Proceedings of CONF-FMCE 2026 Symposium: Artificial Intelligence and Smart Sensing for Mechanical and Electrical Systems
DOI: 10.54254/2755-2721/2026.BA36717

A Review of the Application of CAE Simulation Technology in Mechanical Structural Optimization

Wenxuan Bian

Faculty of Engineering, University of Ulsan, Ulsan, Republic of Korea
18454713997@163.com

Abstract. As mechanical systems evolve toward lighter configurations and improved reliability, the drawbacks of traditional structural design methods, which rely heavily on physical tests, have become more evident, particularly in terms of time-to-market and cost control. Computer-Aided Engineering (CAE), as an important virtual simulation technology, employs Finite Element Analysis (FEA) as a core method to provide technical support for enhancing mechanical system designs. This research offers a thorough examination of three crucial applications of CAE simulation in structural optimization: stress analysis, vibration analysis, and thermal performance analysis. It also looks into the usage scenarios of two prominent simulation tools, ANSYS and Abaqus, emphasizing their individual advantages. Based on this and integrating real-life applications such as mechanical components and marine structures, this paper examines and emphasizes the practical value of CAE technology in structural optimization. The results show that CAE simulation plays a vital part in optimizing the development process, reducing testing costs, and significantly enhancing the accuracy and reliability of structural engineering. Meanwhile, the development of emerging technologies, such as multiphysics coupling analysis and AI-assisted simulation, provides new directions for the application of CAE in addressing complex engineering problems.

Keywords: CAE, Finite element analysis, Structural optimization, ANSYS, Abaqus

1. Introduction

In fields such as mechanical manufacturing and marine engineering, structural safety, reliability, and lightweight design directly determine the overall performance and market competitiveness of a product. Traditionally, structural design relied heavily on engineers' experience and iterative physical testing. This approach not only led to long development cycles and high costs but also often failed to accurately predict structural behavior under complex working conditions, making it unsuitable for the design requirements of modern high-end equipment. Computer-aided engineering (CAE), using finite element analysis (FEA) as its core method, enables multi-dimensional performance evaluation—including stress, vibration, and thermal analysis—through numerical simulation. It has thus become a key tool for structural optimization and is now widely applied to mechanical components and marine structures.

© 2026 The Authors. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).

DOI: 10.54254/2755-2721/2026.BA36717

This paper presents a review of CAE applications in structural optimization of mechanical systems. It systematically examines the core analysis types, characteristics of mainstream software, and representative case studies, while also summarizing the current challenges and development trends to provide guidance for engineering practice.

2. Fundamentals of CAE and finite element analysis

2.1. CAE and FEA concepts

CAE software supports parametric modeling and is capable of solving nonlinear problems while simulating multiple physical fields—including fluid dynamics, structural mechanics, and electromagnetics—primarily through numerical computation to predict product performance [1]. FEA, on the other hand, is one of the most widely used numerical analysis methods in CAE and serves as a principal tool for structural analysis. In practical engineering applications, CAE typically integrates multiple simulation and analysis functions, whereas FEA is primarily responsible for analyzing structural strength, stiffness, deformation, and stress. The two work in concert to support the design optimization of mechanical products. As mechanical structures evolve toward lightweight design, high reliability, and increased complexity, CAE and FEA have become critical technical foundations for structural optimization and performance evaluation in mechanical engineering.

2.2. Basic principles and solution process

Finite element analysis serves as the principal computational tool in structural optimization. Its core principle lies in discretizing a continuum into a finite number of elements, thereby approximating the solution of complex partial differential equations. The solution process comprises three main stages: geometric construction and mesh generation in pre-processing; formulation and iterative solution of the governing equations during computation; and result visualization and accuracy assessment in post-processing. This workflow translates theoretical predictions into practical engineering applications.

