THE USE OF TOPOLOGY OPTIMIZATION IN ENHANCING THE STRUCTURAL PROPERTY
OF AN AUTOMOTIVE FRONT SUB-FRAME

Mr. Da'Quan L Prince-Floyd, Prairie View A&M University

I am DaQuan Prince-Floyd, graduating graduate student in the Mechanical Engineering program at Prairie View A&M University. I have a Bachelors of Science in Mathematics from Huston Tillotson University. Hard work, consistency, and expressed individuality have helped me persevere to obtain my goals. Thank you for allowing me the chance to share my research with you.

Dr. Jianren Zhou, Prairie View A&M University

Professor, Department of Mechanical Engineering, Prairie View A&M University.

Dr. Jaejong Park, Prairie View A&M University

Structural Optimization of an Automotive Front Sub-frame

DaQuan Prince-Floyd, **Jaejong Park, Jianren Zhou**
Department of Mechanical Engineering
Prairie View A&M University
Prairie View, TX 77446

Abstract

The frontal impact is the most common in automotive collision accidents, and bending the sub-frame can directly lead to severe passenger injury and property damage. This research analyzed the crashworthiness, design, mechanical integrity, and optimization of an automotive front sub-frame structure. From the original geometry, a new sub-frame with similar mass and mounting locations is designed. Loads were applied to the front side members of the sub-frame to simulate a common frontal and partial frontal crash. A sub-frame with enhanced structural efficiency was designed using topology optimization. This improvement may preserve the lifespan of the sub-frame, reinforce the protection of passengers and the engine, and improve crashworthiness. Topology optimization is a numerical analysis technique that allows engineers to distribute materials optimally for a specific cost function. Iterative update of design variables typically relies on sensitivity information from performance analysis in each step. A simple parametric study on material candidates and design constraints was executed to evaluate various design options. Sub-frames with optimized geometries were mechanically tested against two different simplified loads mimicking frontal crashes. The dynamic behaviors were also analyzed and compared to the original design for validation.

Introduction

A sub-frame plays a significant role in the protection of automotive inner components and by using a strengthened material, the overall crashworthiness improves [1]. The sub-frame is a structural component attached to the main structure to support key components such as the engine, transmission, and suspension. The fundamental functions are to distribute high chassis loads among a lightweight monocoque body while also isolating noise, vibrations, and harshness produced from other components. Yet, the most important factor considered while manufacturing these parts is durability or crashworthiness [2]. The passenger's safety is of utmost importance in the event of a crash, and crashworthiness measures the structural performance of automotive structures under dynamic loads. The frontal impact is the most common in automotive collision accidents; a general explanation of structural performance is the ability of the structure to absorb the impact energy by dissipating the kinetic energy of the strain energy or internal energy [1, 3].

Optimization techniques required for crash-worthy analysis have a highly dynamic nature. The Equivalent Static Load Method (ESL) estimates dynamic responses at specific time steps using multiple static loads such that the same displacement field is achieved. The Equivalent Static Loads Method for Non-Linear Static Response Structural Optimization (ESLSO) was developed to solve nonlinear static response optimization problems [4]. Linear structural optimization techniques are conducted in correlation to the data acquired from the nonlinear analysis. ESL's have provided accurate results in areas such as nonlinear static response optimization, nonlinear dynamic response

optimization, linear dynamic response optimization, structural optimization for multi-body dynamic systems, and more [5].

