Scientific Research Publishing

World Journal of Engineering and Technology, 2026, 14(4), 1032-1051
https://www.scirp.org/journal/wjet
ISSN Online: 2331-4249
ISSN Print: 2331-4222

FEA-Based Structural Evaluation of Heavy-Duty Automotive Lift Systems

Srinath Rajure

Western Michigan University, Kalamazoo, Michigan
Email: srinath0472@gmail.com

How to cite this paper: Rajure, S. (2026) FEA-Based Structural Evaluation of Heavy-Duty Automotive Lift Systems. *World Journal of Engineering and Technology*, **14**, 1032-1051.
https://doi.org/10.4236/wjet.2026.144062

Received: September 9, 2026
Accepted: November 7, 2026
Published: November 10, 2026

Copyright © 2026 by author(s) and Scientific Research Publishing Inc.
This work is licensed under the Creative Commons Attribution International License (CC BY 4.0).
http://creativecommons.org/licenses/by/4.0/

Open Access

Abstract

A heavy-duty automotive car lift is usually employed, primarily for the purpose of providing structural strength and safety, in the maintenance and repair of motor vehicles. Structural reliability and safety are among the most important uses of heavy-duty car lifts in vehicle maintenance and repair, and are closely related to high lifting duties. The purpose of this study is to investigate a generic, four-post heavy-duty lift system in the automotive industry using Finite Element Analysis (FEA) in a realistic structural investigation to determine the regions of high stress for analysis of good load-bearing performance in the automotive industry. A three-dimensional computer-aided design (CAD) model was developed and analyzed for the maximum rated vehicle loads. Structural responses like von Mises stress, total deformation, and factor of safety were numerically analyzed using linear static analysis. A parametric study was also performed to examine how varying the applied loads and structural member thickness would affect system performance in order to gain insight into these parameters. The analysis results indicate that the loads applied to the structure do not surpass the material's yield strength, and that the observed deformations are within permissible service specifications, both favorable aspects indicating that the proposed design is structurally adequate. Member thickness was an important factor, with thicknesses significantly lowering deflections and stress concentrations, leading to good stability. In the field of auto maintenance, the proposed method in this paper is a new technique that can be applied to the primary design, Analysis and Optimization of heavy-duty automotive lift system, and is cost-effective and efficient.

Keywords

Finite Element Analysis (FEA), Automotive Lift System, Structural Evaluation, Von Mises Stress, Factor of Safety

1. Introduction

Automotive lift systems that are used for heavy-duty applications in areas such as auto service, commercial vehicle and industrial vehicle maintenance, and manufacturing must provide safe and efficient lifting of vehicles. These systems raise vehicles and cars weighing many tons and make inspection, repair, and maintenance work, and assembly of these vehicles easier and safer. The automotive sector is developing rapidly, and the market for various types of heavy industrial products that are now in use in production on a worldwide scale is catching up; the demand for reliable lifting infrastructure with high load capacity and a structure with high stability continues to increase significantly. The efficacy, safety, life, maintenance requirements, and productivity of these lifting systems can be impacted by heavy static and dynamic loads. When one of the components is broken, such as cross beams, work platforms, lifting platforms, and support columns, it may lead to fatigue accidents, destruction of equipment, and a huge economic loss. Therefore, designing and manufacturing reliable lift systems for the automotive industry is an important issue to take into consideration [1]. The traditional method of designing auto lifts has been based on analytical formulas derived from beam theory, strength of materials principles, and conservative, safety factor-based rules of thumb. These methods are useful for an initial estimate of load-carrying capacity, but do not always reflect the presence of localised areas of high stress, complex variations of deformation, and load bearing between sections of structures in welded joints and other load bearing spans. As lift designs get complex, their advanced simulation brings in demand for the implementation of advanced computational approaches that simulate the operating conditions of realistic dimensional models before they are manufactured. Computer-Aided Design (CAD) combined with Finite Element Analysis (FEA) is one of the most widely used engineering methodologies for determining the behavior of structures for various loading circumstances [2]. FEA splits a larger structure into smaller pieces called finite elements, and solves the governing equations numerically to simulate it and predict information like stress, strain, displacement, reaction forces, and so on, for each finite element in the structure. Besides identifying potential areas of failure, this numerical method could also be used to optimise the dimensions of a structure, confirm it is safe, and drastically cut the number of tests conducted and development time [3].

In recent years, finite element concepts have been widely applied to numerous areas, including mechanical, civil, aerospace, and automotive engineering fields, due to their ability to predict structural performance accurately. Such modern simulation software provides von Mises stress distribution, principal stresses, equivalent strains, total deformations, and factor of safety, which allow evaluation of different design alternatives without the usual construction of a prototype [4]. These calculation methods can enhance the quality and reliability of products, make a big contribution to the reduction of production costs, and optimize the use of materials. A heavy-duty automotive lift system is composed of several sub-

components, including vertical columns, horizontal beams, lift platforms, reinforcement, and welded connections, and each component will have complex load paths during lift operation. It is undesirable to use simple analytical equations, which are still not sufficient to describe the stress concentration areas. When considering stress concentration areas, interactions between the individual parts must be considered [5] [6]. For this reason, FEA has become a key tool for investigating structural performance in lift systems, meeting industrial safety requirements, and producing economical, durable lift equipment suitable for long-term industrial use. What is often missing in the existing literature is a comprehensive parametric investigation. Most researchers analyze the structure under only one loading and report the maximum stress and deformation without investigating the effects of different capacities, structural member thicknesses, or material properties on the overall response of the structure [7]. This approach is not very informative for design and does not allow research results to be transferred to engineering practical situations with changing loading conditions for lift systems. Furthermore, many broad concept convergence studies are not included, and evaluations of the factor of safety and comparisons of different structural configurations are not provided, making the reported numerical results less reliable and reproducible. In repeated service cycles, critical regions may go unrecognized in areas of high stress concentration, leading to fatigue crack initiation and long-term structural degradation [8]. Industrial manufacturers often address these uncertainties by using conservative design principles, oversized structural components, and excess material to maintain an adequate safety factor. This, however, makes the structure more reliable but also brings drawbacks: higher manufacturing costs, heavier structures, higher transportation costs, and a more complicated installation process. Advanced numerical simulations are available, enabling lightweight, structurally efficient designs by analyzing stress distribution and corresponding behavior. However, no readily available, generically named CAD models or standard FE simulation procedures provide sufficient repeatable technical information for heavy-duty automotive lift systems [9]. Existing investigation methods do not combine generic structural modeling, static finite element analysis, parametric comparison, and safety evaluation in one step, which would allow researchers and industrial designers to reproduce them. Therefore, an economical and technically valid methodology is needed to provide important data on structural performance while the lifting equipment is in use and to allow future optimizations and design improvements [10].

