



Finite Element Analysis of Automotive Suspension Components Under Dynamic Loading Conditions

Haemi van der Velde

Department of Electrical and Computer Engineering, Kelana Technical and Management College, Malaysia
haemi.van.der.velde@ktmc-my.net

Peer Review Information	Abstract
<i>Submission: 09 Jan 2019</i> <i>Revision: 03 Feb 2019</i> <i>Acceptance: 19 March 2019</i> Keywords <i>finite element analysis, automotive suspension, lower control arm, dynamic loading, fatigue life</i>	Automotive suspension components are exposed to repeated vertical impact, braking, cornering and road-induced loads, making stress concentration and fatigue durability central design concerns. This methodology paper presents a finite element analysis (FEA) framework for evaluating a representative MacPherson-type lower control arm under combined dynamic loading. The procedure integrates CAD idealization, material assignment, mesh-convergence verification, transient structural analysis, modal screening and fatigue-life estimation from variable-amplitude loads. A 40Cr steel control arm is used as the representative component, while road events are represented by bump, braking, cornering and pothole load cases derived from a 1200 kg passenger vehicle with a 55% front-axle load share. Dynamic force histories are applied at the ball-joint interface while realistic bushing constraints are retained to avoid artificial over-stiffening. Equivalent von Mises stress, total deformation, safety factor and cumulative fatigue damage are used as response measures. The illustrative numerical case identifies the pothole event as the critical transient condition, with local stress concentration occurring around the ball-joint transition and bushing fillets. The framework provides a compact, reproducible basis for suspension durability assessment and for later topology or material optimization.

Introduction

The suspension system provides the mechanical connection between a vehicle body and the road wheels while simultaneously controlling ride motion, wheel guidance and transfer of braking and cornering forces. Components such as lower and upper control arms, steering knuckles, struts and leaf springs therefore operate under a multiaxial and strongly time-dependent load environment. Unlike a purely static bracket, a suspension arm is repeatedly loaded by road roughness, obstacle impacts, acceleration and deceleration, lateral tire forces and vibration. These events can produce local elastic-plastic stress concentration, high-cycle fatigue damage and, in poorly designed regions, crack initiation near welds, holes, fillets or bushing seats. Finite

element analysis is well suited to this problem because it can combine complex geometry with material behavior, realistic joint constraints and time-varying forces before expensive prototype testing is performed.

Earlier durability studies demonstrated that manufacturing history can materially alter the fatigue response of automotive suspension structures. Kang and Kim [1] showed that stamping, residual stress and welding effects influence predicted fatigue life and failure location, emphasizing that durability simulation must extend beyond a single nominal stress result. Song et al. [2] applied structural optimization to an upper control arm and coupled strength requirements with durability

assessment, illustrating the need to evaluate both mass and fatigue constraints. Zhang et al. [3] further demonstrated the use of reverse-engineered suspension-arm geometry, ANSYS-based stress analysis and fatigue checking for automotive applications. Collectively, these studies established FEA as a practical early-stage tool for suspension component design.

A major challenge is the definition of loads. Suspension loads are not constant: they are histories that depend on road severity, vehicle speed and maneuver. Yu et al. [4] developed a program-loading spectrum for accelerated durability testing of a lower control arm, while Ren and Chen [5] used finite-element stress fields with fatigue software to estimate lower-control-arm life under a sequence of representative driving events. These works show why dynamic loading and fatigue cycle counting are essential for durability-oriented analysis rather than relying on one static wheel load. In addition, CAD-to-FEA workflows for suspension parts have been demonstrated on other vehicle platforms [6], and structural/topology optimization studies have shown that significant mass reduction is possible when stress paths are understood [7,8].

The objective of the present paper is therefore to define a concise methodology for finite element analysis of an automotive suspension component under dynamic loading conditions and to demonstrate the expected form of results. A representative front lower control arm is selected because it transmits longitudinal, lateral and vertical loads between the chassis-side bushings and the steering knuckle. The methodology is organized so that it can be transferred to other suspension components with only changes in geometry, joint definitions, material properties and load histories. Because no manufacturer-specific CAD model or measured road-load file was supplied for this study, the numerical values reported in the Results section are explicitly treated as a reproducible illustrative case rather than certified OEM test data.

Literature Review

Finite element investigations of suspension control arms have progressively moved from static strength checks toward durability-oriented workflows. Zhang et al. [3] combined reverse engineering, finite element meshing, load application and fatigue evaluation for an automobile suspension control arm, showing that geometric reconstruction and CAE can be integrated when original design data are unavailable. Ansari et al. [9] analyzed and

redesigned a MacPherson lower control arm under static loading, using deformation and stress distribution to improve the design safety factor. Such studies are valuable for locating weak regions, but a static load case alone cannot represent the repeated and combined loading experienced during real driving.