Structural Optimization

Topology optimization distributes the limited amount of material within the design domain resulting in better structural efficiency against a set of design and performance constraints [6]. This study employed common Solid Isotropic Material with Penalization (SIMP) technique [7]. With SIMP, each structural element within the discretized design domain can have any material density value between 0 (void) and 1, and its stiffness is computed as follows,

$$K = (\rho_e)^p K_0 \; ; \; p > 1 \tag{1}$$

where K is the element stiffness, ρ_e is the element density, p is the penalization factor which usually is three or larger, and K_0 is the stiffness of the material. The higher the penalty factor (p), the more aggressive elemental density values are driven to 0 or 1. The sensitivity analysis becomes simpler for traditional compliance minimization since the objective function shown below becomes differentiable.

$$\min_{\rho_e} C = U^T K U = \sum_{e=1}^{n} U_e \, (\rho_e) \, U_e = \sum_{e=1}^{n} (\rho_e)^p \, U_e \, K_0 \, U_e \tag{2}$$

$$Subject \, to: V = f \, (V_0) = \sum_{e=1}^{n} \rho_e V_e \le V^*$$

$$F = KU$$

$$0 < \rho_{min} \le \rho_e \le 1$$

where C is the structural compliance, U is the displacement, U_e is the elemental displacement, V is the volume, V^* is the volumetric constraint, F is the forces applied. ρ_{min} is a small value to avoid any singularities. The readers are directed to [6] for more details on various topology optimization formulations.

Testing and Analysis

The 2014 Honda Accord sub-frame was made from aluminum for the front members (black) and structural steel for the rear member (silver). Using explicit dynamics in ANSYS, a crash analysis of the structure was performed. The force and stress applied to the side members were evaluated for the equivalent static structural analysis. The forces from the dynamic analysis were then applied to the side member using static structural in ANSYS. For the first static analysis, the original material was used. The stresses of the system should be equivalent to prove the equivalent static method is valid. The material of the side members was then converted to structural steel in an attempt to achieve stronger mechanical characteristics while maintaining overall weight, and a thicker sub-frame was designed and used as an initial starting point with aluminum. The stresses from both analyses were different due to the change in material. The side member was then be optimized with a mass constraint so that it matches that of the original structure. The same dynamic analysis was completed for comparison with the original structure.

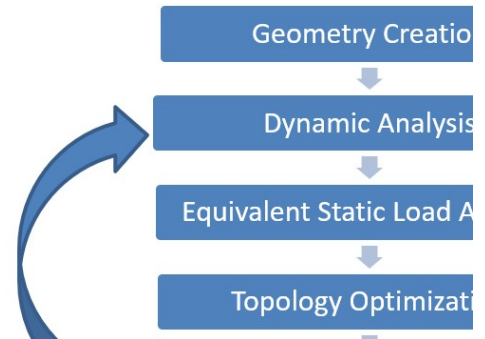The original geometry is obtained from the NHTSA, used in

Figure 1. The workflow used in this approach.

the Structural Countermeasure Research Program [7, 8]. NX is used to create a solid non-optimized model of the geometry, and the features to the model are added in ANSYS spaceclaim. The orange part shown in Fig. 2 is a wall to simulate a frontal impact and side impact.
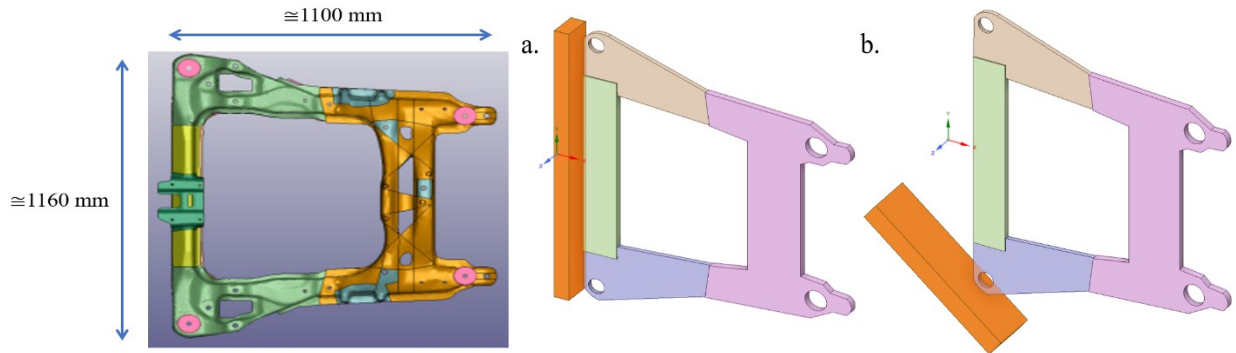

Figure 2. Left: sub-frame K file from NHTSA; a) and b): solid model subject to wall for frontal and side impact simulation.

