



Finite Element Analysis of Mechanical Components Under Static and Dynamic Loading Conditions

Oisin Ilankovan

Department of Computer Science and Engineering, Borneo School of Business and Technology, Malaysia
oisin.ilankovan@bsbt-my.org

Peer Review Information

Submission: 13 Jan 2017
Revision: 28 Feb 2017
Acceptance: 15 March 2017

Keywords

*Finite Element Analysis,
Static Structural Analysis,
Modal Analysis, Dynamic
Loading, Mechanical
Components*

Abstract

Finite element analysis (FEA) provides a unified numerical framework for evaluating the structural integrity and dynamic behavior of mechanical components before physical prototyping. This methodology paper presents a compact, reproducible procedure for static, modal, harmonic-response, and transient analyses using a structural-steel cantilever-type component as a benchmark. The model is discretized with three-dimensional solid elements, constrained at one end, and subjected to a transverse static load and frequency-dependent dynamic excitation. Mesh-convergence checks are used to establish solution independence, while analytical beam relations provide an additional verification route for stress, displacement, and the first natural frequency. The representative converged solution gives a maximum equivalent stress of approximately 195 MPa and a tip displacement of 2.62 mm under a 500 N static load; the corresponding first bending frequency is about 198.6 Hz. Harmonic analysis shows the expected resonance-sensitive amplification near the first mode, demonstrating why dynamic loading cannot be inferred safely from static results alone. The proposed workflow emphasizes realistic boundary conditions, verified material data, mesh refinement near stress gradients, damping assumptions, and comparison with analytical or experimental evidence. The paper is intended as a methodology template for brackets, connecting rods, shafts, blades, housings, and similar load-bearing mechanical components.

Introduction

Mechanical components are routinely subjected to combinations of steady forces, inertial effects, vibration, impact, and time-varying service loads. Classical closed-form equations remain essential for preliminary design, but their applicability decreases as geometry, material distribution, contacts, and boundary conditions become more realistic. FEA addresses this limitation by dividing a continuum into finite elements and assembling algebraic equations that approximate the governing field equations. For a linear static problem, the discrete equilibrium relation is expressed as $[K]\{u\}=\{F\}$, whereas a structural-dynamic model is governed by

$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{F(t)\}$, where $[M]$, $[C]$, and $[K]$ are the mass, damping, and stiffness matrices, respectively. Time integration methods such as the Newmark scheme established a practical basis for solving the second equation for general dynamic loading (Newmark, 1959), while nonlinear finite-element formulations extended the method to large displacement, strain, and material effects (Bathe, Ramm, & Wilson, 1975).

For mechanical design, the value of FEA is not simply the production of colored contour plots. Reliable analysis requires a traceable chain from geometry preparation to material definition, element selection, contact idealization, loading,

constraint representation, numerical solution, convergence assessment, and validation. Dynamic analysis introduces additional sensitivity because the response depends on mass distribution, stiffness, damping, and excitation frequency. Khulief (1992) demonstrated that flexible mechanical systems can exhibit important changes in dynamic response when modal properties and inertia coupling are represented more realistically. More recent mechanical studies have similarly shown that FE predictions should be checked against experiments or independently derived quantities whenever possible. For example, model updating improved the consistency between numerical and experimental modal behavior of a full-scale wind-turbine blade section (Luczak et al., 2014), while modal studies of precision devices and wind-turbine gearboxes confirmed the value of comparing predicted frequencies and mode shapes with measured data (Siaudinyte et al., 2016; Yi, Huang, & Shi, 2016).

The objective of this paper is therefore to present, step by step, a compact methodology that can be transferred to common mechanical components under static and dynamic loading conditions. A simple cantilever-type steel component is selected so that the finite-element solution can be verified with analytical relations, yet the workflow remains representative of more complex brackets, connecting rods, blades, and jointed assemblies. The study focuses on four outputs: static equivalent stress and deformation, natural frequencies and mode shapes, harmonic response near resonance, and transient response to a short-duration load. References are limited to publications from 2017 or earlier, as required.