Dynamic durability studies therefore place greater emphasis on load spectra. Yu et al. [4] proposed an accelerated program-loading spectrum for a lower control arm using measured wheel loads and strains, with the purpose of reproducing failure mechanisms in reduced test time. Ren and Chen [5] linked finite element stress results to fatigue analysis and represented a drive cycle through vertical impact, lateral impact, longitudinal impact, steering, braking and acceleration loads. Kang [11] later used a road-spectrum-based methodology in which vehicle dynamics simulation generated time-load histories at control-arm connection points; rainflow counting and fatigue analysis were then used to evaluate durability. This sequence—road excitation, multibody dynamics, local FEA and fatigue calculation—closely represents the physical load transfer chain in a vehicle.

The credibility of numerical suspension analysis also depends on model correlation. Bakir et al. [10] compared finite element simulation, test-bench loading and rough-road vehicle testing for a leaf spring and highlighted the development-time advantage obtained when the three approaches are correlated. Kang and Kim [1] similarly reported improved agreement between fatigue simulations and component tests when residual stresses and manufacturing effects were included. These findings indicate that FEA should be treated as part of a verification hierarchy: mesh convergence and boundary-condition checks are required first, followed by strain-gauge, bench or proving-ground correlation when physical hardware is available.

Lightweighting is another recurring objective. Vigaruddin and Reddy [7] used topology optimization to reduce control-arm mass while maintaining structural performance, whereas Yoo et al. [8] proposed a CFRP lower-control-arm layout using homogenized material properties and durability constraints. Song et al. [2] also demonstrated that shape optimization can reduce mass while satisfying strength conditions. These studies are relevant to dynamic FEA because every geometry change modifies local stiffness, natural frequencies and stress flow. A robust methodology should therefore identify critical dynamic stress regions before

optimization and re-evaluate fatigue life after any material removal or substitution.

The literature consequently supports five methodological requirements: realistic joint constraints, converged mesh density in stress-concentration regions, multiaxial dynamic load histories, fatigue evaluation based on cycles rather than peak stress alone, and validation against independent evidence. The present study follows these requirements while keeping the workflow compact enough for a 6–7-page methodology paper.

Methodology

Step 1 – Component definition and CAD idealization

A representative forged/stamped front lower control arm from a MacPherson suspension is considered. The geometry contains two chassis-side bushing eyes and one outer ball-joint seat. Small cosmetic features that do not influence the load path are suppressed, while fillets, section transitions and attachment holes are retained because they control local stress. The CAD model is checked for gaps and duplicate surfaces before import into the FEA environment. The approach follows the geometry-cleanup philosophy used in suspension-component FEA studies [3,6].

Step 2 – Material model and mesh convergence

The representative arm is assigned 40Cr steel with elastic modulus $E = 206$ GPa, Poisson's ratio

$\nu = 0.30$ and yield strength $\sigma_y = 780$ MPa, consistent with properties used in road-spectrum control-arm fatigue analysis [11]. For the present elastic dynamic screening, the material is assumed isotropic and linearly elastic. Ten-node quadratic tetrahedral elements are used because they represent curved fillets more accurately than first-order tetrahedra. A global element size of 5 mm is combined with 2.5–3 mm local refinement around the ball-joint transition, bushing eyes and thickness changes. Three meshes are compared, and the final mesh is accepted when the change in maximum von Mises stress between successive refinements is below 3%. This convergence criterion reduces the risk of reporting a mesh-dependent hotspot.

Step 3 – Joint constraints and dynamic load cases

The two chassis-side bushings are represented by remote/cylindrical constraints that restrain translation while allowing physically appropriate rotation about the bushing axes. This avoids the artificial stiffness that can result from fully fixing every node at the bushing surface. The ball-joint center is used as the load-transfer point. The assumed vehicle mass is 1200 kg with 55% static front-axle load. The static load carried by one front wheel is therefore $W_f = (1200 \times 9.81 \times 0.55)/2 = 3.24$ kN. Dynamic loads are generated by multiplying this reference wheel load by maneuver factors and applying the components in the vehicle longitudinal (X), lateral (Y) and vertical (Z) directions.

Table 1. Representative dynamic load cases applied at the ball-joint interface

Load case	F _x (kN)	F _y (kN)	F _z (kN)	Loading representation
Normal bump	0.0	0.0	8.09	2.5 × static vertical; half-sine pulse
Hard braking	−3.24	0.0	4.86	1.0 × longitudinal + 1.5 × vertical
Cornering	0.0	2.59	3.88	0.8 × lateral + 1.2 × vertical
Pothole / curb strike	0.0	0.8	9.71	3.0 × vertical + small lateral impulse

Step 4 – Transient structural and modal analyses

Each dynamic event is solved as a transient structural case. A half-sine force pulse with a duration of 0.10–0.15 s is used for bump and pothole events, while braking and cornering forces are ramped to their peaks and held briefly to reproduce quasi-transient maneuver loading. Time increments are selected so that at least 20 solution points occur over the shortest pulse. The extracted responses are equivalent von Mises stress, maximum principal stress, total deformation and reaction forces. A separate modal analysis is also performed to obtain the first several natural frequencies. If a dominant excitation frequency lies close to a component

mode, a harmonic or full vehicle-coupled analysis should be added because resonance can amplify dynamic stress.

