side Member Using Topography Optimization," 11th International LS-DYNA Users Conference, Detroit, MI, June 6-8, 2010.

[11] ESLDYNA USERS MANUAL, Version 12.1, Vanderplaats Research and Development Inc, September 2012

[12] LS-DYNA KEYWORD USER'S MANUAL VOLUME I, Version 971, Livermore Software Technology Corporation, May 2007

[13] GENESIS USERS MANUAL, Version 12.2, Vanderplaats Research and Development Inc, September 2013

[14] http://www.ncac.gwu.edu/vml/models.html