Nonlinear dynamic analysis was performed using ANSYS explicit dynamics. The material of the front members was aluminum as in the original sub-frame. The original 15mm aluminum side member sub-frame was used to analyze the forces and stress after impact. Both frontal and side impacts were analyzed at the same initial velocity of -26.822m/s or 60mph in the x-direction. In the analysis setting, the end time of .008s was used. The forces of the impact were obtained at the time step when the sub-frame experienced the highest stress during initial contact, which resulted in values tabulated below.

Table 1. Reaction forces at the maximum stress

	Time step (s)	Average stress (Pa)	Contact reaction forces (N)		
			x	y	z
Frontal impact	1.2e-3	2.611e8	-2.9e5	1.1e5	-7.4e3
Side impact	2e-3	1.1e8	1.7e4	5.8e4	-2.0e3

The external forces were applied to the same geometry using ANSYS Static Structural to conduct the Equivalent Static analysis. These force values were enacting on the weight of the sub-frame only and not the weight of an entire automobile. The stress of the equivalent static simulation should be identical or in the range to validate the equivalent static method. The equivalent static structural method was conducted using the forces obtained from the explicit dynamic analysis using the original 15mm side member sub-frame. The forces obtained from the frontal and side-impact are applied in the x, y, and z in those same directions on the side member. These forces imitated the dynamic loads as the static loads to the model. Loads and boundary conditions were applied to the model. Fixed supports were used to hold to the regions outside of the side member. The forces calculated were obtained explicitly for the side member, and fixed support isolated those forces from the remainder of the system.

The stresses obtained from the static response should equal or in the range of the stress from the dynamic response. As shown in the static results, the stress values and contour for both frontal and side impacts are close. The maximum stress applied to this system was 6.0807e7 Pa. The stress calculated was between the stress values from the dynamic analysis 1.8905e6 Pa to 4.0523e9 Pa. The equivalent static load method for the side impact is validated.

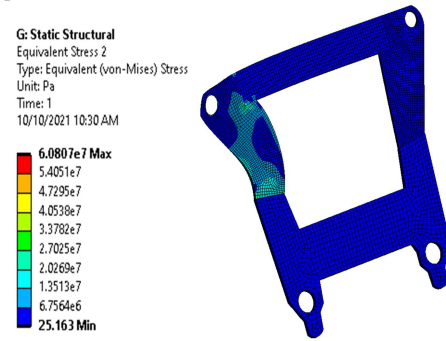- Frontal Impact Results
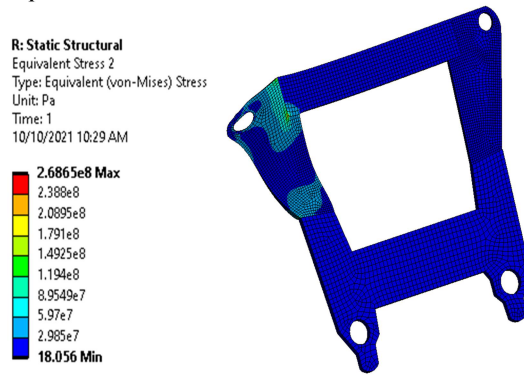
- Side Impact Results

Figure 3. Equivalent static structural analysis results. Left: frontal impact, Right: Side impact.

Design optimization and computational validation

The side member of the sub-frame was selected to be optimized due to its largest deformation. For more design options in optimization, two different thickness options were used (e.g., 10 mm and 20 mm). The material of the 10mm sub-frame and original 15mm sub-frame is changed from aluminum to structural steel. Appropriate mass constraints were employed to make the final weight similar to each other, as shown in table 2.

