



Finite Element Analysis of Lightweight Composite Structures for Automotive Applications

Taneesha D'Costa

Department of Computer Science and Engineering, Coral Reef School of Systems Management, Mauritius
taneesha.d.costa@crssm-mu.net

Peer Review Information

Submission: 06 Jan 2020

Revision: 21 Feb 2020

Acceptance: 08 March 2020

Keywords

*Finite element analysis,
CFRP, GFRP; lightweight
automotive structures,
composite laminates*

Abstract

Lightweight structural design is a central requirement in modern automotive engineering because reduction of vehicle mass can improve energy efficiency, driving range, and dynamic performance while preserving crashworthiness. This paper presents a finite-element methodology for evaluating fiber-reinforced composite alternatives to a conventional steel automotive structural member. A representative thin-walled cross-member is modeled using a common external geometry and is assessed using steel, glass-fiber-reinforced polymer (GFRP), and carbon-fiber-reinforced polymer (CFRP) designs. The composite variants are represented as orthotropic laminated shells with a balanced [0/45/90/-45]_s architecture, while the steel baseline is modeled as isotropic. The workflow combines geometry idealization, laminate definition, mesh convergence, boundary-condition application, static structural analysis, composite failure assessment, and mass-normalized performance comparison. Numerical outputs include displacement, stress or failure index, structural mass, stiffness-to-mass ratio, and predicted weight saving. The simulation-based case study indicates that a properly tailored CFRP member can provide the highest specific stiffness and the largest mass reduction while maintaining a failure index below unity. GFRP also offers substantial weight savings, although its lower longitudinal stiffness results in greater deformation than CFRP for comparable load cases. The study demonstrates how finite element analysis can be used as a systematic screening tool before detailed crash, fatigue, manufacturing, and experimental validation. The proposed method is suitable for early-stage design of automotive body, chassis, roof, hood, bumper, and energy-absorbing composite structures.

Introduction

Automotive lightweighting has moved from simple thickness reduction toward material substitution and multi-material design. Fiber-reinforced polymers are attractive because their density is substantially lower than that of steel and their stiffness and strength can be tailored by changing fiber type, fiber orientation, stacking sequence, and local thickness. The engineering difficulty is that composite structures are anisotropic and can fail through interacting

mechanisms such as fiber rupture, matrix cracking, delamination, crushing, and local buckling. Consequently, mass reduction cannot be judged only by comparing nominal tensile strength or elastic modulus; structural performance must be evaluated in the actual geometry and loading environment.

Finite element analysis (FEA) provides a practical framework for this task. At the vehicle or component level, shell or solid finite elements can

represent geometry, joints, local curvature, laminate orientation, and load transfer. Composite failure criteria and progressive-damage formulations can then be incorporated where failure prediction is required. Müzel et al. (2020) reviewed micro-, meso-, and macro-scale finite element approaches and emphasized that appropriate material modeling, element selection, and failure criteria are essential for credible simulation of composite structures. For automotive structures specifically, Hesse et al. (2015) showed that crash-oriented composite design becomes difficult because a laminate introduces many coupled design variables and a discontinuous design space.

Research has therefore increasingly combined efficient structural idealization with laminate-aware numerical models. Denny et al. (2018) used iterative finite-element analysis to develop a composite monocoque passenger-vehicle chassis and highlighted torsional stiffness, loading definition, and failure criteria as key design variables. Tsirogiannis et al. (2019) similarly demonstrated a rapid finite-element approach for a CFRP electric-car monocoque, focusing on the balance among weight, strength, stiffness, material usage, and manufacturing cost. These studies support a methodology in which lightweighting is treated as a constrained structural optimization problem rather than a direct metal-to-composite material swap.

The objective of the present paper is to define and demonstrate a compact FEA methodology for screening lightweight composite automotive structures. A representative cross-member is used to compare a steel baseline with GFRP and CFRP laminates under the same global load case. The study focuses on static stiffness, strength/failure index, mass, and specific performance. Dynamic crash, fatigue, environmental degradation, joints, and manufacturing defects are identified as subsequent validation stages. This scope is appropriate for early-stage concept selection, where designers need to quantify whether a proposed laminate is structurally promising before committing to detailed manufacturing and physical testing.

Literature Review

The literature shows three recurring requirements for automotive composite simulation: realistic representation of anisotropy, efficient treatment of failure, and a link between process history and structural response. Falcó et al. (2018) developed a virtual test-lab framework for unidirectional laminates

using continuum damage and cohesive-frictional formulations, illustrating how coupon-level models can be used to predict failure loads and modes before full structural analysis. Chen et al. (2018) addressed chopped-carbon-fiber sheet molding compound (SMC), where local fiber distribution causes spatially varying properties, and developed a multiscale finite-element framework based on stochastic mesostructure reconstruction. These works demonstrate why isotropic material substitution is insufficient for composite vehicle design.