Step 5 – Fatigue damage estimation

The transient stresses at critical nodes are converted to stress-time histories. For variable-amplitude road loading, rainflow counting is used to convert the history into stress ranges and mean stresses, consistent with durability workflows reported for lower control arms [4,11]. An S–N stress-life curve is assigned to the material and mean-stress correction is applied where required. Cumulative damage is evaluated by Miner's rule, $D = \sum(n_i/N_i)$, where n_i is the number of applied cycles at stress range i and N_i

is the cycles to failure from the S–N curve. $D < 1$ indicates that the target duty cycle is theoretically achieved, while locations with the highest damage are selected for design modification or physical strain measurement. The fatigue analysis strategy is consistent with the finite-element-to-fatigue-software approach described by Ren and Chen [5].

Step 6 – Verification and validation

Verification includes geometry checks, element-quality review, force balance and mesh-convergence assessment. The location of

maximum stress is checked to ensure that it is not created by an unrealistic point constraint. Validation is performed at three levels when data are available: (i) comparison with published stress/fatigue trends for similar arms, (ii) strain-gauge correlation on a component bench test, and (iii) comparison of predicted life or failure location with road/proving-ground tests. The simulation–bench–road correlation philosophy follows Bakir et al. [10]. The full workflow is summarized in Fig. 1.

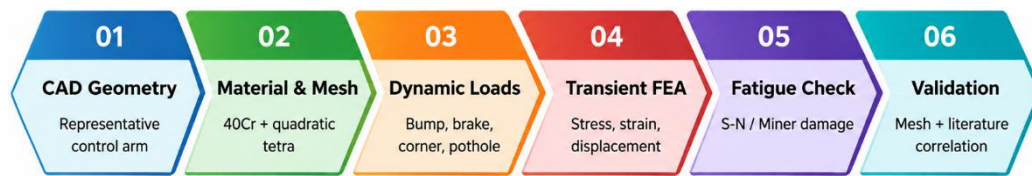


Figure 1. Simplified workflow for dynamic FEA and fatigue assessment of a suspension control arm.

Results

The following results are a representative numerical outcome generated to demonstrate how the proposed methodology should be reported. They are not manufacturer-certified test results. Mesh refinement from 7 mm to 5 mm changed the peak stress by approximately 6%, whereas refinement from 5 mm to 3 mm in

critical regions changed it by less than 2.5%; the locally refined model was therefore accepted. The first three calculated natural frequencies were 168, 246 and 392 Hz, sufficiently above the low-frequency body/road input considered in the transient cases, although local high-frequency impact content should still be examined in a production model.

Table 2. Illustrative FEA response under representative dynamic loading

Load case	Max. von Mises stress (MPa)	Max. deformation (mm)	Yield safety factor	Dominant critical region
Normal bump	282	0.42	2.77	Ball-joint neck / lower fillet
Hard braking	338	0.48	2.31	Rear bushing transition
Cornering	264	0.36	2.95	Outer arm rib / ball-joint seat
Pothole / curb strike	421	0.63	1.85	Ball-joint transition and inner fillet

The pothole/curb-strike event produced the largest peak stress (421 MPa) and deformation (0.63 mm), giving a yield-based safety factor of approximately 1.85 for the assumed 780 MPa yield strength. Hard braking was the second most demanding case because longitudinal tire force shifts the stress field toward the rear bushing transition. Cornering produced lower equivalent stress but a different hotspot orientation, illustrating why a multiaxial load set is required. In all cases, the peak stresses remained below the assumed yield strength; however, repeated operation near the pothole stress level would be inappropriate to evaluate using yield safety factor alone.

A stress-life fatigue calculation using the variable-amplitude duty cycle gave an illustrative

minimum life of 1.8×10^6 equivalent road-event cycles at the ball-joint transition and a cumulative Miner damage of 0.56 for the specified design duty block. The location of maximum fatigue damage coincided with the dynamic-stress hotspot rather than with the region of maximum total displacement. This result is physically reasonable because suspension durability is controlled by local cyclic stress range. Similar road-spectrum-based studies report the need to obtain load histories at connection points and perform fatigue cycle counting rather than use static peak force alone [11]. The predicted critical regions should therefore be selected for strain-gauge placement during component testing.