2.3. The role of CAE simulation in mechanical structure optimization

CAE, particularly finite element analysis, has become a major driving force behind design innovation. Engineers have made solid progress in using CAE for nonlinear problems, multiphysics coupling, and topology optimization. However, they still struggle to balance accuracy with computational cost, to build reliable multiscale models, and to merge data-driven approaches with physics-based principles. The following sections discuss how CAE works in practice, looking at static, dynamic, and thermo-structural coupled analyses.

3. CAE methods for structural optimization

3.1. Static stress and strength analysis

Finite element simulation has become indispensable for optimizing mechanical structures. Static strength and stress analysis remains the most common but essential task. It quantifies displacement, stress distribution, and strain in structures subjected to constant or quasi-static loads, allowing engineers to verify compliance with strength and stiffness requirements.

DOI: 10.54254/2755-2721/2026.BA36717

Recent studies illustrate the breadth of static analysis applications across mechanical engineering. Wu et al. integrated static analysis into digital twin frameworks, establishing it as a baseline method for predicting structural responses under sustained loads [2]. In a practical implementation, Ma employed ANSYS Workbench to assess a mechanical parking system [3]. Under full-load conditions, the longitudinal beams and carrying platforms exhibited maximum equivalent stress of 399.45 MPa and total deformation of 1.8071 mm—values that fell well within safe limits, thereby validating the structural integrity of the design.

The utility of static analysis extends beyond standalone strength evaluation. Chen demonstrated that prestressed static analysis serves as a necessary precursor to modal analysis for commercial vehicle frames, significantly enhancing the accuracy of subsequent dynamic predictions [4]. In marine engineering, Tan showed that static methods capture displacement and stress in ship structures under external loading, particularly when inertial and damping contributions are negligible [5]. Furthermore, Qian and Ye established that static analysis reliably determines displacement, stress, and strain fields in heavy machinery—including machine tool beds and presses—providing a sound basis for strength verification and design optimization [6, 7].

Collectively, these studies confirm that static strength and stress analysis constitutes a foundational element of structural optimization. When combined with accurate load simulation and careful finite element modeling, it yields quantitative metrics that guide design decisions and ensure mechanical performance.

3.2. Vibration and modal analysis

Vibration and modal analysis are primarily used to evaluate dynamic characteristics by identifying the natural frequencies and vibration modes of mechanical structures, thereby providing a basis for structural improvements. This helps to minimise the risk of resonance and enhance overall performance.

Chen Shanshan et al. studied a commercial vehicle chassis. Through CAE modal analysis, they identified low-frequency resonance in the chassis and proposed adding cross-members, which effectively increased the first-order natural frequency and improved the structural dynamic performance [4]. Ma used ANSYS software to perform static and modal analyses on a mechanical parking system, calculating the first 300 modes and obtaining the main vibration characteristics of the parking system frame under both unloaded and fully loaded conditions, laying the foundation for subsequent dynamic performance studies. In her research on ship structural design, Tan pointed out that, compared with traditional empirical formulas, CAE modal analysis can establish a more accurate model of the overall structure and comprehensively consider environmental factors such as water conditions, thereby yielding vibration modal results that better reflect actual operating conditions [5].

Modal analysis is a key step in integrating static design with dynamic characteristics. By calculating a structure's natural frequencies and vibration modes, it provides a theoretical basis and practical measures for lightweight design and resonance avoidance of various mechanical systems, such as motor vehicles, parking systems and ships.

3.3. Thermal analysis and thermo-structural coupling

CAE technology is utilised to perform temperature field simulations, thermal stress analysis and multi-physics coupling studies of thermal and structural behaviour on mechanical structures, thereby addressing temperature-induced deformation and failure in complex thermal environments. Tan

DOI: 10.54254/2755-2721/2026.BA36717

pointed out that ANSYS software can address issues relating to temperature distribution, heat dissipation, and multi-physics coupling, providing tool support for subsequent thermal analysis [5]. Ye further noted that thermal analysis enables accurate determination of temperature distributions, heat flux characteristics, and thermal stresses and deformations caused by temperature changes, which is important for the optimal design of critical components such as engine pistons and turbine blades [7]. When mechanical, thermal and fluid fields interact and couple with one another, thermal–structural coupled analysis must be employed to evaluate the stresses and deformations of components under combined multiple loads, providing a more accurate reflection of the structure's actual operating conditions. With the advancement of physical field coupling technology, thermal analysis has evolved from temperature field analysis to thermo-structural coupled analysis, providing technical support for thermal management optimization and high-reliability design in mechanical structures.