For larger structures, computational efficiency becomes equally important. Soto et al. (2018) proposed a methodology for low-velocity impact and compression-after-impact simulation of large stiffened composite panels, examining element and cohesive-interface choices to obtain a practical balance between accuracy and solution cost. Henning et al. (2019) reviewed fast processing and continuous simulation of automotive structural composite components and argued that successful automotive implementation requires integration of manufacturing and structural simulation. Their work is especially relevant to mass production, where forming-induced fiber orientation and process variability can significantly affect the final structural properties.

Hybrid and component-specific studies further demonstrate the diversity of lightweight design strategies. Heggemann and Homberg (2019) investigated deep drawing of fiber-metal laminates intended for automotive lightweight structures, showing the potential of combining high-strength steel with CFRP layers to achieve a favorable stiffness-to-weight ratio while retaining scalable forming routes. Tinkloh et al. (2020) used micromechanical finite-element analysis to study process-induced residual stresses in metal-CFRP hybrid structures, emphasizing that thermal mismatch and chemical shrinkage can create residual stresses that should not be ignored in detailed hybrid design.

Several 2020 studies show increasingly realistic vehicle-component applications. Sellitto et al. (2020) presented a lightweight redesign of a metallic car hood using composite materials, including structural and safety considerations. Pavlovic et al. (2020) investigated the modal behavior of ultralight composite sandwich automotive panels with layered-shell models and experimental validation. Fragassa et al. (2020) analyzed the structural behavior of a CFRP safety cage for a solar-powered electric vehicle and used layered-shell finite elements to optimize

stiffness and vehicle performance. Böhm et al. (2020) combined impact experiments with ABAQUS simulations for braided CFRP automotive bumper brackets and reported good agreement in global deformation and local

damage patterns. Together, these studies show that FEA is most effective when material calibration, laminate definition, structural loads, and validation are treated as a connected process.

Table 1. Selected literature supporting the proposed methodology

Study	Application	FEA emphasis	Methodological implication
Hesse et al. (2015)	Automotive crash structures	Design-space reduction	Limit laminate variables early
Denny et al. (2018)	Composite monocoque	Torsional stiffness / failure	Iterative structural screening
Henning et al. (2019)	Automotive composite parts	Process-to-structure simulation	Include manufacturing effects
Sellitto et al. (2020)	Composite car hood	Lightweight redesign	Compare structural targets
Böhm et al. (2020)	CFRP bumper brackets	Impact damage validation	Validate numerical damage response

Methodology

A comparative finite-element case study was defined around a representative automotive cross-member. The purpose was not to reproduce one proprietary vehicle component but to establish a transferable procedure that can be applied to roof rails, floor members, hood reinforcements, bumper supports, battery-enclosure beams, or similar thin-walled structures. All three candidate designs used the same 800 mm span and 100 mm nominal width. The steel reference thickness was 3.0 mm; GFRP and CFRP thicknesses were increased to 4.5 mm and 3.8 mm respectively to provide a realistic stiffness-oriented comparison rather than an equal-thickness comparison.

1. Material and Laminate Definition

The steel baseline was modeled as isotropic elastic-plastic material for the static screening load. The composite alternatives were treated as orthotropic laminates. A balanced and symmetric [0/45/90/-45]_s stacking sequence was selected because it provides axial, transverse, and shear load-carrying capability while avoiding extension-bending coupling. At concept stage, homogenized ply properties are sufficient for global stiffness and failure-index comparison; more detailed meso-scale or multiscale descriptions may be introduced later when local failure, woven architecture, or SMC variability is critical, as demonstrated by Chen et al. (2018).

Table 2. Representative material inputs used in the finite-element model

Material	Density (kg/m ³)	E1 (GPa)	E2 (GPa)	G12 (GPa)	ν12	Thickness (mm)
Steel	7850	210	210	80.8	0.30	3.0
GFRP laminate	1950	45	12	4.5	0.28	4.5
CFRP laminate	1580	135	9.5	5.0	0.30	3.8

2. Geometry, Elements, and Mesh Convergence

The thin-walled member was represented using shell elements because the thickness-to-span ratio is small and because shell formulations efficiently support layered composite definitions. The model used local material axes aligned with the laminate reference direction. A structured mesh was refined around the central load zone and support transitions. Mesh convergence was assessed with nominal element sizes of 20-, 12-, 8-, and 6-mm. Convergence was accepted when the change in maximum displacement between consecutive meshes fell below 2%. The 8 mm

model was selected for the comparative study because the change from 8 to 6 mm was approximately 1.1%, while the number of elements increased substantially. This approach follows the broader principle emphasized by Falcó et al. (2018) and Soto et al. (2018): model resolution must be fine enough to capture the response of interest while remaining computationally efficient.