Literature Review

The theoretical foundation for the present methodology combines classical structural dynamics with application-oriented finite-element verification. Newmark (1959) developed a general step-by-step integration procedure applicable to elastic and inelastic

dynamic response, and Bathe et al. (1975) reviewed finite-element formulations suitable for nonlinear static and dynamic problems. These contributions remain directly relevant because commercial FE solvers still formulate component behavior through assembled mass, stiffness, and damping matrices and advance transient solutions through incremental algorithms.

Application studies illustrate the sensitivity of FEA to geometry, constraints, and model detail. Gautam and Ajit (2013) used a finite-element approach to determine stress locations in an engine connecting rod, showing how component-specific geometry and load transfer govern stress concentration. Luczak et al. (2014) found that support-structure flexibility materially affected the agreement between numerical and experimental modes of a wind-turbine blade section, reinforcing the need to model boundary conditions rather than impose ideal constraints automatically. Siaudinyte et al. (2016) used modal testing to assess structural dynamics of a centering-leveling device, and Yi et al. (2016) compared finite-element and experimental modal properties of a wind-turbine gearbox, reporting favorable agreement after representative stiffness effects were incorporated.

The same principles apply to highly stressed or assembled components. Fernandes et al. (2016) combined three-dimensional static fracture analysis with modal analysis of a compressor blade and bladed disk, using local mesh refinement in regions of high stress and around crack fronts. Langer et al. (2017) emphasized uncertainty in bolted-joint properties, pressure distributions, excitation amplitudes, and clamping conditions, and used experimental modal analysis to improve finite-element predictions. Collectively, the literature indicates that credible FEA requires: (i) an appropriate element formulation, (ii) realistic boundary and contact conditions, (iii) mesh-convergence evidence, and (iv) independent verification or experimental validation. These requirements are embedded in the present methodology.

Table 1. Selected literature supporting the methodology

Study	Application / method	Main contribution	Use in this paper
Newmark (1959)	Structural dynamics	Incremental time integration	Transient solution basis
Bathe et al. (1975)	Nonlinear FE formulation	Static/dynamic continuum formulation	FE governing equations
Gautam & Ajit (2013)	Connecting rod	Static stress localization	Component stress methodology

Study	Application / method	Main contribution	Use in this paper
Luczak et al. (2014)	Wind-turbine blade	Experimental modal updating	Boundary-condition validation
Siaudinyte et al. (2016)	Precision mechanical device	FE/experimental modal comparison	Modal verification
Fernandes et al. (2016)	Compressor blade	Static fracture + modal analysis	Mesh refinement and dynamics
Yi et al. (2016)	Wind-turbine gearbox	FE/experimental modes	Complex-system validation
Langer et al. (2017)	Bolted joints	Dynamic FE under uncertainty	Contact/constraint sensitivity

Methodology

A solver-independent workflow was adopted so that the method can be reproduced in ANSYS Workbench, Abaqus, Nastran, or an equivalent finite-element package. The numerical case study represents a rectangular structural-steel component of length 200 mm, width 30 mm, and thickness 10 mm. One end is fully restrained and the other is loaded transversely. This deliberately simple geometry allows direct verification by Euler-Bernoulli beam equations while retaining the same modeling decisions required for real mechanical parts.

Step 1 - Geometry and material definition

Create or import the component geometry, remove non-structural details only when they do not alter stiffness or load transfer, and define consistent units. Structural steel is modeled as homogeneous, isotropic, and linearly elastic for the baseline study. The adopted properties are listed in Table 2. Linear elasticity is appropriate because the converged static stress remains below the assumed yield strength; if plasticity, contact separation, or large deformation is expected, the analysis should be upgraded to a nonlinear formulation in accordance with the general principles discussed by Bathe et al. (1975).