The current study tackles these research drawbacks by creating a generic 3-D model of the heavy-duty four-post automotive lift system and conducting a detailed structural analysis and assessment using Finite Element Analysis. A realistic CAD model is built and loaded with a static load based on the rated lifting capacity; stress distribution and deformation behavior, equivalent strain, and overall structural stability are studied. Using this numerical methodology, one can visualize the regions of the lift structure where load bearing occurs and provide quan-

titative data on the lift structure's mechanical behavior under operational conditions.

This study is summarized by the following items, which are its major scope:

1. Capability building for a generic CAD model of a four-post high-load performance automotive lift system with realistic structural components for numerical analysis.
2. Structural analysis without motion = Finite Element Analysis & Computation of von Mises stress, equivalent strain, maximum total deformation, reaction forces, F.O.S.—on the basis of the maximum rated loading condition.
3. Parametric study to understand the effect of load-carrying capacity and thickness of the structural members on the distribution of stresses, their deformation, and the structural performance.
4. Critical stress concentration areas are mapped in the vicinity of beam-column interfaces and welded joints, identifying possible vulnerable areas and increasing structural design reliability.
5. Development of a cost-effective structural evaluation approach, based on FEA, yielding repeatable technical results that can be used for preliminary design validation, optimization analysis, and future development of a heavy-duty automotive lift system.

2. Background Work

As the need for safe, reliable, heavy-duty vehicle maintenance equipment grows, heavy-duty automotive lift systems have become increasingly advanced. Modern lifting systems should support commercial trucks or buses, construction vehicles, and industrial machinery, providing structural stability during repeated loading. Traditional design relied on analytical beam theory and empirical methods, with high safety factors to account for uncertainties in loading and material behavior. However, the growing complexity of welded assemblies, interconnected beams and columns, etc., has shown the shortcomings of the traditional analytical approach in predicting localized stress or deformation characteristics. Finally, the use of CAD and FEA has become a common practice for structural integrity analysis prior to the production of a prototype product compared to experimental testing, as it allows the simulation of the product to be designed with stresses, deformations, reaction forces, and the possibility of failure locations under realistic loads [5]. More recently, advances in computational mechanics have enabled numerical simulation methods to optimize structural members, minimize material use, and enhance the safety factor of lifting systems in general [6]. Advanced mesh convergence analysis, realistic boundary conditions, and material modelling have also been highlighted to enhance the predictive capabilities of the finite element method and, therefore, increase prediction reliability [7]. Although many new flight concepts have been introduced in the past few decades, most ongoing tests are limited to the hydraulic aspects of the lift mechanism, its kinematics, or the mechanical performance of the actuators, rather than a thorough evaluation of the

heavy-duty automotive lift framework structure [8]. Furthermore, comparative analyses across different load capacities and structural dimensions are limited; therefore, technical data to support preliminary design validation remain scarce [9]. Critical stress concentration regions (CCRs) near beam-column joints and at welded joints remain the dominant factors of structural vulnerability and require careful, detailed numerical study to ensure long-term operational safety and design reliability [10]. The growing need for reliable design assessment and optimization of heavy-duty auto lift systems also underscores the need for a method that provides repeatable FEA results with reduced complexity and cost. A comparative summary of the existing research, including the methodologies, principal findings, and identified limitations, is presented in Table 1.

Table 1. Summary of existing research on structural evaluation of automotive lift systems.

Ref.	Research Focus	Methodology	Key Findings	Identified Limitation
[5]	Structural behavior of lifting frames	Linear static FEA	Identified stress concentration near support members under rated load	Limited investigation of deformation and safety factor
[6]	CAD-integrated lift structure design	CAD modeling with numerical validation	Improved structural visualization and reduced development time	Parametric performance analysis not included
[7]	Finite element modeling techniques	Mesh refinement and convergence analysis	Demonstrated improved numerical accuracy with optimized mesh density	Application restricted to simplified structural models
[8]	Hydraulic and lifting mechanism evaluation	Mechanical and kinematic simulation	Enhanced lifting efficiency and operational performance	Structural framework analysis was not comprehensively performed
[9]	Static structural assessment of steel assemblies	Stress and deformation analysis	Established a relationship between applied load and structural response	Single loading condition considered
[10]	Welded joint structural reliability	Finite element stress evaluation	Beam-column interfaces identified as critical stress regions	Absence of load variation studies
[11]	High-load mechanical support structures	Static FEA with safety assessment	Factor of safety remained within acceptable engineering limits	Generic automotive lift model not developed
[12]	Steel frame optimization	Structural optimization using numerical methods	Material utilization is reduced while maintaining strength	Reaction force analysis omitted
[13]	Heavy-duty maintenance equipment	Computational structural analysis	Significant reduction in prototype development cost	No investigation of member thickness variation
[14]	Industrial lifting platform performance	CAD-FEA integrated approach	Accurate prediction of deformation and stress distribution	Equivalent strain evaluation not reported
[15]	Structural validation of mechanical support systems	Static finite element simulation	Numerical analysis improved preliminary design validation	Comprehensive parametric comparison absent

3. Generic Lift System Design

Automotive lift systems ensure the safe lifting of commercial vehicles or industrial equipment that do not affect the integrity of the load, even in heavy lifting conditions. The most important parts of these types of systems, which transfer the loading of the vehicle to the foundation, are the vertical support columns, the longitudinal beams and the cross members, lifting platforms, reinforcement plates, and

welded connections. For the ability of the lift to perform, the ability of individual components to handle the load is only part of the answer; the performance of the components in conveying the load is the other. A four-post lift configuration is chosen as an example to simulate because, in an automotive application, the configuration is generic such that using a consistent numerical evaluation methodology is possible with this configuration. Eliminates the need to depend on a special industrial design scheme to retain the essential features of the heavy-duty lift systems of proven design. This configuration reduces load variations across all components, thus enhancing the stability and boundary conditions definition for finite element analysis. From there, ONEWKB model elements are illustrated and constructed in a three-dimensional CAD and simulated with realistic dimension continuity and dimensions that emulate realistic load paths and stress transfer mechanisms. The design emphasis is on simplicity of structure, efficiency of calculations, and the potential for optimization in the future for various loading and member dimensions. The generic representation also allows for repeatable numerical investigation without the issues of geometric complexities that might impact the results. The von Mises stress, Equivalent strain, Maximum total deformation, Reaction forces, and Maximum factor of safety evaluations were very detailed for only the maximum rated loading condition, with no consideration of motion of the system, as the main focus of this research is the structural evaluation of the lift system in the stationary load-bearing frame. This type of modeling gives a common platform for parametric studies intended for further optimization by finite element tool-based computational analysis, and helps to pinpoint key stress concentration areas and preliminary validation of the structure for optimal design [15] [16].