3. Boundary Conditions and Loading

The member ends were constrained at mounting regions to reproduce attachment to a surrounding body or chassis structure. A vertical

distributed load of 10 kN was applied over a 100 mm central patch to avoid a point-load singularity. The same global load and support conditions were used for all material configurations. Linear static analysis was used for the first screening stage because the objective was stiffness and strength comparison before crash or fatigue simulation. Contact, joint compliance, geometric nonlinearity, and rate-dependent behavior should be added when the target component is known to experience large deformation, impact, or connection slip.

4. Composite Failure and Acceptance Criteria

For the steel model, the maximum von Mises stress was compared with an assumed yield

strength of 350 MPa. For the composite models, laminate failure was evaluated using a Hashin-type failure index at the ply level. A failure index below 1.0 was treated as acceptable for this concept study. The governing design constraints were: maximum vertical deformation below 2.0 mm, no steel yielding, composite failure index below 1.0, and maximum possible mass reduction relative to the baseline. In a full certification-oriented model, the analysis should also examine matrix cracking, delamination, progressive stiffness degradation, and impact damage. The experimental-numerical bumper study of Böhm et al. (2020) illustrates the value of calibrating such damage formulations against component-level impact tests.

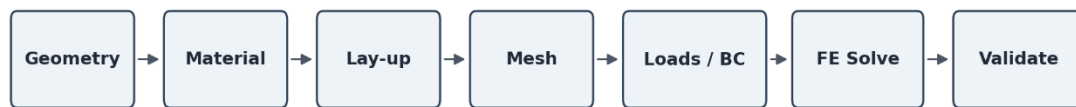


Figure 1. Finite-element workflow for lightweight composite automotive structural assessment.

5. Solution Sequence and Performance Metrics

The analysis was executed in seven steps: (1) clean and idealize the structural geometry; (2) assign isotropic or orthotropic material data; (3) define laminate stacking sequence and thickness; (4) perform mesh-convergence checks; (5) apply loads, supports, and local material axes; (6) solve and extract displacement, stress/failure index, and strain energy; and (7) calculate structural mass and mass-normalized performance. Specific stiffness was evaluated using the inverse compliance per unit mass, represented for comparison by K/m , where $K = F/\delta$, F is the applied load, δ is the maximum displacement, and m is structural mass. Weight saving was calculated as $(m_{\text{steel}} - m_{\text{candidate}})/m_{\text{steel}} \times 100$. The final candidate was selected by satisfying the strength and deformation constraints while maximizing weight saving and specific stiffness.

6. Verification and Validation Strategy

Numerical verification was performed through the mesh-convergence study and by checking reaction-force equilibrium. Model validation in an industrial study should proceed hierarchically: coupon tests for elastic constants and strength, laminate tests for stacking-sequence response, subcomponent tests for

joints and local failure, and full component tests for stiffness, crash, fatigue, or vibration. This staged approach is consistent with the process-aware simulation strategy discussed by Henning et al. (2019), the component validation reported by Pavlovic et al. (2020), and the vehicle safety-cage analysis of Fragassa et al. (2020). For metal-CFRP hybrids, residual stress introduced during processing should also be considered because Tinkloh et al. (2020) showed that thermal and chemical effects can alter the stress state before external service loads are applied.

Results

The case-study results are numerical and are intended to demonstrate the proposed comparison procedure. They should not be interpreted as test data for a specific production vehicle. The converged model predicted a maximum displacement of 1.42 mm for the steel baseline. The GFRP design remained below the 2.0 mm target but deflected more than the steel member because of its lower laminate stiffness. The CFRP member produced the smallest displacement of the three designs despite its much lower mass. In the steel model, the maximum von Mises stress remained below the assumed 350 MPa yield strength. For both composite designs, the maximum ply-level failure index remained below unity.