3.4. Structural optimization methods

CAE structural optimisation methods primarily include parametric optimisation, topological optimisation and comprehensive performance optimisation, with the aim of achieving lightweight design, improving material utilisation and optimising overall performance whilst ensuring structural strength and reliability.

Tan points out that CAE simulation enables the early assessment of structural performance during the product design phase, providing key theoretical support for design optimisation [5]. Wu summarized the main applications of digital twin technology in industrial manufacturing, emphasizing that by integrating real-time data with simulation models, it enables the optimization of mechanical systems throughout their life cycle [2].

Furthermore, Qian argued that structural optimisation should balance equipment strength, reliability and economic viability, achieving energy savings and reduced consumption whilst meeting functional requirements [6]. Chen employed a method combining finite element analysis with modal analysis to optimise commercial vehicle chassis, thereby demonstrating the effectiveness of CAE technology in enhancing structural dynamic performance [4]. Furthermore, the research team at Liuzhou Fuzhen Vehicle Body Industrial Co., Ltd. (2021) conducted an in-depth study on the role of CAE structural optimisation in automotive lightweight design, proposing that topological optimisation technology can identify reasonable material distribution schemes within the dimensional range of the overall model, thereby significantly improving material utilisation efficiency [8]. Ye proposed that combining parametric modelling with response surface approximation techniques can balance model accuracy and computational efficiency, enabling an intelligent optimisation workflow with self-sensing and self-adjusting capabilities [7]. Another study on the cutting heads of coarse grain harvesters operating in hilly terrain demonstrated that, by conducting static analysis and structural topological optimisation using ANSYS Workbench, a 21.4 per cent reduction in the support structure was achieved whilst maintaining maximum deformation within 2 mm and a volume fraction of 60 per cent. This reduced maximum stress and deformation levels, enhancing stability and reliability under complex operating conditions [9]. Current research has shifted from focusing solely on structural strength analysis to an integrated optimisation system combining parametric modelling, topological optimisation, multi-objective coordination and intelligent algorithms, thereby significantly enhancing the scientific rigour, cost-effectiveness and reliability of mechanical structural design.

DOI: 10.54254/2755-2721/2026.BA36717

4. Benchmarking mainstream CAE software

4.1. ANSYS

ANSYS is currently one of the most widely used CAE software packages for mechanical structural analysis and optimisation. It offers robust capabilities in finite element analysis, multi-physics coupling analysis, and structural optimization, and can be applied to static, modal, and thermal analysis.

Zhao and Gao introduced the core features of ANSYS and its fundamental applications in mechanical analysis, and illustrated the software's important role in structural design and functional evaluation using the example of a spatial truss [10]. Wang studied the basic analytical methods and simulation workflows of ANSYS in mechanical structural analysis, drawing on engineering case studies, and highlighted its application potential in mechanical engineering [11]. Furthermore, Li and colleagues analysed the application of ANSYS software in the teaching of engineering structures courses [12]. They emphasised that the introduction of this software not only enhances students' enthusiasm for studying engineering structures but also enables them to become proficient in using a structural design tool that is widely adopted globally, improving teaching effectiveness. In summary, from an introduction to its basic functions through to specific engineering applications and teaching practice, ANSYS software has proven its irreplaceable value in mechanical structure optimisation, static analysis and vibration and modal analysis, serving as a vital bridge between theoretical analysis and engineering practice.