Table 2. Design parameters for topology optimization

Models	Original	1	2	3
Thickness (mm)	15	15	10	20
Material	Aluminum	Structural steel	Structural steel	Aluminum
Mass (kg)	2.77	7.87	5.23	3.69
Mass % constraint	N/A	35	50	75
Final mass (kg)	2.77	1.84	2.21	2.73

The same mesh, forces, and boundary conditions were applied to the three alternate sub-frame models. The termination was controlled by either density change less than 1% or iteration count of 100. The side member was the only portion that was set as a design region within ANSYS Topology Optimization, as shown in Fig. 4. The equivalent static loads are utilized throughout the optimization of all three models. To retain welding spots and easier fabrication, the hole in the side member is retained, and appropriate manufacturing constraint is used. A penalization factor of 5 is used during the optimization process for a better distribution of material.
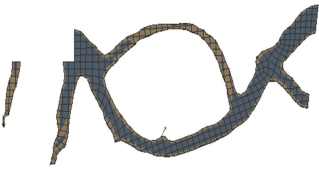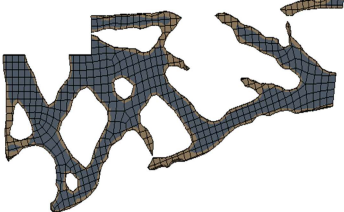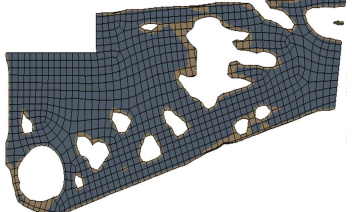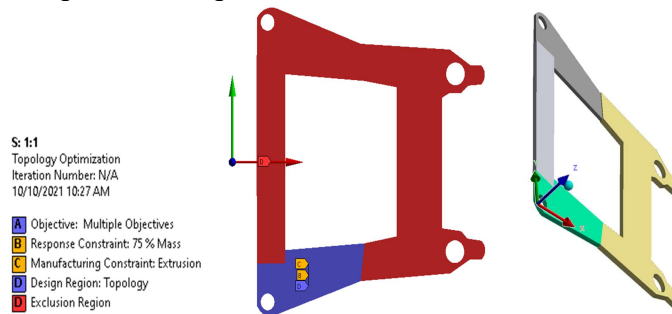

Figure 4. The side member (blue) is the design domain. The equivalent static loads are applied using coordinate system shown on right.

Table 3. Topology optimization results from all three models.

Case	15 mm structural steel (35%)	10 mm structural steel (50%)	20 mm aluminum (75%)
Mass	1.84 kg	2.21 kg	2.73 kg

The 15 mm structural steel sub-frame did not meet design criteria. The distribution of the material in ordinance with the mass constraint did not provide a sustainable structure. This structure could not be reattached to the original assembly. The 10 mm structural steel resulted in a fully connected geometry, but only one member connects the whole geometry in the middle. The optimization parameters such as penalization factor and filter sizes were controlled, resulting in an unnoticeable difference in the final geometry. The 20mm aluminum sub-frame was the only variable design to meet the mass criteria to the acceptable accuracy (~1.3%). The original mass of the structure was calculated at 3.69 kg, and the mass of the optimized region was 2.73 kg. The design of this sub-frame had the capability to be assembled to original parts of the sub-frame and meet design constraints.