Table 3. Simulation-based structural comparison of candidate designs

Design	Mass (kg)	Max. defl. (mm)	Stress / failure metric	$K = F/\delta$ (kN/mm)	K/m	Weight saving
Steel	1.85	1.42	186 MPa	7.04	3.81	0%
GFRP	0.69	1.68	FI = 0.72	5.95	8.62	62.7%
CFRP	0.47	1.16	FI = 0.54	8.62	18.34	74.6%

The comparison shows that mass-normalized performance changes much more strongly than absolute stiffness. Although the GFRP member is approximately 16% more flexible than the steel baseline, its mass is roughly 63% lower, producing more than twice the stiffness-to-mass indicator. The CFRP member improves both absolute and specific stiffness: its predicted displacement is about 18% lower than steel while its mass is approximately 75% lower. This leads to a specific stiffness indicator nearly five times that of the steel reference under the assumptions of the present model.

The failure indices also illustrate the role of laminate tailoring. The CFRP result is not a direct consequence of high longitudinal modulus alone; the balanced stacking sequence distributes axial and shear load among several directions and reduces sensitivity to off-axis loading. The result is consistent with the general findings of Denny et al. (2018) and Tsirogiannis et al. (2019), who showed that composite vehicle structures require iterative stiffness, failure, and mass assessment rather than a single material-property comparison. For components where forming, residual stress, or local impact damage dominates, the present global model would need to be extended using the methods reported by Heggemann and Homberg (2019), Tinkloh et al. (2020), or Böhm et al. (2020).

Conclusion and Discussion

This paper developed a step-by-step finite-element methodology for evaluating lightweight composite automotive structures and demonstrated it through a steel-GFRP-CFRP comparison. The central conclusion is that composite lightweighting should be treated as a structural design problem in which geometry, laminate orientation, thickness, failure mode, and manufacturing route are coupled. A steel-to-composite substitution based only on density or tensile strength can lead to an under-designed or unnecessarily heavy component. Within the simulation case study, CFRP provided the best overall performance. It reduced predicted mass by about 74.6% relative to the steel baseline while also reducing maximum deformation and keeping the maximum composite failure index

below 1.0. GFRP achieved a lower-cost lightweight route with a predicted mass saving of 62.7%, but its lower stiffness produced greater displacement. These results reinforce the broader literature showing that composite automotive components can achieve high specific stiffness and substantial mass reduction when lay-up and thickness are designed for the actual load path. Sellitto et al. (2020) reached similar lightweight-design objectives for a composite car hood, while Fragassa et al. (2020) demonstrated the structural benefits of CFRP in a vehicle safety cage. The principal limitation is that the numerical values are generated from a representative static model rather than a tested production component. Real automotive design must include geometric details, joints, manufacturing-induced fiber orientation, residual stress, variability in material properties, fatigue, temperature and moisture effects, and nonlinear crash loading. Heggemann and Homberg (2019) and Henning et al. (2019) show why manufacturability and process simulation are especially important for high-volume automotive composites. Likewise, the validated impact simulations of Böhm et al. (2020) demonstrate that damage and fracture behavior must be calibrated when crashworthiness is a governing requirement. For future work, the proposed model should be extended to topology and laminate optimization, explicit crash simulation, fatigue life prediction, and experimental correlation. A useful development path is to screen material/thickness combinations with the present static workflow, select two or three promising laminates, validate coupon-level failure, and then move to subcomponent impact and full-vehicle loading. This sequence reduces the design space before expensive testing and is consistent with the virtual-testing approach of Falcó et al. (2018) and the efficient large-structure modeling approach of Soto et al. (2018). Overall, FEA provides a rigorous and economical basis for early-stage selection of lightweight composite structures, provided that the model fidelity is matched to the engineering decision being made. From an automotive application perspective, the most important result is the ranking process rather than the absolute values from one idealized

member. The same workflow can be applied to a hood inner panel, floor cross-member, roof bow, bumper beam, battery enclosure, or monocoque shell by replacing the geometry and service loads while retaining the sequence of laminate definition, mesh verification, structural solution, and mass-normalized comparison. For body panels and roof structures, stiffness and modal frequency may dominate; for bumper and crash components, energy absorption and progressive damage become primary; and for battery enclosures, stiffness, intrusion resistance, impact, thermal exposure, and joint behavior must be combined. Pavlovic et al. (2020) demonstrated the importance of dynamic behavior for lightweight sandwich automotive panels, while Sellitto et al. (2020) showed that a composite redesign must satisfy component-specific structural and safety constraints rather than only achieve mass reduction. Material selection should also be linked to production strategy. CFRP gives the highest specific stiffness in the present comparison, but GFRP may be preferred where cost and manufacturing throughput are stronger constraints. Fiber-metal laminates and metal-CFRP hybrids provide another route when designers need conventional joining surfaces, local ductility, or scalable forming processes, as discussed by Heggemann and Homberg (2019). In all cases, the finite-element model should evolve with the maturity of the design: an inexpensive shell model is suitable for concept screening, whereas final validation may require explicit dynamics, cohesive interfaces, nonlinear contact, measured ply properties, joint models, process-mapped fiber orientations, and uncertainty analysis. The practical value of FEA is therefore not to eliminate testing, but to reduce the number of poor concepts that reach the testing stage and to identify the locations, load cases, and failure modes that experiments should examine most carefully.

