

Large Scale Structural Optimization using GENESIS®, ANSYS® and the Equivalent Static Load Method

Hong Dong

Vanderplaats Research & Development, Inc.
47100 Gardenbrook, Suite 115
Novi, MI 48375, USA

Juan Pablo Leiva

Vanderplaats Research & Development, Inc.
47100 Gardenbrook, Suite 115
Novi, MI 48375, USA

Phani Adduri

Vanderplaats Research & Development, Inc.
47100 Gardenbrook, Suite 115
Novi, MI 48375, USA

Takashi Miki

DENSO Corporation
1-1, Showacho, Kariya-shi, Aichi, 448-8661, Japan

Tomomitsu Fukuoka

DENSO Corporation
1-1, Showacho, Kariya-shi, Aichi, 448-8661, Japan

This paper describes GSAM (GENESIS Structural Optimization for ANSYS Mechanical) which is a computer application that integrates the GENESIS® structural optimization software with the ANSYS® analysis software. One capability of GSAM is to allow its users to optimize structures in which their nonlinear responses are calculated by ANSYS®. GSAM utilizes GENESIS's six types of optimizations which are: sizing, shape, topology, topometry, topography, and freeform. GSAM can utilize the Equivalent Static Load method. A key advantage of this method is that it only requires a small number of analysis calls. This advantage enables its users to optimize very large scale nonlinear problems in a reasonable amount of time. Three problems will be presented here to demonstrate this tool. The first example will highlight the ability of performing topology optimization with nonlinear transient analysis. The second example will show topology optimization for reducing plastic residual deformation. The third and last example will demonstrate the optimization of a structure with nonlinear material using topography optimization.

Keywords: sizing, shape, topology, topometry, topography, freeform, optimization, equivalent static load method

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 next section.

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 extensively in literatures 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:

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

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 loadcase. 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.

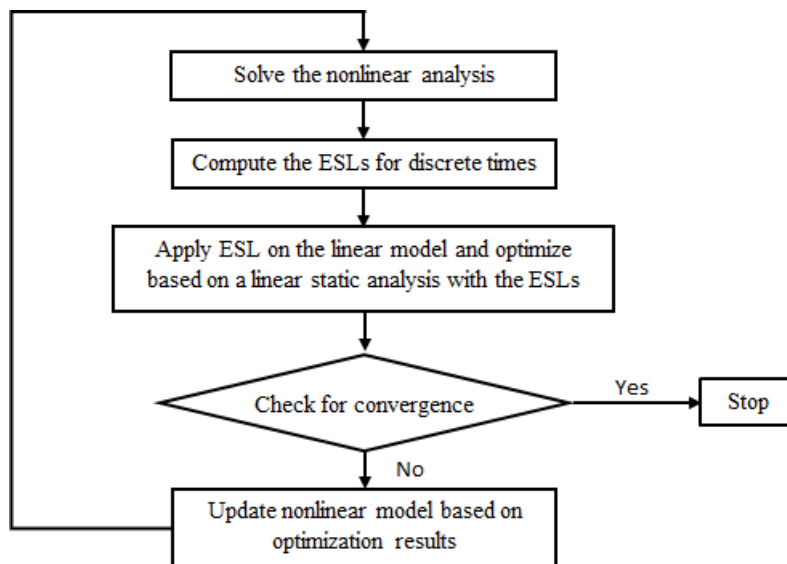


Figure 1. Flowchart for optimization based on ESL Method

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 a predefined convergence criteria is satisfied. A flowchart illustrating the procedure is given in Figure 1.

ESL Implementation in GSAM for ANSYS

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 an ANSYS analysis is described.

ESL for ANSYS in GSAM [7] is an implementation of the ESL methodology to perform optimization based on an ANSYS [8] analysis. GENESIS [9] is used as the linear structural optimization package to perform the optimization. GENESIS itself uses the BGIDOT [10] optimizer developed by Vanderplaats [11]. The ESL for ANSYS implementation provides the capability to use structural optimization techniques in GENESIS for efficient design. The flowchart of ESL for ANSYS is shown in Figure 2:

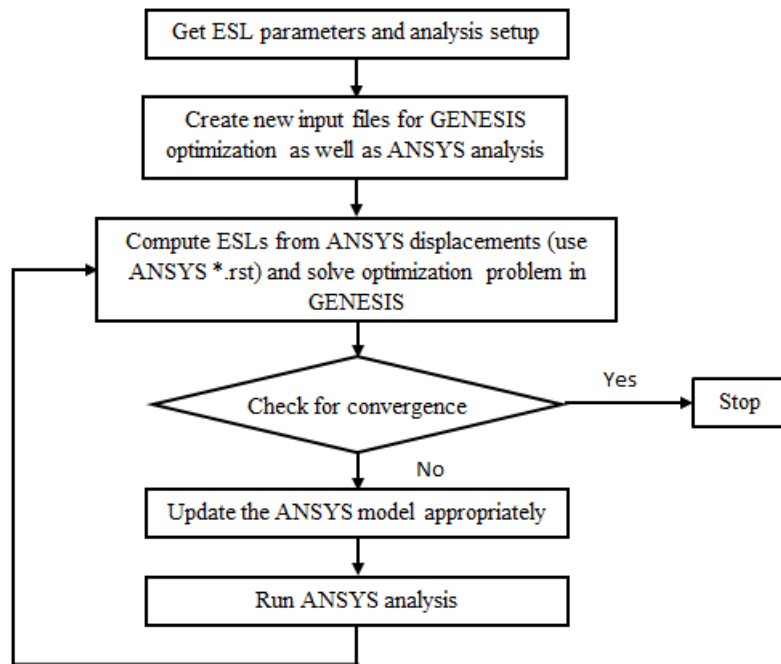


Figure 2. Flowchart of ESL for ANSYS

As a first step, the user will need to specify which ANSYS analysis systems are to be designed with the ESL method. The analysis systems that can be designed with ESL method include: Static and Transient. Other type of analysis systems (Modal, Harmonic, Random Vibrations, and Buckling) will be designed based on the GENESIS FEA. The analysis systems to be designed based on ESL for ANSYS and the analysis systems to be designed based on GENESIS FEA can be mixed in one optimization.

ESL for ANSYS will create new input files for GENESIS based on user defined ESL parameters and analysis setup. The ANSYS input file is also created, which will be updated based on the initial design at the first cycle and will be updated based on the optimization results after one GENESIS optimization run.

The newly created GENESIS input files are used to start the first GENESIS optimization run. With ESL for ANSYS, GENESIS can read the displacements from ANSYS *.rst file. Each set of

displacements is treated as one load case in GENESIS for the analysis. GENESIS will convert them to ESLs and analyze the structure.

Based on the defined design data for GENESIS, optimization is performed and the ANSYS model is changed to reflect the optimization process. Once the optimization is done, the ANSYS model is updated with the results from the GENESIS optimization. After updating the model, ANSYS analysis is carried out to analyze for a new set of displacements. The new set of displacements (from ANSYS *.rst file) 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 ESL for ANSYS. The computational cost for the entire optimization typically would take about 5 to 10 ANSYS analyses.

ESL for ANSYS 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 ANSYS 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.

Examples

Topology optimization of a bumper beam

In this example, topology optimization is performed to design a bumper beam. The loading condition is shown in Figure 3. The beam is constrained at the ends on its backside by two rigid cylinders. The beam is impacted by another rigid cylinder at the center with an initial velocity of 5000 mm/s in Y direction. Topology optimization is performed to obtain a stiff structure with a given mass requirement. The objective for the optimization is to minimize the strain energy of the beam subject to a mass constraint of 30% of the initial mass of the beam.

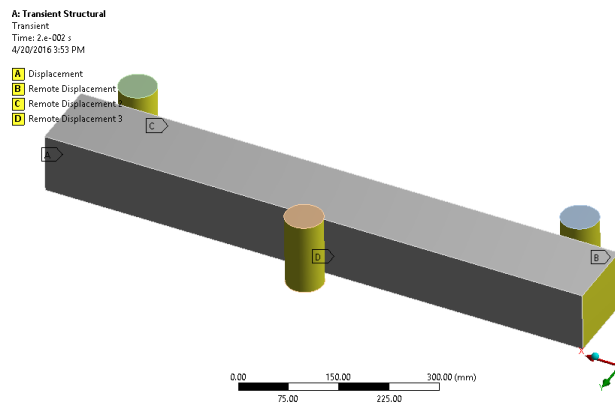


Figure 3. Loading conditions for the bumper beam

There is a total of 7 load cases in the GENESIS model which corresponds to 7 time steps in the ANSYS transient analysis model. The time steps are: 0.001, 0.0025, 0.0075, 0.01, 0.0125, 0.0175, and 0.2. The optimization converges in 9 ESL cycles which correspond to 10 ANSYS simulations. Figure 4 shows the topology optimization result for the beam. The maximum deformation is reduced from 92.49 mm to 41.99 mm as shown in Figure 5 and Figure 6.