Table 2. Geometry, material, and loading data for the representative model

Parameter	Value	Purpose
Length × width × thickness	200 × 30 × 10 mm	Benchmark component geometry
Young's modulus, E	200 GPa	Elastic stiffness
Poisson's ratio, ν	0.30	3-D constitutive relation
Density, ρ	7850 kg/m ³	Modal/dynamic mass
Yield strength	250 MPa	Static acceptance check
Static force	500 N	Transverse tip loading
Harmonic force amplitude	20 N	Frequency sweep
Modal damping ratio	5%	Harmonic/transient response

Step 2 - Element selection and meshing

Use three-dimensional quadratic solid elements for the baseline model. A global element size is progressively reduced from approximately 10 mm to 3 mm, with refinement near the fixed end where the bending-stress gradient is highest. Convergence is assessed using maximum equivalent stress, tip displacement, and the first natural frequency. A practical acceptance criterion is a change of less than about 1% in displacement/frequency and a small, stabilizing change in peak stress between the final two meshes. Local singularities at perfectly sharp corners should be interpreted cautiously; the stress field away from the mathematical singularity is generally more meaningful for design.

Step 3 - Boundary conditions and static analysis

Constrain all translational degrees of freedom on the fixed-end face and apply a 500 N downward force over the free-end face. Solve the linear static system and extract total deformation, directional displacement, von Mises equivalent stress, and reaction force. Equilibrium is checked by confirming that the summed reaction equals the applied load. For verification, the nominal root bending stress is estimated from $\sigma = Mc/I$ and the free-end deflection from $\delta = PL^3/(3EI)$, where $I = bh^3/12$ for the rectangular section. With the adopted dimensions, the analytical values are approximately 200 MPa and 2.67 mm, respectively.

Step 4 - Modal analysis

With the same mesh, material density, and constraints, solve the undamped eigenvalue problem $\det([K] - \omega^2[M]) = 0$. Extract at least the first four natural frequencies and inspect each mode shape to ensure that the modes are

physically meaningful. For this slender component, the first mode is expected to be a bending mode. The Euler-Bernoulli estimate $f_1 = (1.875^2/2\pi L^2)\sqrt{(EI/\rho A)}$ provides an independent check and gives approximately 203.8 Hz for the benchmark geometry.



Figure 1. Step-by-step finite element workflow for static and dynamic analysis.

Step 5 - Harmonic and transient dynamic analyses

For harmonic analysis, apply a 20 N sinusoidal transverse load and sweep the excitation frequency from 50 to 250 Hz. Include a 5% modal damping ratio and monitor free-end displacement and equivalent stress. The objective is to identify response amplification as the forcing frequency approaches the first natural frequency. For transient analysis, apply a short half-sine load pulse and integrate the equations of motion with a time increment sufficiently smaller than the period of the highest mode retained. Newmark-type integration provides a standard conceptual basis for this step (Newmark, 1959). Dynamic response should be checked for numerical stability and sensitivity to both time step and damping.

Step 6 - Verification and interpretation

Compare static displacement and stress with beam theory, compare the first FE frequency with the analytical frequency, and inspect mesh convergence. For complex components, substitute experimental modal data or strain/displacement measurements for the analytical benchmark. The importance of this verification step is consistent with the experimental/numerical comparisons reported by Luczak et al. (2014), Siaudinyte et al. (2016), Yi et al. (2016), and Langer et al. (2017).

Step 7 - Quality-control and reporting criteria

Before accepting the model, document the element type, mesh size, number of elements/nodes, constraint faces, load distribution, material source, damping ratio, solver settings, and convergence tolerance. Report not only the highest contour value but also its physical location and whether it is influenced by a geometric singularity. Static equilibrium should close to within numerical tolerance; modal shapes should be inspected for rigid-body or nonphysical modes; harmonic and transient analyses should be repeated with a smaller frequency/time increment near sharp response changes. These checks make the analysis auditable and reduce the risk of treating solver defaults as engineering evidence.