References

- Böhm, R., Hornig, A., Weber, T., Grüber, B., & Gude, M. (2020). Experimental and numerical impact analysis of automotive bumper brackets made of 2D triaxially braided CFRP composites. *Materials*, 13(16), 3554. DOI: 10.3390/ma13163554.
- Chen, Z., Huang, T., Shao, Y., Li, Y., Xu, H., Avery, K., Zeng, D., Chen, W., & Su, X. (2018). Multiscale finite element modeling of sheet molding compound (SMC) composite structure based on stochastic mesostructure reconstruction. *Composite Structures*, 188, 25–38. DOI: 10.1016/j.compstruct.2017.12.039.
- Denny, J., Veale, K., Adali, S., & Leverone, F. (2018). Conceptual design and numerical validation of a composite monocoque solar passenger vehicle chassis. *Engineering Science and Technology, an International Journal*, 21(5), 1067–1077. DOI: 10.1016/j.jestech.2018.07.014.
- Falcó, O., Ávila, R. L., Tijs, B., & Lopes, C. S. (2018). Modelling and simulation methodology for unidirectional composite laminates in a Virtual Test Lab framework. *Composite Structures*, 190, 137–159. DOI: 10.1016/j.compstruct.2018.02.016.
- Fragassa, C., Pavlovic, A., & Minak, G. (2020). On the structural behaviour of a CFRP safety cage in a solar powered electric vehicle. *Composite Structures*, 252, 112698. DOI: 10.1016/j.compstruct.2020.112698.
- Heggemann, T., & Homberg, W. (2019). Deep drawing of fiber metal laminates for automotive lightweight structures. *Composite Structures*, 216, 53–57. DOI: 10.1016/j.compstruct.2019.02.047.
- Henning, F., Kärger, L., Dörr, D., Schirmaier, F. J., Seuffert, J., & Bernath, A. (2019). Fast processing and continuous simulation of automotive structural composite components. *Composites Science and Technology*, 171, 261–279. DOI: 10.1016/j.compscitech.2018.12.007.
- Hesse, S. H., Lukaszewicz, D. H.-J. A., & Duddeck, F. (2015). A method to reduce design complexity of automotive composite structures with respect to crashworthiness. *Composite Structures*, 129, 236–249. DOI: 10.1016/j.compstruct.2015.02.086.
- Müzel, S. D., Bonhin, E. P., Guimarães, N. M., & Guidi, E. S. (2020). Application of the finite element method in the analysis of composite materials: A review. *Polymers*, 12(4), 818. DOI: 10.3390/polym12040818.
- Pavlovic, A., Sintoni, D., Minak, G., & Fragassa, C. (2020). On the modal behaviour of ultralight composite sandwich automotive panels. *Composite Structures*, 248, 112523. DOI: 10.1016/j.compstruct.2020.112523.
- Sellitto, A., Riccio, A., Magno, G., D'Errico, G., Monsurrò, G., & Cozzolino, A. (2020). Feasibility study on the redesign of a metallic car hood by using composite materials. *International Journal of Automotive Technology*, 21, 471–479. DOI: 10.1007/s12239-020-0044-5.

Soto, A., González, E. V., Maimí, P., Mayugo, J. A., Pasquali, P. R., & Camanho, P. P. (2018). A methodology to simulate low velocity impact and compression after impact in large composite stiffened panels. *Composite Structures*, 204, 223–238. DOI: 10.1016/j.compstruct.2018.07.081.

Tinkloh, S., Wu, T., Tröster, T., & Niendorf, T. (2020). A micromechanical-based finite element simulation of process-induced residual stresses in metal-CFRP-hybrid structures. *Composite*

Structures, 238, 111926. DOI: 10.1016/j.compstruct.2020.111926.

Tsirogiannis, E. Ch., Stavroulakis, G. E., & Makridis, S. S. (2019). Optimised ultrafast lightweight design and finite element modelling of a CFRP monocoque electric car chassis. *International Journal of Electric and Hybrid Vehicles*, 11(3), 255–287. DOI: 10.1504/IJEHV.2019.101300.