The 20 mm optimized aluminum sub-frame was post-processed in CAD software, as shown in Fig 5. While this might affect the optimality of the topology optimized geometry, post-processing was inevitable to minimize the potential manufacturing complexity and stress concentration. A similar explicit dynamic simulation was done on both the original subframe and one with a 20 mm topology optimized side member, which resulted in the stress contour shown in Fig 6.
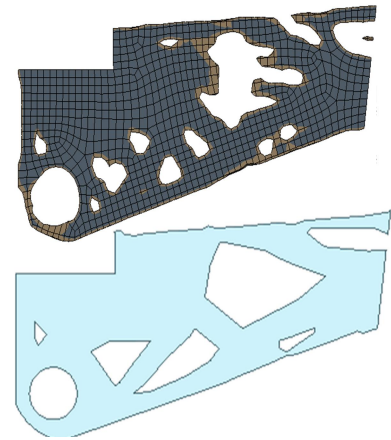
Figure 5. Topology optimized geometry was postprocessed using manual intervention.
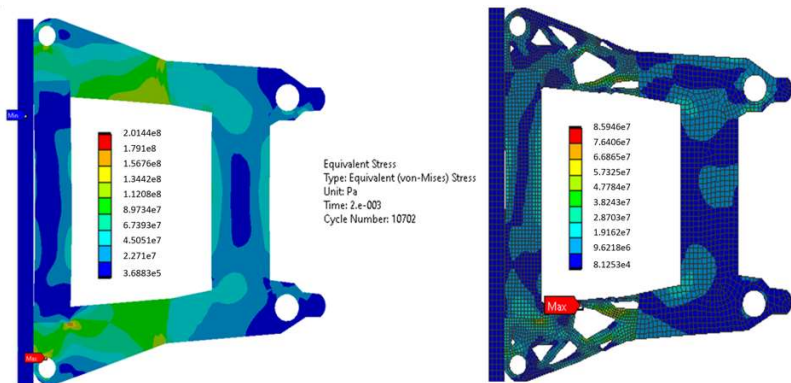
Figure 6. Sub-frame with 20 mm optimized side member resulted in a more than 50% in maximum stress compared to original 15 mm side member.

The stresses from the optimized structure were lower than that of the original structure. The maximum stress was measured at 43% compared to the maximum stress of the original sub-frame subjected to

the same load. The materials in optimized model were better utilized as the areas with negligible stresses (blue) were considerably less especially at the front of the vehicle. This shows that the sub-frame with optimized side members might react better upon impact.

Mechanical Experiment Validation

To further evaluate the structural behavior of the optimized design, experiments were performed. The sub-frame geometries were 3D printed and mechanically tested to quantify their limits. For simplicity, only static behaviors were examined using experiment models fabricated via a common desktop 3D printed with Fused Deposition Modeling (FDM) and ABS in a smaller scale (see Fig 7.1). Gradually increasing load from displacement controlled Instron 5582 was applied similar to loads that would be applied in a frontal impact utilizing customized fixtures.
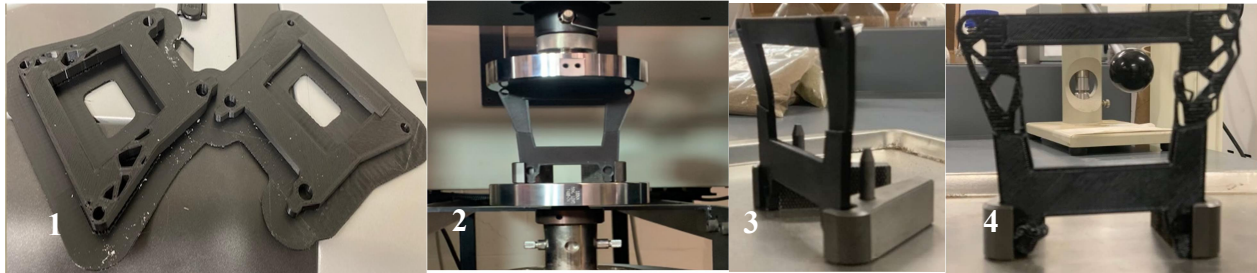

Figure 7. 1) 3D printed ABS experiment models, 2) Mechanical experiment, 3) and 4) Load was gradually increased until fracture (controlled by displacement rate).