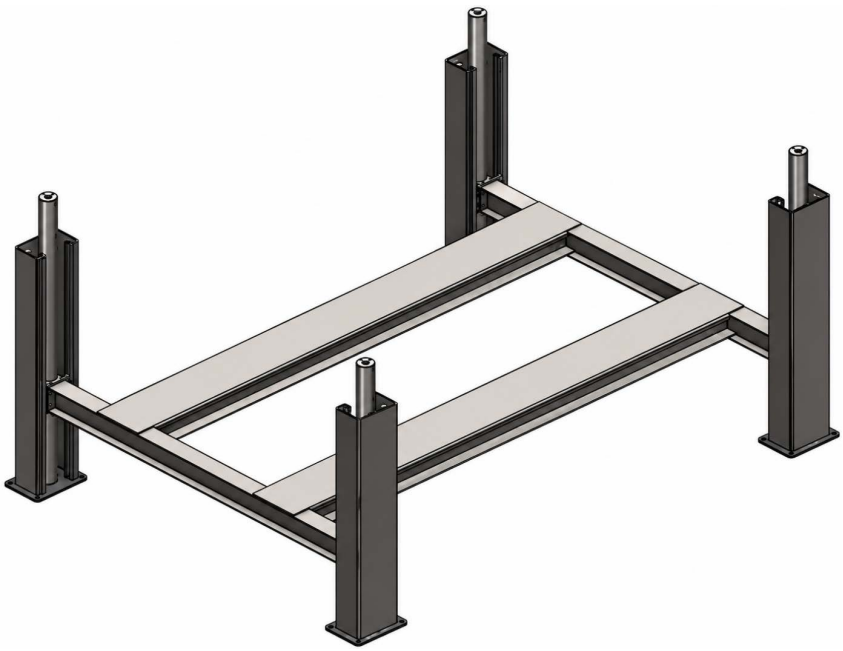


Figure 1. Conceptual representation of the generic four-post heavy-duty automotive lift system adopted for finite element structural evaluation.

Various lift system configurations of vehicles are considered, but for this study the simplified representation of the concept in **Figure 1** is chosen. Geometry retains this principle when it comes to load assessment, even though it makes it simpler to make use of commercial heavy-duty lifting systems without having to carry a number of secondaries like hydraulic cylinders, chains, and synchronizers. Once the load is placed upon the lifting platform, it passes under it, long and wide (side-to-side), through the horizontal/longitudinal beams before passing vertically through the four vertical support beams to the fixed foundation. This simple load path can be used to accurately calculate the propagation of the load and earthquake response under deformation, with lower computer calculation complexity. Moreover, using this standardized set-up, investigations of a similar geometry can be performed for loading capacities and thicknesses of the structural members, yielding reproducible technical results for initial design verification. For this generic figure, the configuration of the structure, material, and component representations are explained below, which correspond to an FEM design requirement for the figure, and the max. rated loading is used to evaluate the von Mises stress, equivalent strain, and max. deformation, reaction forces, and max. factor of safety for the figure [17] [18].

3.1. CAD-Based Structural Configuration

It is a representative 4-post lift structure, which is formed of 4 column heights, 2 vehicle platforms, and 4 column bases by four vertical columns, used for lifting. The configuration was chosen for its wide application in industrial manufacturing, its excellent load-distribution characteristics, and high operating stability in heavy commercial vehicles. The vertical columns act as the primary load-carrying girders; the horizontal columns evenly distribute the load within the structure; vehicle loads are transferred to the ground through fixed supports. Reinforcement plates are used to stiffen beam-column joints to reduce maximum stresses at detail locations. It was assumed that welded joints form a seamless joint, and a realistic representation of load transfer throughout the assembly is obtained.

The CAD model employs simple geometry, but contains features that are important to the mechanical purpose of the system in a real vehicle but are not overly complicated. Manufacturing and industrial work also adopt a large amount of rectangular hollow section steel as the structural member, which has an excellent strength-to-weight ratio, so that the rectangular hollow section steel is optimally designed with high efficiency for use as structural members. Stations for fixed loading do not feature any hydraulic cylinders, pulleys, cables, or moving mechanisms, so that only static loading conditions can be represented in the structural analysis. The simplified representation significantly reduces the calculation expenses, even if the accuracy of the calculation of the stress distribution and deformation characteristics, reaction forces, and structural reliability is not lost. The CAD model can then be put in a generic form so it can feed into finite element simulations and then be parametrically varied to simulate different thickness or load-carrying capacity changes of structural members [17] [18].

3.2. Structural Components and Material Representation

The structural frame comprises multiple interdependent components with varying functions and load-bearing capacities that can withstand the loadings imposed during heavy vehicle transit while ensuring dimensional stability and structural integrity. The four support columns carry vehicle loads to the supporting structure through the longitudinal girders. Extra cross beams provide additional lateral stiffness and prevent excessive loading-induced camber, and reinforcement plates are used as needed for greater stiffness at critical intersections. Welded joints join individual structural members to form a load-carrying assembly; they are an important factor in the overall stress distribution of the lift system.

The assumption in numerical analysis is that all structural elements have homogeneous mechanical properties in all directions, i.e., they are isotropic. Assumption for numerical analysis: All the structural elements have homogeneous mechanical properties in all directions, i.e. they are isotropic. Material properties have been assigned throughout the model to ensure a finite element calculation consistent with the model properties, and to build comparison parametric models. The simplified material representation eliminates uncertainties caused by material discontinuities while still maintaining the elastic response of the structural framework under the given loading conditions. Total deformation, reaction forces, Factor of safety, von Mises stress and equivalent strain are reasonable for a linear elastic approach in this investigation. To improve the reproducibility of the computational model, these parameters are based on typical material properties and can be compared directly across different loading conditions and structural configurations considered in the parametric analysis [19].

3.3. Design Considerations for Structural Evaluation

This generic lift system focuses on the need for a structural evaluation through finite element analysis and is not dedicated to operational motion analysis. Fixed boundary conditions are applied to the bottom of the support columns, and the uniformly distributed loading of the vehicles on the lifting platform represents the maximum rated operating conditions. However, in this case, the structural response is completely defined by the imposed load on the support structure, and it is possible to analyse specifically the regions of stress concentration and deformation characteristics. The location of beam-column connections, welded sections, and areas with sharp changes in geometry or abrupt changes in stiffness are given special consideration, as they are very likely to develop stress concentrations that may cause failure.