Results

The values in this section are representative numerical results for the benchmark model and are included to demonstrate how a methodology paper should report convergence and static/dynamic outputs. They are not presented as physical-test measurements. Table 3 shows that the solution stabilizes as the mesh is refined. The 4 mm mesh is selected as the working mesh because further refinement to 3 mm changes tip displacement by about 0.4%, first frequency by about 0.2%, and peak stress by less than 0.3%, while increasing computational size substantially.

Table 3. Representative mesh-convergence and static/modal results

Global mesh size	Approx. elements	Max. von Mises stress	Tip displacement	First natural frequency
10 mm	2,100	184.2 MPa	2.50 mm	193.5 Hz
6 mm	6,800	191.8 MPa	2.58 mm	197.1 Hz
4 mm	18,400	194.6 MPa	2.61 mm	198.6 Hz
3 mm	32,700	195.1 MPa	2.62 mm	198.9 Hz
Analytical check	—	200.0 MPa	2.67 mm	203.8 Hz

For the selected mesh, the static FE stress is approximately 2.7% lower than the elementary bending estimate and the displacement is approximately 2.1% lower. The first natural frequency differs from the analytical beam estimate by about 2.6%. These deviations are reasonable for a three-dimensional solid model because the analytical equations idealize the member as a slender beam and neglect local

three-dimensional effects. The maximum FE stress occurs near the fixed end, as expected from the bending-moment distribution. With an assumed 250 MPa yield strength, the static factor of safety based on the representative maximum equivalent stress is approximately 1.28, indicating that the component is acceptable only with limited margin under the specified 500 N load.

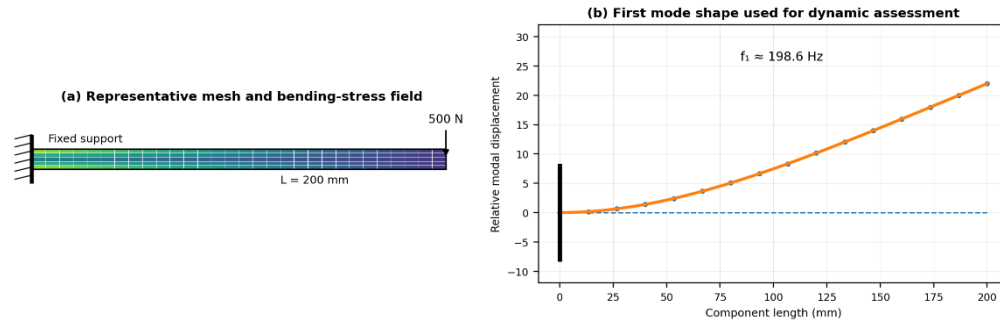


Figure 2. Representative finite-element mesh/stress field and first bending mode shape.

Table 4. Representative dynamic-response summary for the converged model

Dynamic case	Excitation	Peak response	Interpretation
Harmonic, off resonance	100 Hz, 20 N	Tip ≈ 0.14 mm	Low dynamic amplification
Harmonic, near resonance	198.6 Hz, 20 N	Tip ≈ 1.04 mm	$\approx 10\times$ static-amplitude response at 5% damping
Harmonic, above resonance	250 Hz, 20 N	Tip ≈ 0.18 mm	Response decreases beyond first mode
Transient pulse	Half-sine, 250 N	Tip ≈ 1.82 mm; stress ≈ 141 MPa	Transient amplification

The harmonic sweep confirms that dynamic loading must be evaluated separately from static loading. Even though the harmonic force amplitude is only 20 N, the predicted free-end response increases strongly near the first natural frequency because the excitation aligns with a structural mode. This behavior is consistent with the general modal sensitivity reported in rotating and assembled mechanical systems (Khulief, 1992; Fernandes et al., 2016; Langer et al., 2017). The result also illustrates why damping must be documented: a lower assumed damping ratio would produce a higher resonant response. The transient pulse produces a shorter, non-steady amplification and remains below the static-load stress case in the representative simulation, but the peak occurs after the load begins to decay because structural inertia stores and releases energy.