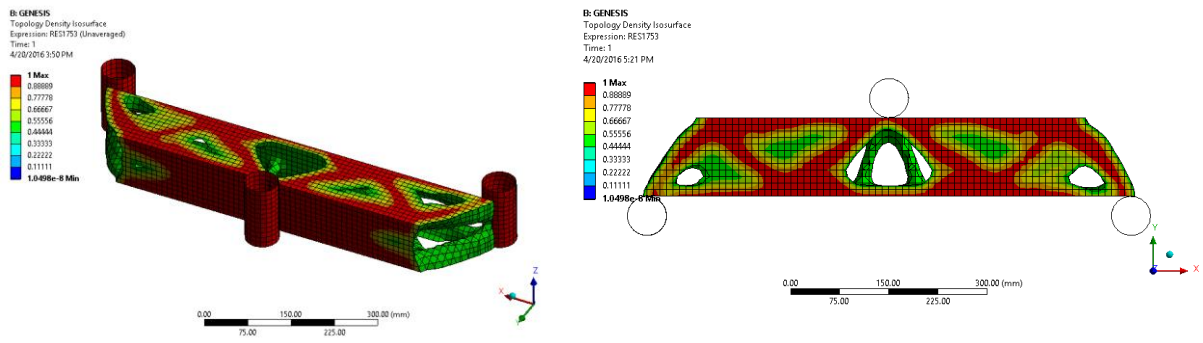


Figure 4. Topology density isosurface result for the bumper beam

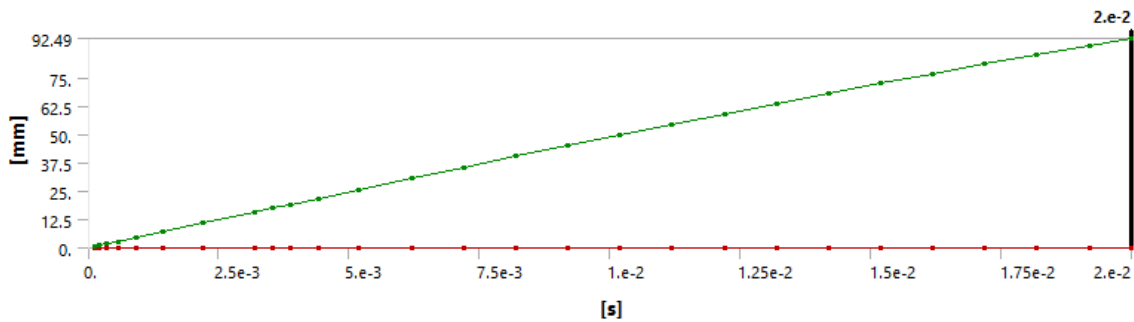


Figure 5. Displacement for initial design (with uniform density value of 0.3)

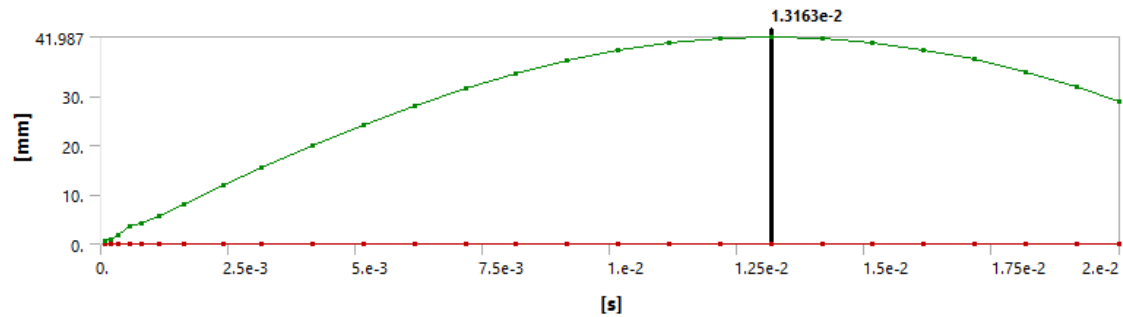


Figure 6. Displacement for optimal design

Table 1 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 ANSYS analysis time.

	ANSYS	GENESIS	Total
Elapse time (secs)	14,937	825	15,222

Table 1. Elapse time

Topology optimization of a pressured rail (courtesy of DENSO CORP.)