The adopted design procedure also allows a systematic parametric study, since the magnitudes of loads and the thickness of the concerned members are ensured to change systematically around the design, without changing the overall geometrical setup. This flexibility allows the boundary conditions and material properties to carry over to several different design scenarios. A computational framework is developed to obtain reproducible technical data on variations in von Mises stress,

equivalent strain, maximum total deformation, reaction force, and factor of safety under various working conditions. Furthermore, the simplified generic geometry serves as the starting point for subsequent optimization studies aimed at reducing weight, optimizing load distribution, and improving design reliability. This standardized modeling approach aligns with the goal of building a cost-effective, reproducible optimization methodology for automotive heavy-duty lift systems based on FEA, and with the concept for future structural optimization and industrial product validation [20].

4. Mathematical Modeling

A sufficient mathematical model of the relationship between the load applied to the vehicle and the supporting structure's reaction force should be used to evaluate the heavy-duty automotive lift system. To successfully define load transfer, reaction force, stress formation, response to static load, and structural safety, it is necessary to develop an FEA. In this study, a 4-post stationary structure system is assumed to study the arranged system. The distributed load applied to the lifting system is assumed to be applied over the entire spread area of the lifting system, while the vertical load-carrying columnar group and the load-carrying longitudinal columnar group constructed on the fixed foundation are assumed to be the distributed load carriers. The mathematical formulation uses the principles of static equilibrium, elasticity theory, and strength of materials to verify the numerical simulation results. The analytical model is applied directly to achieve the objectives of this research: von Mises stress, equivalent strain, Maximum total deformation, reaction forces, and factor of safety under the maximum rated loading condition. In addition, the formulation provides a standardized way to study the effects of load-carrying capacity and structural member thickness in a parametric study, yielding valid technical data for reuse in preliminary structural property validation, structural parameter optimization analysis, and the development of various automotive lift systems for heavy-duty trailer applications [21] [22].

Mathematical Framework for Structural Evaluation of Heavy-Duty Automotive Lift System

Input:

m, g, E, σy, L, A, I, t

Number of support columns (n = 4)

Output:

W, Ri, σa, σb, δ, σvm, εeq, FOS

Begin

1. Initialize structural parameters.
2. Compute vehicle load:

$$W = m \times g$$

3. Compute load per support column:

$$P = W/n$$

4. Determine reaction forces:

$$Ri = W/n$$

Verify ΣRi=W

5. Compute structural response:

Axial Stress:

$$\sigma a = P/A$$

Bending Stress:

$$\sigma b = \left(M \times y\right)/I$$

Maximum Deformation:

$$\delta = \left(P \times L^3\right)/\left(48 \times E \times I\right)$$

Equivalent Stress:

$$\sigma vm = \sqrt{\left[\left(\left(\sigma 1 - \sigma 2\right)^2 + \left(\sigma 2 - \sigma 3\right)^2 + \left(\sigma 3 - \sigma 1\right)^2\right)/2\right]}$$

Equivalent Strain:

$$\varepsilon eq = \sqrt{\left[\left(2/3\right)\left(\varepsilon 1^2 + \varepsilon 2^2 + \varepsilon 3^2\right)\right]}$$

Factor of Safety:

FOS = σy/σvm

6. Perform parametric analysis:

For different load capacities

Recalculate W, P, σa, σb, δ, σvm, εeq and FOS

End

For different structural member thicknesses

Update A and I

Recalculate σa, σb, δ, σvm, εeq and FOS

End

7. Evaluate structural safety:

If FOS > 2

Design = SAFE

Else

Design = OPTIMIZATION REQUIRED

End

8. Report:

W, Ri, σa, σb, δ, σvm, εeq, FOS

Critical Beam–Column and Welded Joint Stress Regions

End

The mathematical structure developed in this study is systematically implemented in a computational algorithm that merges static equilibrium, strength of materials, and finite element pre-processing into a single structural evaluation procedure. First, all material properties and geometric parameters are initialized, and the total vehicle load is then calculated, assuming it is distributed evenly. The load inertia and spring reaction forces, bending stresses, maximum deformations, equivalent von Mises stress, equivalent stress, and factor of safety were then determined using well-known analytical relations. In addition to providing theoretical estimates to check the FE results, the algorithm also systematically studies parametric changes in the thickness of the structural members and the load-car-

rying capacity, without changing the geometry or boundary conditions. This standardized computational workflow across operating scenarios can consistently capture key beam-column interfaces, stress concentrations at welded joints, and overall structural behavior. The RnL protocol is repeatable for further study, prevents computational inconsistencies, and provides a strong analytical foundation to validate preliminary design studies, optimize a heavy-duty automotive lift system, and develop a future heavy-duty automotive lift system through finite element computational techniques [23].

The model was a simplified mathematical model built from the analytical development of its elements and directly connected to the main goal of the research, which concerns the study using the finite element method. The reaction force equations are also used to determine static equilibrium in fixed supports, and the vehicle load is calculated and applied as the loading condition for the generic CAD model. The stress and deformation equations provide theoretical reference values for comparison with the obtained FEA results; conversely, the equations for equivalent stress, equivalent strain, and factor of safety are used for a comprehensive assessment of structural performance. In addition, the device described above can address the structural validity of the proposed parametric study by repeatedly changing the load-carrying capacity and thickness of structural members, thereby ensuring structural validity. Therefore, for the first design, the mathematical formulation is a low-cost, repeatable method for verifying the design, optimizing it for performance and durability, characterizing the transition regions between beams and columns, and continuing to develop heavy-duty automotive lifting systems through finite element computational analysis [24] [25].

5. Finite Element Methodology

Based on the generic four-post heavy-duty automotive lift system developed in Sections 3 and 4, the finite element approach in this paper provides a logical, organized numerical tool to estimate the structure's properties. This three-dimensional CAD model is the computational domain and encompasses the vertical columns of support, longitudinal beams, cross beams, lifting platforms, reinforcing plates, and any welds- the basic structural skeleton that supports the loads. Motion components like hydraulic cylinders, pulleys, and cables are not included because the overall objective of this research is to determine the structure of a machine or system when it is motionless and operating. The structure is modeled in three dimensions using finite element analysis, and it can simulate stress distribution, equivalent strain, deformation characteristics, and reaction forces when loaded to the maximum rated load. The structural steel is assumed to be homogeneous, isotropic, and linear elastic during the simulation. A uniformly distributed static load (VcmL), equivalent to the rated vehicle capacity, is applied to the lifting platforms and transferred to the four support columns via fixtures fixed to the bottom. Areas with high stress are meshed with a refined mesh around the beam-column interfaces and welded joints to increase accu-

racy. The displacement field is then calculated for the whole structure, and the equivalent von Mises stress, equivalent strain, maximum total deformation, reaction forces, and maximum equivalent safety are determined using the finite element solver. Furthermore, the developed computational model enables the proposed parametric study by varying load-carrying capacity while keeping the same geometry, material properties, and boundary conditions for different structural member thicknesses. It enables repeatable structural assessment, identifies critical load-bearing regions, and provides a cost-effective framework for initial structural validation, optimization analysis, and further development for heavy-duty automotive lift systems. The finite element mesh and applied loading configuration of the four-post lift model are illustrated in Figure 2. The adopted modeling parameters, their configurations, and their intended purposes are summarized in Table 2.