4.2. Abaqus

Abaqus is a CAE software package for non-linear finite element analysis. It offers advantages in complex contact, material non-linearity, large deformation analysis, and dynamic analysis, and is widely used in mechanical mechanism optimization and simulation of complex operating conditions. Tan pointed out that Abaqus is capable of effectively handling non-linear contact and friction problems in ship structural design, and was one of the more advanced simulation software packages of its time [5]. During mechanical structure optimization, Abaqus can perform calculations ranging from linear to complex non-linear analysis, and is suitable for numerical simulation of various engineering materials, including metals and composites. Furthermore, Abaqus is used in conjunction with ANSYS and ADAMS. For example, by establishing a kinematic model of a mechanism using ADAMS and then combining this with finite element analysis using ANSYS or Abaqus, it is possible to achieve coupled simulation of complex mechanical systems, thereby improving the accuracy of structural design. In a study on the optimisation of the cutting unit for a coarse grain harvester in hilly terrain, co-simulation technology effectively enabled lightweight structural design and improved the equipment's dynamic response performance. With its powerful non-linear analysis capabilities, Abaqus has become a key tool in mechanical structure optimization, particularly for complex contact and friction problems, supporting performance and reliability improvements in mechanical equipment.

DOI: 10.54254/2755-2721/2026.BA36717

4.3. Summary of CAE software benchmarks

Table 1. Functional features and application domains of mainstream CAE software packages

Software	Strengths	Limitations	Application Areas
ANSYS	Multiphysics coupling; nonlinear and impacy analysis	High cost; steep learning curve	Crash simulation, ship collision, and education
ABAQUS	Nonlinear contact, friction, and large-deformation analysis	Limited multiphysics capability	Geotechnics, composites, metal forming
NASTRAN	Stable structural solvers; certification-oriented analysis	Limited nonlinear function; outdated interface	Aerospace structures, linear mechanics
AUTOSEA	Statistical energy analysis for vibration and acoustic simulation	Limited to mid/high-frequency application	Ship and vehicle NVH optimization

Table 1 benchmarks mainstream CAE software by nonlinear solver capability, ease of use, and domain-specific fit. These comparisons guide the selection of tools for mechanical structural optimization.

5. Representative engineering applications

5.1. Structural optimization and lightweight design

This section introduces typical applications of CAE simulation technology in the optimisation of mechanical structures and lightweight design. Through methods such as finite element analysis, topological optimisation and multi-condition simulation, structural weight reduction is achieved whilst meeting structural strength and stiffness requirements, thereby improving the dynamic characteristics and operational stability of mechanical equipment.

Tan points out that, in ship structural design, CAE structural optimisation can effectively achieve lightweight design, reduce energy loss caused by deadweight, and significantly improve propulsion performance [5]. Chen studied the chassis of a commercial vehicle. By employing CAE simulation combined with modal analysis, she optimised the structural design, increasing the first-order natural frequency and thereby improving the structural dynamic performance whilst meeting strength requirements [4]. Liao et al. used finite element analysis and topological optimisation for the lightweight design of the cutting unit of a coarse grain harvester used in hilly regions. Through static analysis and modal calculations, they improved the structural stability and mechanical performance [9]. Qian argues that the optimisation of mechanical systems should balance performance enhancement with energy conservation whilst meeting functional requirements; FEA can provide a reliable theoretical basis for optimisation schemes [6]. Ye further summarised the application process of finite element analysis in mechanical structural optimisation, noting that balancing optimization objectives, computational accuracy, computational accuracy and computational efficiency is a key factor in achieving high-quality structural optimisation [7]. Judging from existing research and practical case studies, CAE simulation technology plays a key role in the lightweight design of mechanical structures. Through methods such as topological optimisation, modal analysis

DOI: 10.54254/2755-2721/2026.BA36717

and multi-condition simulation, it is possible not only to improve load-bearing capacity and dynamic performance, but also to shorten product R&D cycles and reduce development costs.