In this example, topology optimization is performed to design a pressured rail. The loading condition is shown in Figure 7. The rail is fixed at one end surface. A pressure is applied on the inner channels. There are three load steps for the pressure load as shown in Figure 8. The rail is modeled with a multi-linear plastic material. The stress-strain curve is shown in Figure 9.

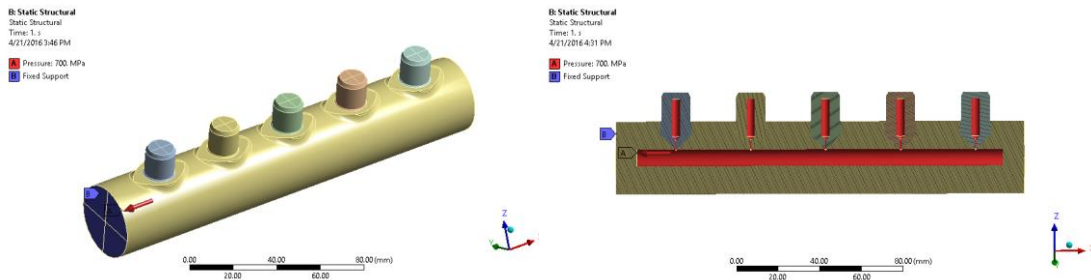


Figure 7. Loading conditions for the pressured rail

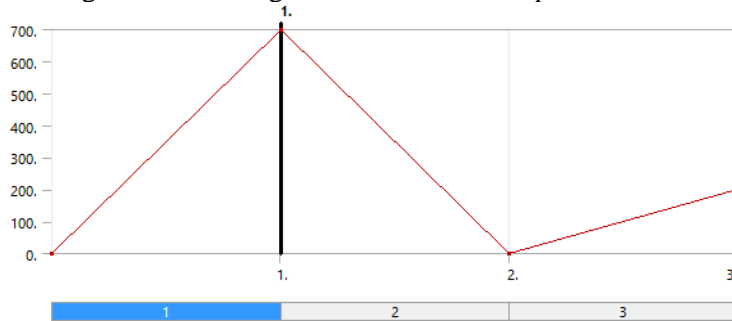


Figure 8. Pressure loading

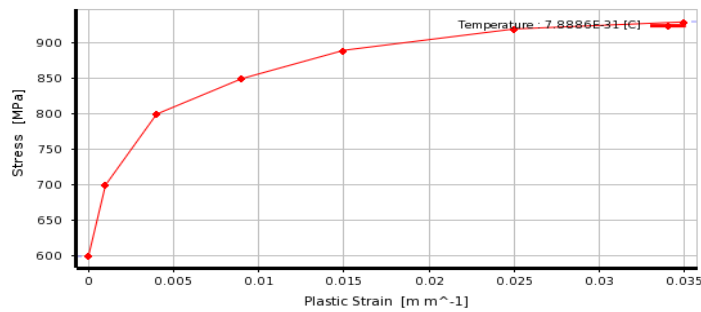


Figure 9. Stress-strain curve of the multi-linear material

The main design requirement is that the residual deformation at load step 2 (unloading step) is less than the given values. The details are shown in Figure 10. Another requirement is that the stresses at load step 3 are to be below a certain value.