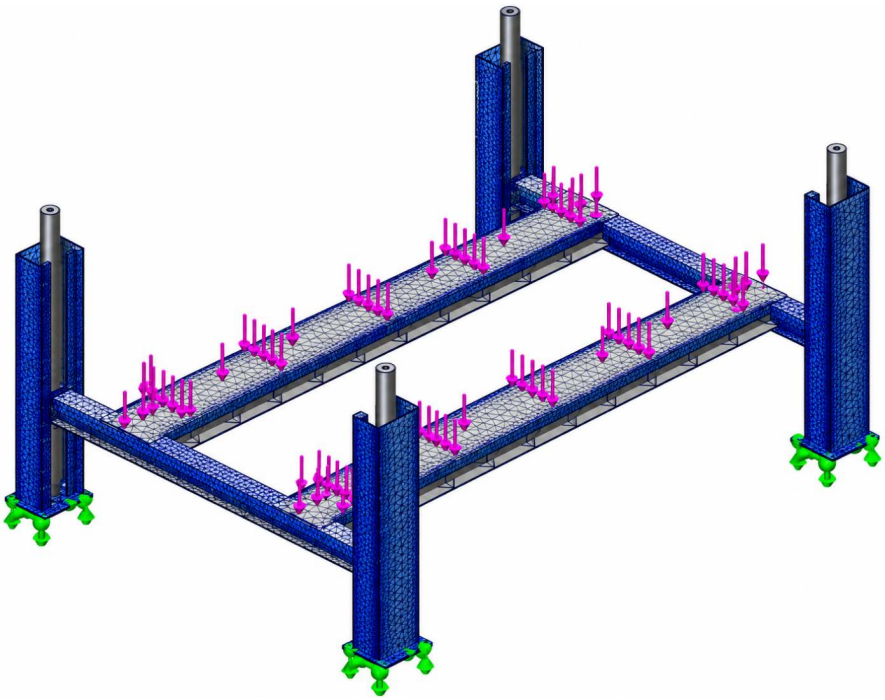


Figure 2. Finite element mesh and loading configuration of the four-post lift model.

Table 2. Finite element modeling parameters for structural evaluation.

S. No.	Analysis Parameter	Adopted Configuration	Purpose in Study	Expected Outcome
1	Structural Model	Generic Four-Post Automotive Lift	Standardized computational geometry	Repeatable numerical evaluation
2	Analysis Type	Linear Static Structural FEA	Stationary structural assessment	Stress and deformation prediction
3	CAD Representation	Three-Dimensional Solid Model	Accurate load path simulation	Realistic structural behavior
4	Material Model	Homogeneous, Isotropic Structural Steel	Consistent material response	Reliable elastic analysis

Continued

5	Loading Condition	Uniform Static Vehicle Load	Simulate maximum rated capacity	Uniform load distribution
6	Boundary Condition	Fixed Supports at Four Base Columns	Maintain structural equilibrium	Accurate reaction force computation
7	Mesh Strategy	Global Mesh with Local Refinement	Improve solution accuracy	Better stress localization
8	Critical Evaluation Zones	Beam-Column Interfaces and Welded Joints	Identify vulnerable regions	Stress concentration mapping
9	Primary Response Variables	von Mises Stress, Equivalent Strain, Total Deformation	Structural performance assessment	Structural integrity validation
10	Secondary Response Variables	Reaction Forces and Factor of Safety	Safety and stability evaluation	Load-bearing capability assessment
11	Parametric Variables	Load Capacity and Member Thickness	Comparative structural analysis	Design optimization support
12	Computational Objective	Preliminary Design Validation using FEA	Cost-effective structural evaluation	Repeatable technical data for future optimization

6. Boundary Conditions

Defining boundary conditions, loading constraints, and support assumptions for the structure's real operating conditions is key to the accuracy of any finite element analysis. In this work, the effects of the physics of the generic four-post heavy-duty lift system of an automobile, such as motion and hydraulic action, are neglected in favor of longitudinal steady force loading to evaluate the integrity of the four-post system. The base plates of the four vertical support columns are assumed to be fixed to the foundation, and all degrees of freedom of translation in both vertical and horizontal directions and all degrees of rotation are assumed to be fixed. This assumption requires the structure to transfer all structural loads related to distorting action to the base. For testing, a uniformly distributed static load that corresponds to the maximum rated lifting capacity is applied over the lifting platforms to simulate the loading of a heavy commercial vehicle. This eliminates localized point loads and provides a realistic loading case that represents how the load is shared through the longitudinal beams and cross members via reinforcement plates and vertical columns. When the structural steel is analyzed, it is assumed to be homogeneous, isotropic, and linearly elastic, and deformations and stress distribution can be evaluated within the elastic range. At the contact section of the welded joint, it is assumed that there is no relative displacement between adjacent structural members; the contact condition is perfect, and the characteristics of the structural members are tightly correlated with how the forces are transmitted. To improve numerical accuracy in regions with anticipated large geometric changes that produce stress concentrations, mesh refinement was used near beam-column interfaces and welded regions. The boundary conditions are stable and help specify equivalent stress, maximum total deformation, reaction forces, and factor-of-safety values, and they're applicable to the proposed parametric study with variations in load-carrying capacity and structural-component

thickness. The chosen loading and support conditions provide a standard, repeatable basis for preliminary experimental validation and optimization, as well as for further development of heavy-duty automotive lifting systems. The fixed-support boundary conditions and the distribution of applied loads across the four-post lift structure are illustrated in Figure 3.