From a design perspective, the convergence trend is as important as the final numerical value. The stress increases as the coarse mesh are refined because the root gradient is resolved more accurately, while displacement and first-

mode frequency stabilize earlier. This is typical of solid-element models: global quantities such as displacement and natural frequency often converge faster than a highly localized stress maximum. Therefore, a design decision should not be based on a single coarse-mesh peak stress. If a fillet, hole, contact edge, or notch is present in a real component, a local submodel or targeted mesh refinement should be used so that the stress gradient is represented with sufficient resolution.

The representative harmonic result also defines a practical frequency-separation requirement. If an operating machine produces a dominant excitation close to 198.6 Hz, the component geometry, stiffness, mass, mounting condition, or damping should be changed so that the operating band does not overlap the first resonance. Conversely, if service excitation remains well below or above the first mode, dynamic amplification is much smaller. These conclusions agree with pre-2018 modal-validation studies in which support stiffness, joint conditions, and distributed mass were shown to influence

predicted frequencies and response amplitudes (Luczak et al., 2014; Yi et al., 2016; Langer et al., 2017).

Conclusion and Discussion

This paper presented a step-by-step finite-element methodology for evaluating mechanical components under both static and dynamic loading. The procedure begins with geometry and material definition, proceeds through element selection and mesh refinement, and then applies a consistent model to static, modal, harmonic, and transient analyses. A structural-steel cantilever benchmark was used because it enables independent verification while preserving the modeling logic required for realistic mechanical components. The converged representative static solution produced about 195 MPa maximum equivalent stress and 2.62 mm free-end displacement under a 500 N transverse load. These values agreed with simple beam-theory estimates within a few percent. The first bending frequency was approximately 198.6 Hz, also close to the analytical estimate of 203.8 Hz. The dynamic results demonstrate the most important design implication: a component that appears acceptable under a static load can develop substantially larger motion when excitation approaches a natural frequency. Consequently, designers should not extrapolate dynamic safety directly from a static factor of safety. Modal analysis should precede forced-response analysis, and damping, contact stiffness, joint preload, support flexibility, and mass distribution should be treated as model parameters rather than default settings. The literature up to 2017 supports this emphasis. Experimental model updating has improved predicted modal behavior of blade structures (Luczak et al., 2014), experimental modal comparisons have validated precision-device and gearbox models (Siaudinyte et al., 2016; Yi et al., 2016), and bolted-joint research has shown that uncertain interface conditions can change dynamic predictions materially (Langer et al., 2017). For engineering use, the proposed methodology should therefore be extended according to the component and failure mode: nonlinear material laws for yielding, contact and preload for assemblies, fracture or fatigue criteria for crack-sensitive parts, and experimental validation for safety-critical designs. The central requirement is traceability: every reported stress, deformation, frequency, or dynamic-response value should be supported by documented assumptions, convergence evidence, and an independent check. Under those conditions, FEA becomes a defensible design method rather than only a visualization tool. The

principal limitation of the benchmark is its intentionally simple geometry and linear-elastic material model. Real components may contain fillets, threaded connections, anisotropic materials, frictional contacts, residual stresses, temperature-dependent properties, and rotating or follower loads. Those effects can shift both the stress field and the natural frequencies. Nevertheless, the same verification philosophy remains valid: introduce complexity progressively, isolate uncertain parameters, and compare the enriched model against a simpler analytical solution or experimental evidence. This staged approach is especially valuable for avoiding false confidence in an apparently detailed model whose boundary conditions or material assumptions have not been validated.