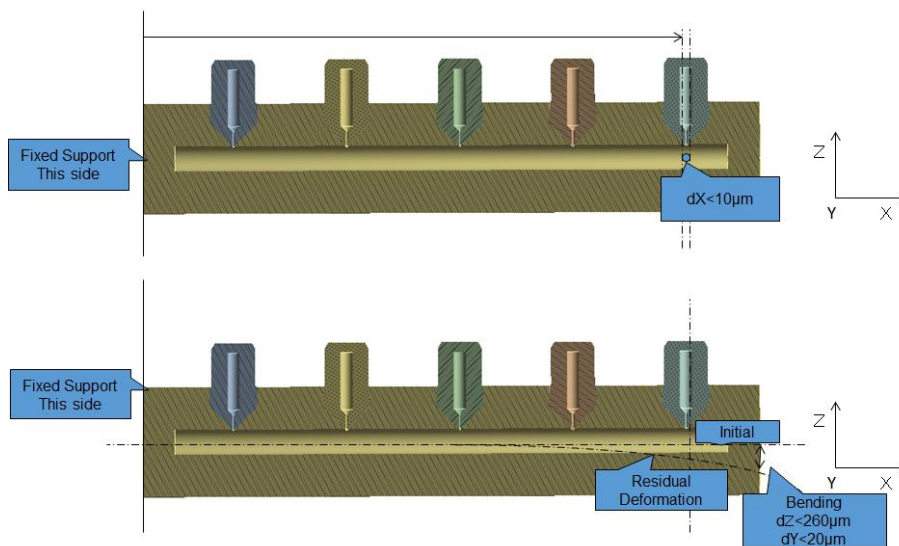


Figure 10. Displacement requirement

Topology optimization is performed to satisfy the design requirements with a given mass. The objective for the optimization is to minimize strain energy of the rail subject to a mass constraint of 80% of initial mass and displacement requirement as described above. The optimization finished with 5 ESL cycles, i.e. 6 ANSYS simulations, with the displacement requirement satisfied. Stress value for load step 3 is recomputed with the optimized model, which is very close to the requirement. The topology design result is shown in Figure 11.

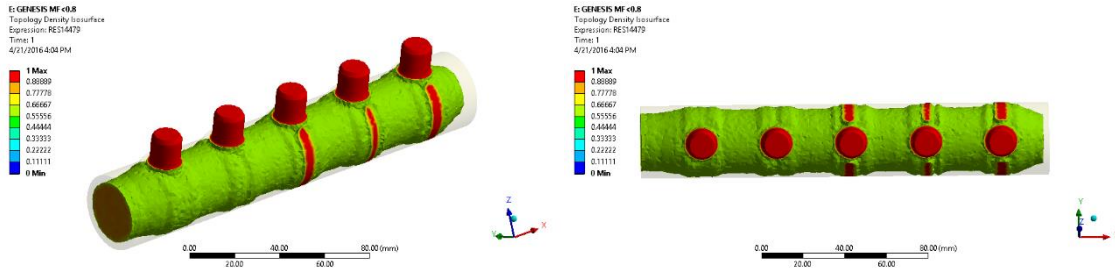


Figure 11. Topology density isosurface result for the pressured rail

Topography optimization of an oil pan

In this example, topography optimization is performed to design an oil pan. The oil pan is fixed at each of the screw locations. The loading condition is shown in Figure 12. A pressure load is applied on the bottom surface of the oil pan. A bilinear plastic material is used for the oil pan. The stress-strain curve for this material is shown in Figure 13.

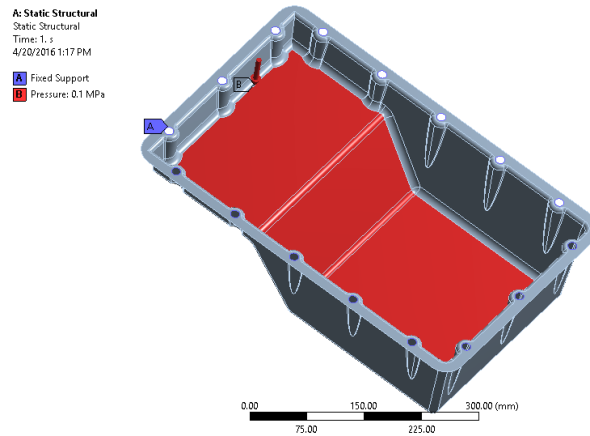


Figure 12. Loading condition for the oil pan

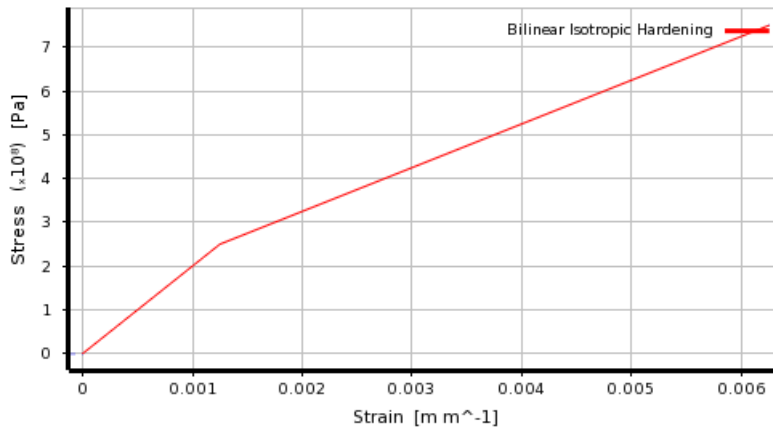


Figure 13. Stress-strain curve of the bilinear material

Topography optimization is performed to make the oil pan stiffer by creating beads on the surfaces. The objective for the optimization is to minimize the maximum displacement of the grids on the surfaces. The allowable perturbation magnitude is 5 mm. In addition, the grids are only allowed to move along the positive Z direction. The topography region and the optimization result is shown in Figure 14. The optimization finished with 5 ESL cycles, i.e. 6 ANSYS simulations. The maximum deformation is reduced from 3.44 mm to 2.38 mm (Figure 15).