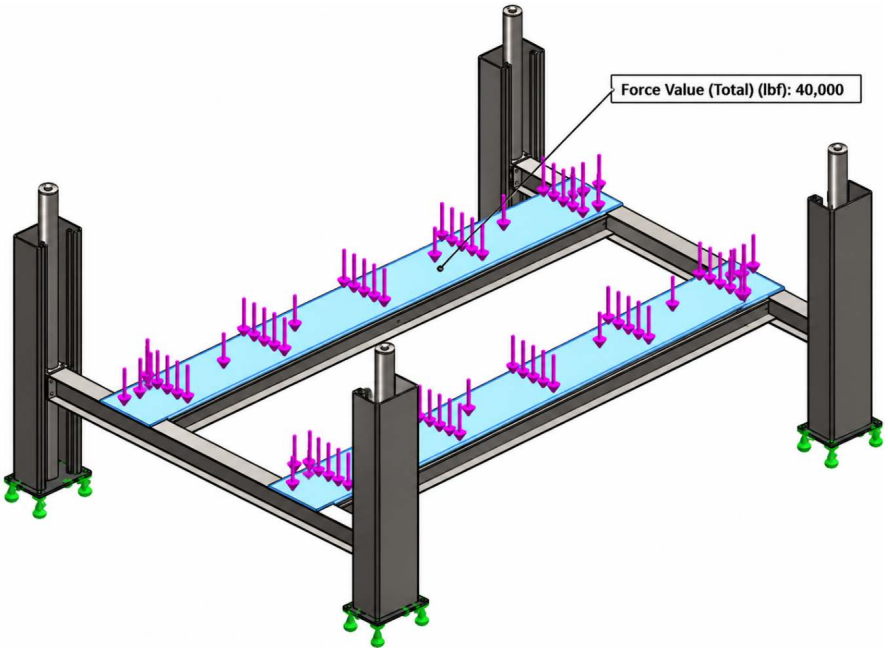


Figure 3. Boundary conditions and loading configuration of the generic four-post automotive lift system.

7. Static Structural Analysis

The static structural analysis assesses the load-bearing capability of the proposed generic four-post heavy-duty automotive lift system, its ability to resist loads, and whether the lift system is structurally sound when loaded to its maximum rated load. The analysis synthesizes the work in the sections above: the generic CAD model generated in Section 3, the mathematical modelling, and the FE methodology described in Sections 4 and 5, resulting in a complete analytical tool for structural assessment. The mathematical model includes the governing relationships for the loads acting on the vehicle, forces, bending stress, deformation, equivalent strain, and factor of safety; the three-dimensional CAD model consists of the primary load-bearing parts, support columns, longitudinal beam sections and cross members, lifting platforms, reinforcement plates, and welded interfaces. These analytical formulations provide reference values for theoretical analyses, which will be validated in finite element computations under the same loading and boundary conditions. The load is applied to the entire vehicle, and numerical calculations of stress and deformations are performed over the entire structural assembly, equally distributed among the lifting platforms. The reaction forces at the fixed supports are calculated and stored, along with the displacement field from

the finite element solver, from which the von Mises equivalent stress, equivalent strain field, maximum total deformation, and the factor of safety (which is the ratio of the maximum equivalent stress to the yield strength of the material of the structure) can be determined. Another advantage of the methodology is the ability to make controlled variations in structural behavior by changing "load capacity" and "member thickness" while keeping the overall geometric configuration unchanged. The focus will be on interfaces between the beam and the column, and welded connections, since abrupt stiffness and/or shape changes are assumed in these areas, where local stress concentrations are most likely to occur. Finally, the proposed method provides a quantitative assessment of the expected mechanical response, the stability and safety of the lift system, and its final static structural analysis confirms the proposed cost-effective finite element framework for preliminary design assessment and further structural optimisation. These are obtained through mathematical modelling and checked against numerical predictions using finite element calculations, ensuring consistency between theoretical and numerical results. The structure's geometry is defined by a generic CAD model, the mathematical form defines the load and stress relationships governing the structure, and the finite element methodology yields detailed stress and/or deformation fields under identical loading conditions. The technical data set obtained through this methodical approach can be used meaningfully for future optimization of the automotive heavy-duty lift system, preliminary design validation, and parametric analysis of the technical data and factor of safety, and can be reused to test the correct basic function. The resulting von Mises stress distribution is presented in Figure 4, while the total displacement distribution under the applied loading condition is shown in Figure 5. The corresponding equivalent strain distribution is illustrated in Figure 6. The numerical outcomes of the static finite element analysis under proof loading are summarized in Table 3.

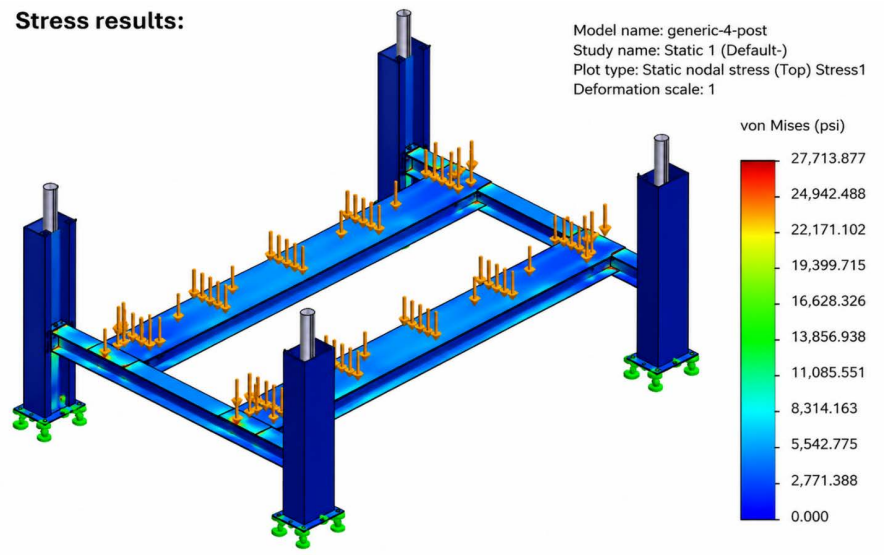


Figure 4. von Mises stress distribution obtained from the static FEA of the four-post lift.