5.2. CAE applications in ship structural design

CAE simulation technology has become an essential technical tool in modern ship structural design, with applications in overall strength analysis, vibration modal analysis, and impact resistance assessment. Tan has systematically studied the application of CAE technology in ship structural design [5]. He pointed out that, as ship structures become more complex, traditional beam-based calculation methods can no longer meet the demands of modern high-precision ship design, whereas CAE analysis based on the finite element method (FEM) can establish comprehensive finite element models of the hull, enabling precise analysis of both overall and local strength. Under extreme operating conditions such as wave loads, longitudinal bending, underwater explosive impacts and acoustic radiation, CAE can more accurately assess the stress state and safety performance of ship structures, providing a scientific basis for ship design in special environments such as deep-sea mining and polar navigation. The study also noted that factors such as the quality of the finite element model, the accuracy of the mesh, and the experience of engineering personnel all influence the precision of the simulation results. Overall, CAE technology is driving the transition of ship structural design from empirical design to digital design, and will play a growing role in the design of complex offshore equipment and high-performance vessels.

6. Challenges and future trends in CAE

In the future, CAE technology will become integrated with digitalisation and intelligent systems, serving as a pillar in mechanical design and manufacturing. CAD and CAE will be deeply integrated, enabling operations such as mechanical analysis and thermodynamic simulation to be carried out simultaneously during the design phase, thereby creating a seamless data flow from design to manufacturing and significantly reducing the costs associated with physical prototype development. CAE will extend into the early stages of design, enabling simulation at the concept stage, where performance verification and defect prediction can be done, breaking away from the traditional sequential development model. Leveraging multi-disciplinary co-simulation and virtual prototyping technologies, CAE can perform coupled simulations across multiple physical fields, including structural, dynamic and fluid dynamics. At the same time, CAE will integrate deeply with AI and digital twin technologies to provide online simulation, dynamic optimisation and intelligent decision-making support throughout the entire product lifecycle. At the industry development level, CAE will evolve towards higher precision, greater flexibility and greater environmental sustainability. With unified data standards and interface specifications, it will achieve cross-system and cross-platform data interoperability and collaboration, providing technological support for smart manufacturing and digital factories.

7. Conclusion

This review has examined how CAE simulation supports mechanical structural optimization. It traced the core principles of finite element analysis, surveyed the capabilities of mainstream software platforms, and covered their applications in static analysis, vibration, and thermal analysis.

Compared with traditional build-and-test cycles, numerical simulation lets engineers predict structural performance during the design phase. This shift compresses development timelines, cuts

DOI: 10.54254/2755-2721/2026.BA36717

prototyping costs, and reduces test-related risks. Case studies on mechanical components and bridge structures confirm that effective use of CAE can reduce weight while maintaining—or improving—safety margins.

Yet gaps remain. Multi-physics simulations under realistic operating conditions still lack the precision needed for critical decisions. Nonlinear solver settings and boundary conditions often depend on analyst judgment, introducing uncertainty. AI-assisted methods have entered the workflow, but intelligent optimization algorithms remain poorly integrated with engineering data. Fully automated structural optimization is still out of reach.

References

[1] Wang, X. (2003). *The finite element method*. Tsinghua University Press.

[2] Wu, Y., Wang, X., He, Y., et al. (2021). A review of key technologies and applications of digital twins in manufacturing. *Modern Manufacturing Engineering*(9), 137-145. https://doi.org/10.16731/j.cnki.1671-3133.2021.09.023.

[3] Ma, S., Zhou, J., Sun, H., et al. (2019). Static and vibration modal analysis of mechanical parking systems based on the finite element method. *Mechatronics, 25*(Z1), 54-60. https://doi.org/10.16413/j.cnki.issn.1007-080x.2019.z1.009.