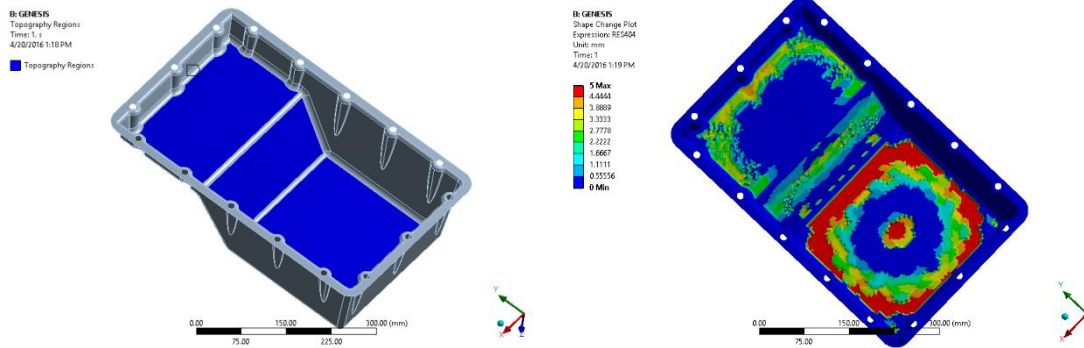


Figure 14. Topography design: (a) design region, (b) shape change result

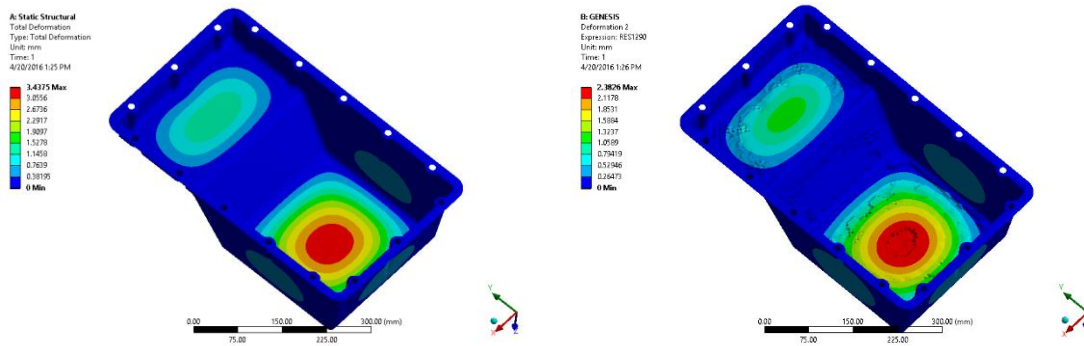


Figure 15. Displacements: (a) initial design, (b) optimal design

Summary

A design package, GSAM that allows its users to perform optimization based on both linear and nonlinear loading conditions has been presented. GSAM utilizes the ESL method to couple ANSYS nonlinear structural analysis with the GENESIS structural optimization software. This implementation typically achieves an optimum quickly with few ANSYS simulations. Total computational cost is low even for a large number of design variables.

Three examples, two with topology optimization and another with topography optimization, are presented in this paper. The first example demonstrates that topology optimization can be successfully performed for nonlinear transient analysis with large deformations. The second example shows topology design for reducing plastic residual deformation. The last example illustrates topography optimization for an oil pan with nonlinear material properties. Even though only topology and topography optimization problems are shown in the examples, other types of optimization such as: sizing, topometry, freeform, and shape optimization have been implemented and successfully tested.

References

- [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 15.0, 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] GENESIS Structural Optimization for ANSYS Mechanical User's Manual, Version 15.0, Vanderplaats R&D, Inc.
- [8] ANSYS Mechanical User's Guide, Version 17.0, ANSYS, Inc.
- [9] GENESIS User's Manual, Version 15.0, Vanderplaats R&D, Inc.
- [10] BIGDOT User's Manual, Version 2.0, Vanderplaats R&D, Inc.
- [11] 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.