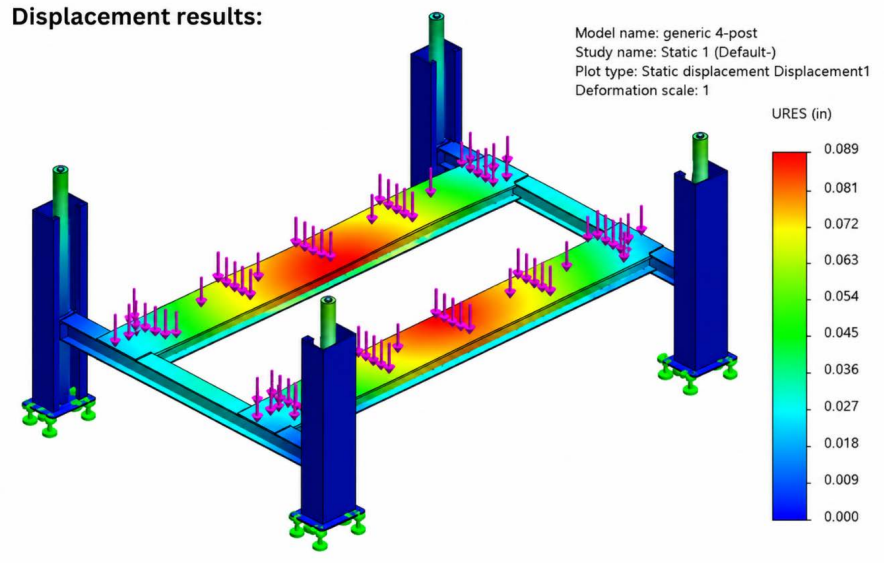


Figure 5. Total displacement distribution of the four-post lift under the applied loading condition.

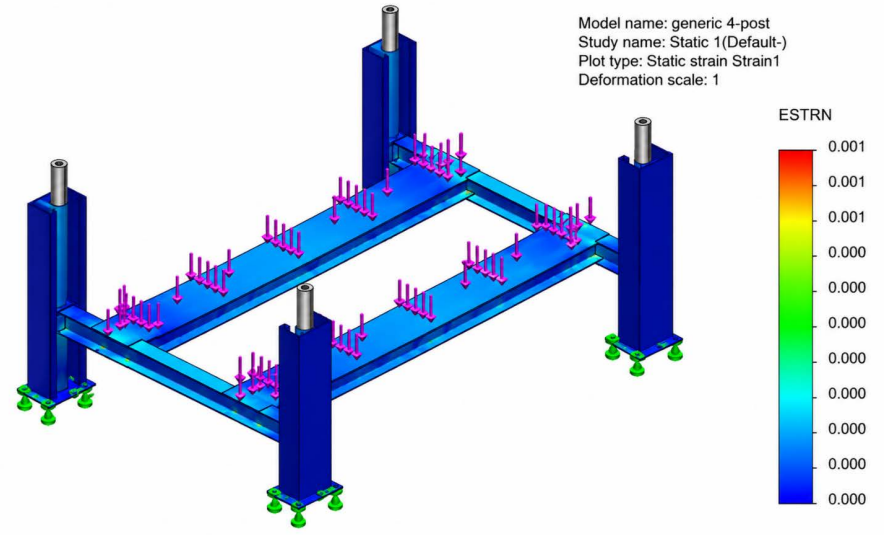


Figure 6. Equivalent strain distribution obtained from the static finite element analysis.

Table 3. Static FEA results for the four-post automotive lift under proof loading.

S. No.	Analysis Parameter	Derived From	Unit	Obtained Result	Structural Interpretation
1	Total lift capacity	product specification	lbf	40,000	Rated lift capacity
2	Total Applied Load	ALI Proof Load Requirement	lbf	60,000	150% of rated lift capacity
3	Load per Support Column	Load Distribution Equation	lbf	15,000	Uniform load transfer to four columns
4	Maximum von Mises Stress	Finite Element Solution	psi	27,713	Below ASTM A36 yield strength (36,000 psi)
5	Maximum Equivalent Strain	Finite Element Solution	in/in	0.001	Elastic structural behavior
6	Maximum Total Deformation	Displacement Field	in	0.089	Within acceptable service limits

Continued

7	Maximum Reaction Force per Column	Fixed Support Calculation	lbf	15,000	Balanced structural equilibrium
8	Maximum Beam Bending Stress	Mathematical & FEA Validation	psi	27,713	Safe beam performance under proof load
9	Minimum Factor of Safety	Yield Strength/von Mises Stress	-	1.30	Meets proof-load structural requirements
10	Critical Stress Location	Stress Contour Analysis	-	Beam-to-Column Connection Region	Primary region for design review
11	Critical Deformation Location	Displacement Contour Analysis	-	Mid-span of Runway/Track	Maximum elastic displacement
12	Overall Structural Status	Combined FEA& Analytical Assessment	-	Safe	Suitable for preliminary design validation

8. Discussion

The present study shows that FEA is an effective tool for analyzing the structural performance of a generic four-post heavy-duty automotive lift system, but several limitations must be considered when interpreting the results. This computational model is based on a largely stationary structural configuration (without hydraulic cylinders, lifting systems, wires, pulleys, locking systems, etc., which can affect actual load-transfer characteristics during lifting operations). The structural analysis assumes homogeneous, isotropic, linear-elastic material properties, while in reality structural steel components may behave nonlinearly, undergo plastic deformation, have residual stresses, exhibit imperfect manufacturing, and show varying material properties under extreme loads. Likewise, the assumption of "perfect welded joints" implies no weld defects, no heat-affected zone, and no fatigue crack initiation, all of which can significantly affect the structure's long-term durability. It is assumed that the applied loading condition is a uniform static load equal to the maximum rated vehicle capacity and that, in reality, lifting operations are often unevenly loaded, dynamically stressed, vibrating, subjected to repeated loading during engagement cycles, and subject to additional stress concentrations. In addition, the adopted boundary conditions assume perfectly fixed column bases, but foundation flexibility and installation tolerances can change the actual structural response. Several geometric details, which are not necessarily identical to those used in commercially made lift systems, are deliberately omitted from the CAD model and the numerical calculations to establish a standard computational framework and keep the calculations as simple as possible. While the mathematical model developed in this investigation is used to theoretically validate reaction forces, stress distribution, stress-strain behavior, and factor of safety, it is not included in the scope of this experimental verification, which involves full-scale physical testing of the developed model. Therefore, the results presented here are preliminary and intended for comparison and optimization studies, not for complete industrial certification. Despite these drawbacks, the methodology used to select critical interfaces of beam-column structures, regions of stress concentra-

tion in welded small structures, and overall structural integrity can be considered reliable and repeatable and serves as a basis for future scientific numerical and experimental investigations.