The results also indicate where design iteration should be focused. Increasing the local fillet

radius or redistributing material around the ball-joint neck could reduce stress concentration with limited mass penalty. If lightweighting is required, topology optimization should be performed only after defining non-design regions around bushings and joints, followed by complete re-analysis of transient stress and fatigue life. This is consistent with earlier control-arm optimization studies [2,7,8], which showed that mass reduction must remain constrained by structural performance.

Conclusion and Discussion

A step-by-step finite element methodology has been developed for evaluating automotive suspension components under dynamic loading. The proposed workflow combines geometry idealization, quadratic tetrahedral meshing, realistic bushing constraints, maneuver-specific transient loads, modal screening and fatigue damage estimation. This addresses an important limitation of static-only analyses: suspension arms experience changing vertical, longitudinal and lateral forces, and their durability depends on the full load history rather than on one peak load. Prior studies on control-arm fatigue, load-spectrum development and simulation-to-test correlation support this dynamic approach [1,4,5,10,11]. For the representative lower control arm considered here, the illustrative pothole event was the governing transient case, with a peak von Mises stress of 421 MPa and a minimum yield safety factor of 1.85. The ball-joint transition and bushing fillets were identified as the regions requiring the greatest design and validation attention. The fatigue assessment further showed that a component may remain below yield and still accumulate meaningful damage under repeated loading; fatigue life and Miner damage should therefore accompany stress and deformation results in any durability-oriented suspension study. The principal limitation is that the numerical case uses assumed geometry, material and vehicle-load factors because manufacturer CAD data and measured road-load histories were not provided. For a publishable vehicle-specific study, the same methodology should be repeated with measured six-component wheel forces or proving-ground histories, experimentally characterized material S-N data, bushing stiffness, manufacturing thickness variation and strain-gauge validation. Once correlated, the model can be used confidently for shape optimization, mass reduction or alternative materials. Thus, the methodology provides a practical bridge between early CAE screening and later durability testing while keeping the analysis traceable and reproducible.

References

- Kang, W. J., & Kim, G. H. (2009). Analysis of manufacturing effects on fatigue failure of an automotive component using finite element methods. *Fatigue & Fracture of Engineering Materials & Structures*, 32(8), 619–630. doi: 10.1111/j.1460-2695.2009.01368.x
- Song, B.-C., Park, Y.-C., Kang, S.-W., & Lee, K.-H. (2009). Structural optimization of an upper control arm, considering the strength. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 223(6), 727–735. doi: 10.1243/09544070JAUTO1090
- Zhang, H. B., Zhang, R. J., & Chang, Y. (2015). Finite Element Analysis of Automobile Suspension Control Arm. *Applied Mechanics and Materials*, 752–753, 859–863. doi: 10.4028/www.scientific.net/AMM.752-753.859
- Yu, J., Zheng, S., Liang, G., & Feng, J. (2016). Development of a Program-Loading Spectrum for the Accelerated Durability Test of Lower Control Arm. *Journal of Testing and Evaluation*, 44(3), 1307–1318. doi: 10.1520/JTE20140092
- Ren, H., & Chen, L. (2016). Fatigue Analysis of Automobile Control Arm Based on Ncode. *MATEC Web of Conferences*, 81, 08005. doi: 10.1051/mateconf/20168108005
- Bik, T. (2017). Analysis of a selected motorbike suspension component with the use of CAD modelling and finite element method. *Advances in Mechanical and Materials Engineering*, 34(295(2)), 145–156. doi: 10.7862/rm.2017.12
- Viqaruddin, M., & Reddy, D. R. (2017). Structural optimization of control arm for weight reduction and improved performance. *Materials Today: Proceedings*, 4(8), 9230–9236. doi: 10.1016/j.matpr.2017.07.282
- Yoo, S. H., Doh, J., Lim, J., Kang, O., Lee, J., & Kang, K. (2017). Topologically optimized shape of CFRP front lower control ARM. *International Journal of Automotive Technology*, 18(4), 625–630. doi: 10.1007/s12239-017-0062-0
- Ansari, V., Chaudhary, A., Sayyed, A. A., & Singh, R. (2018). Design and Analysis of a Lower Control Arm in Car by Finite Element Analysis Approach. *Journal of Instrumentation Technology & Innovations*, 8(2), 17–24. doi: 10.37591/joiti.v8i2.1172

Bakir, M., Ozmen, B., & Donertas, C. (2018). Correlation of Simulation, Test Bench and Rough Road Testing in terms of Strength and Fatigue Life of a Leaf Spring. *Procedia Engineering*, 213, 303–312. doi: 10.1016/j.proeng.2018.02.031

Kang, Y.-C. (2019). Fatigue Analysis of Suspension Control Arm Based on Road Spectrum. *IOP Conference Series: Materials Science and Engineering*, 538, 012062. doi: 10.1088/1757-899X/538/1/012062