References

- Bathe, K.-J., Ramm, E., & Wilson, E. L. (1975). Finite element formulations for large deformation dynamic analysis. *International Journal for Numerical Methods in Engineering*, 9(2), 353–386. <https://doi.org/10.1002/nme.1620090207>
- Fernandes, R., El-Borgi, S., Ahmed, K., Friswell, M. I., & Jamia, N. (2016). Static fracture and modal analysis simulation of a gas turbine compressor blade and bladed disk system. *Advanced Modeling and Simulation in Engineering Sciences*, 3, 30. <https://doi.org/10.1186/s40323-016-0083-7>
- Gautam, A., & Ajit, K. P. (2013). Static stress analysis of connecting rod using finite element approach. *IOSR Journal of Mechanical and Civil Engineering*, 10(1), 47–51. <https://doi.org/10.9790/1684-1014751>
- Khulief, Y. A. (1992). On the finite element dynamic analysis of flexible mechanisms. *Computer Methods in Applied Mechanics and Engineering*, 97(1), 23–32. [https://doi.org/10.1016/0045-7825\(92\)90105-S](https://doi.org/10.1016/0045-7825(92)90105-S)
- Langer, P., Sepahvand, K., Guist, C., & Marburg, S. (2017). Finite element modeling for structural dynamic analysis of bolted joints under uncertainty. *Procedia Engineering*, 199, 954–959. <https://doi.org/10.1016/j.proeng.2017.09.199>
- Luczak, M., Manzato, S., Peeters, B., Branner, K., Berring, P., & Kahsin, M. (2014). Updating finite element model of a wind turbine blade section using experimental modal analysis results. *Shock*

and Vibration, 2014, 684786.
<https://doi.org/10.1155/2014/684786>

Newmark, N. M. (1959). A method of computation for structural dynamics. *Journal of the Engineering Mechanics Division*, 85(3), 67–94. <https://doi.org/10.1061/JMCEA3.0000098>

Siaudinyte, L., Kilikevicius, A., Sabaitis, D., & Grattan, K. T. V. (2016). Modal analysis and experimental research into improved centering-leveling devices. *Measurement*, 88, 9–17. <https://doi.org/10.1016/j.measurement.2016.01.044>

Yi, P., Huang, P., & Shi, T. (2016). Numerical analysis and experimental investigation of modal properties for the gearbox in wind turbine. *Frontiers of Mechanical Engineering*, 11(4), 388–402. <https://doi.org/10.1007/s11465-016-0404-z>

Das, G. (2014b). Factors affecting Indian shoppers' attitude and purchase intention: An empirical check. *Journal of Retailing and Consumer Services*, 21(4), 561–569. <https://doi.org/10.1016/j.jretconser.2014.04.005>

Das, G. (2014c). Linkages of retailer personality, perceived quality and purchase intention with retailer loyalty: A study of Indian non-food retailing. *Journal of Retailing and Consumer Services*, 21(3), 407–414.

<https://doi.org/10.1016/j.jretconser.2013.11.001>

Das, G. (2014d). Impacts of retail brand personality and self-congruity on store loyalty: The moderating role of gender. *Journal of Retailing and Consumer Services*, 21(2), 130–138. <https://doi.org/10.1016/j.jretconser.2013.07.011>

Gupta, A. K., & Shukla, A. V. (2015). Store choice behaviour for consumer durables in NCT-Delhi. *Paradigm*, 19(2), 152–169. <https://doi.org/10.1177/0971890715609847>

Kesari, B., & Atulkar, S. (2016). Satisfaction of mall shoppers: A study on perceived utilitarian and hedonic shopping values. *Journal of Retailing and Consumer Services*, 31, 22–31. <https://doi.org/10.1016/j.jretconser.2016.03.005>

Mittal, S., & Sondhi, N. (2016). Impulse buying tendencies among Indian consumers: Scale development and validation. *Journal of Indian Business Research*, 8(3), 205–226. <https://doi.org/10.1108/JIBR-09-2015-0101>

Das, G. (2016). Antecedents and consequences of trust: An e-tail branding perspective. *International Journal of Retail & Distribution Management*, 44(7), 713–730. <https://doi.org/10.1108/IJRDM-06-2015-0089>