Advanced numerical and experimental investigations of heavy-duty automotive lift systems can take many forms based on the methodology presented in this work and will be offered here. This study does not include nonlinear material models, such as yielding, plastic deformation, strain hardening, and progressive failure mechanisms under overload conditions, but future research could include them to improve structural predictions. By considering operational structural behavior in terms of vehicle movement, impact loading, vibration effects, and transient loading conditions, a more realistic representation of operational behavior can be achieved than the present static evaluation approach. Another field of interest is fatigue life evaluation of beam-column interfaces and welded joints under repeated static load cycles, which involve continuous stress variation during industrial service and are known to cause structural deterioration over time. In addition, the developed generic CAD model can be expanded to incorporate functional components such as hydraulic actuators, synchronization mechanisms, locking assemblies, etc., to analyse how structural performance relates to lifting actions. Topology optimization and size optimization techniques can also be incorporated with finite element analysis to reduce the weight of the resulting structure while maintaining optimum stress distribution and factor of safety, thereby improving material utilization and manufacturing efficiency. Further improvements in computational accuracy and simulation time for complex structural assemblies of these components could be achieved through advanced mesh-adaptation algorithms and multi-objective optimization approaches. Experimental verification, such as with strain gauges, displacement sensors, and full-scale load testing, would increase confidence in numerical predictions and enable direct comparison of analytical, numerical, and experimental results. Adding high-strength alloys, composite reinforcement materials, and 3D printing technology could help create lightweight yet structurally sound lift systems for future auto maintenance shops. In addition, digital twin technologies and machine learning surrogate models can be combined with a finite element model to facilitate real-time structural health monitoring, condition prediction and automatic structural optimization. These potential future research pathways can transform the proposed cost-effective finite element methodology into a comprehensive intelligent design system capable of supporting preliminary validation, structural design optimisation and life cycle assessment, along with performance optimisation of heavy-duty automotive lifting systems under different industrial conditions.

9. Conclusion

Using finite element analysis (FEA), the base material in question was used for the 4-post lift construction. The total rated lift capacity of the product is 40,000 lbf, and the structure was evaluated at a proof load condition equivalent to 150% of

the rated lift capacity: 60,000 lbf applied across the runways/tracks. The analysis was conducted for the purposes of assessing the structural performance and meeting the criteria as per Automotive Lift Institute (ALI) requirements. The FEA results showed the highest von Mises stress to be 27,713 psi (27.7 ksi), and the highest displacement was 0.089 in. The maximum calculated stress is still lower than the yield strength of ASTM A36 steel, 36 ksi, thus meeting the criteria for yielding under the proof load applied. In addition, the relatively small calculated displacement indicates good stiffness and load-bearing capacity of the structure. Given the above result, it can be seen that the 4-post lift structure is able to resist its required proof load, 150% of the rating, or 60,000 lbf, at satisfactory stress and deflection levels when considering its rating for a lift, or capacity, of 40,000 lbf. Thus, based on the analysis data, it is seen that sufficient structural integrity and load-bearing capacity are suggested, which relates to the intended use of the product, and thus also to the structure's requirements according to the desired ALI classification. This will, in turn, allow the FEA to conclude whether or not the lift is structurally capable of enduring the overload condition it was designed for.

Conflicts of Interest

The author declares no conflicts of interest regarding the publication of this paper.

References

[1] Cook, R.D. (2007) Concepts and Applications of Finite Element Analysis. John Wiley & Sons.

[2] Zienkiewicz, O.C., Taylor, R.L. and Zhu, J.Z. (2005) The Finite Element Method: Its Basis and Fundamentals. Butterworth-Heinemann.

[3] Bathe, K.J. (1996) Finite Element Procedures. Prentice Hall.

[4] Finger, S. and Dixon, J.R. (1989) A Review of Research in Mechanical Engineering Design. Part II: Representations, Analysis, and Design for the Life Cycle. *Research in Engineering Design*, **1**, 121-137. https://doi.org/10.1007/bf01580205

[5] Shigley, J.E., Mitchell, L.D. and Saunders, H. (1985) Mechanical Engineering Design (4th ed.). *Journal of Mechanisms*, *Transmissions*, *and Automation in Design*, **107**, 145.

[6] Rao, S.S. (2010) The Finite Element Method in Engineering. Elsevier.

[7] Rencis, J.J. and Grandin Jr., H.T. (2021) A General Structured Procedure to Solve Machine Design Problems. 2021, *the ASEE Virtual Annual Conference Content Access*, 26-29 July 2021.

[8] Belytschko, T., Liu, W.K., Moran, B. and Elkhodary, K. (2014) Nonlinear Finite Elements for Continua and Structures. John Wiley & Sons.

[9] Chandrupatla, T., Chandrupatla, T.R. and Belegundu, A. (2021) Introduction to Finite Elements in Engineering. Cambridge University Press.

[10] Juvinall, R.C. and Marshek, K.M. (2019) Fundamentals of Machine Component Design. John Wiley & Sons.

[11] Fung, Y.C., Tong, P. and Chen, X. (2017) Classical and Computational Solid Mechanics. 2nd Edition, World Scientific.

[12] Espinosa Maldonado, A.A. and Dolovich, A.T. (2026) A New Twist on Torsion: A

Connection between Vector Calculus and Mechanics of Materials. *International Journal of Mechanical Engineering Education*, **54**, 153-172. https://doi.org/10.1177/03064190241286667

[13] Hibbeler, R.C. and Tan, K.H. (2006) Structural Analysis. Pearson Prentice Hall.

[14] Reddy, J.N. (2002) An Introduction to the Finite Element Method. In: Rai, S.N., Ramana, D.V. and Manglik, A., Eds., *Dynamics of Earth's Fluid System*, CRC Press, 199-226. https://doi.org/10.1201/9781003762133-22

[15] Bower, A.F. (2025) Applied Mechanics of Solids. CRC Press. https://doi.org/10.1201/9781003617280

[16] Timoshenko, S.P. and Gere, J.M. (2012) Theory of Elastic Stability. Courier Corporation.

[17] Shackelford, J.F. (2016) Introduction to Materials Science for Engineers. Pearson, 687.

[18] Khonsari, M.M. and Amiri, M. (2012) Introduction to Thermodynamics of Mechanical Fatigue. CRC Press.

[19] Youssef, H.A., Ahmed, M.H. and El-Hofy, H.A. (2023) Manufacturing Technology: Materials, Processes, and Equipment. CRC Press. https://doi.org/10.1201/9781003373209

[20] Hwu, C. (2024) Mechanics of Laminated Composite Structures. CRC Press. https://doi.org/10.1201/9781003470465

[21] Callister Jr., W.D. and Rethwisch, D.G. (2020) Materials Science and Engineering: An Introduction. John Wiley & Sons.

[22] Saad, Y. (2003) Iterative Methods for Sparse Linear Systems. Society for Industrial and Applied Mathematics.

[23] Johnson, K.L. (1987) Contact Mechanics. Cambridge University Press.

[24] Fish, J. and Belytschko, T. (2007) A First Course in Finite Elements. Vol. 1, Wiley. https://doi.org/10.1002/9780470510858

[25] Crisfield, M.A. (1997) Non-Linear Finite Element Analysis of Solids and Structures: Advanced Topics. John Wiley & Sons, Inc.