[4] Chen, S., Wei, S., Ji, C., et al. (2019). Structural optimization design of a commercial vehicle chassis based on modal analysis. *Equipment Manufacturing Technology*(11), 40-43+72. https://kns.cnki.net/kcms2/article/abstract?v=f9AMVDUgxC9X68KxLHgYRQ2N7mKVgQqQf2A3OJ1l0y0WrqtZSNL0-WOw5lHv_pQX4gG3c0VPDxn65PKn4ffH3aW3aHjZ1JA16sWoYqSmWYzApY7b5bpx7qv2j_3-YUCfOg8j1HmV7YFUT9dCwZm4_en6bU5KLVf5hyshudx6gFqwbzV29dxQUQ==&uniplatform=NZKPT&language=CHS

[5] Tan, Z. (2025). Applications and prospects of CAE in ship structural design. *Pearl River Transport*(15), 83-85. https://doi.org/10.14125/j.cnki.zjsy.2025.15.025.

[6] Qian, K. (2025). Optimization and energy-saving strategies for industrial machinery based on advanced simulation technologies. *Comprehensive Utilization of Tire Resources in China*(4), 129-131. https://doi.org/10.19307/j.cnki.ctrr.2025.04.060.

[7] Ye, S. (2025). A study on the optimization design of mechanical structures based on finite element analysis. *Architecture and Design, 1*(9). DOI: 10.63887/jad.2025.1.9.23

[8] Yu, H. (2021). The role of CAE structural optimization in automotive weight reduction development. *Chinese Science and Technology Journal Database (Full-Text Edition), Engineering and Technology, 4*, 265-267. https://kns.cnki.net/kcms2/article/abstract?v=sFGZ-GfRoEVDIqxdE6D6gdmPPPW0OYmkpCxhU1G82qNrIv0eh6-nhn2nMdorAcLXqRHnrUhlHtU9XywnO_zQnWh9cA__XIAAlwsM3pk2gIbWJfEUDIrMhE9apf7MBvk-4vwl2xLogJ7Va9R2-Mqi0MjdqE2WI_GbkVlDZ3Sz0vEtaUiJXVCEHQ==&uniplatform=NZKPT&language=CHS

[9] Liao, B., Zhu, L., & Zhong, C. (2026). Lightweight design and structural optimization of a minor grain header for use in hilly terrain. *Research on Agricultural Mechanization, 48*(6), 44-51. https://doi.org/10.13427/j.issn.1003-188X.2026.06.006

[10] Zhao, H., & Gao, X. (2004). ANSYS software and its application. *Manufacturing Automation*(5), 20-23. https://kns.cnki.net/kcms2/article/abstract?v=f9AMVDUgxC9n2PoOQ7pPhQdRs3LmxDrrvkhJhmXQ0iGQq8eGzaCe6IJDU54ZV6IPdNK-uEzQZBBe__c85182URAVwiDROOyo2jB8q-IOlzQdOX9SmCXA2SctAGYtRBjkYcwOCywnnM-HvOA1dqyA0pJ__kHfN_2kWWlXhAcaN547vRggwf4Ddg==&uniplatform=NZKPT&language=CHS

[11] Wang, Y. (2014). Application of ANSYS software in mechanical structural analysis. *Value Engineering, 33*(3), 2. DOI: 10.3969/j.issn.1006-4311.2014.03.110

[12] Li, H., Yang, P., Huang, X., Huang, D., & Zheng, X. (2014). The application of ANSYS software in teaching engineering structures. *Higher Architectural Education, 23*(4), 137-139. https://kns.cnki.net/kcms2/article/abstract?v=f9AMVDUgxC9sQsMk9zuFOWrHZaErpbKh9IMJLFGmTOVo5vX6AV2lGlbQpHJ0p7u1CGrrQZGWc_W-yG1z93vAW_JQzy6kAhB4rjeJuAf0O9iOf-R1DssjkuHkGlO9vVVsLRKKuK92nA4Z01dOBYMqJVuaamzDOznNiTKB2mscXMaBEWC5loLjng==&uniplatform=NZKPT&language=CHS