The values obtained were recorded from the Instron Bluehill Universal. Once the structure peaked at .40 kN, the forces dropped immediately. Fig. 7 shows the sub-frame bent at the joint of the side and rear members. The optimized structure underwent the same test, and a maximum force of 0.98 kN was recorded. A thinner structural member within the side member buckled at failure. While this mechanical experiment does not fully represent the highly dynamic nature of the crash, the trend observed was consistent with that from computational validation.
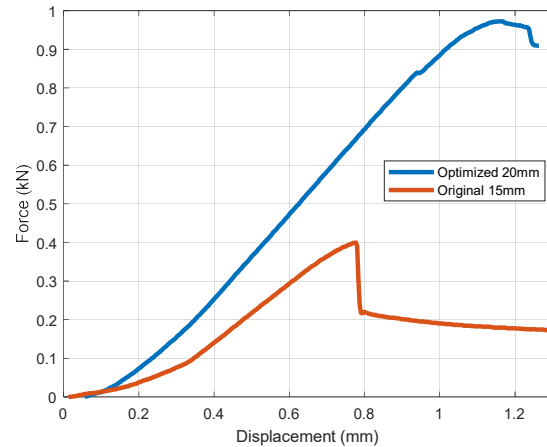

Figure 8. Force vs. displacement measured for both 15 mm sub-frame with original geometry and 20 mm sub-frame with optimized geometry.

Conclusions

In this study, the efficacy of basic topology optimization formulation using ANSYS was studied to characterize the performance of front sub-frame structure subject to complex loading conditions. Equivalent static loads were computed from the explicit dynamic analysis and used as static loads in the topology optimization process. It was discovered that the structural performance of side members of the sub-frame is improved while maintaining its weight efficiency. While not all design parameter combinations yielded feasible results, the optimum design was created from the

thicker side member. Numerical validation study after manual post-processing showed that the materials were better utilized with less non-zero stress areas and lower maximum stress. Much simplified mechanical testing with conventional FDM 3D printer with ABS material also revealed a similar trend. This signifies that the approach shown in this work could be used in earlier design stages for potential savings in the lead time.

References

1. Huang, S. and J. Dong. *Optimization study of vehicle crashworthiness based on two types of frontal impacts*. in *2015 International Conference on Transportation Information and Safety (ICTIS)*. 2015. IEEE.
2. Yang, Z., Q. Peng, and J. Yang. *Lightweight design of B-pillar with TRB concept considering crashworthiness*. in *2012 Third International Conference on Digital Manufacturing & Automation*. 2012. IEEE.
3. Shin, J.-K., et al., *Automotive door design with the ULSAB concept. using structural optimization.* Structural and Multidisciplinary Optimization, 2002. **23**(4): p. 320-327.
4. Ahmad, Z., et al., *Nonlinear response topology optimization using equivalent static loads—case studies.* Engineering Optimization, 2017. **49**(2): p. 252-268.
5. Jeong, S.H., et al., *Topology optimization considering the fatigue constraint of variable amplitude load based on the equivalent static load approach.* Applied Mathematical Modelling, 2018. **56**: p. 626-647.
6. Bendsoe, M.P. and O. Sigmund, *Topology optimization: theory, methods, and applications*. 2003: Springer Science & Business Media.
7. (NHTSA), N.H.T.S.A. *Crash Simulation Vehicle Models*. [cited 2021 Jun 5].
8. Reichert, R. and S. Kan, *Structural Countermeasure/Research Program*. 2019, United States. Department of Transportation. National Highway Traffic Safety ….

DaQuan Prince-Floyd
Mr. Prince-Floyd is a graduate student currently pursuing his Master of Science in Engineering degree at Prairie View A&M University.

Jaejong Park
Dr. Park currently serves as an assistant professor of the Mechanical Engineering Department at Prairie View A&M University. His research interests include structural optimization, computational mechanics, biomimetic design, and artificial intelligence.

Jianren Zhou
Dr. Zhou is a professor at the Department of Mechanical Engineering, Prairie View A&M University. His research work is in areas of material property characterizations, nanomaterials, materials for extreme environments (space radiation and shielding), mechanical design with material considerations.