JOURNAL OF MECHANICAL ENGINEERING, SCIENCE, AND INNOVATION

ITATS

Research Article

Innovative Structural Evaluation of a Dual-Function Cane for Elderly Mobility: A Finite Element Analysis Approach Using SolidWorks

Journal of Mechanical Engineering, Science, and Innovation
e-ISSN: 2776-3536
2025, Vol. 5, No. 1
DOI: 10.31284/j.jmesi.2025.v5i1.7649
ejurnal.itats.ac.id/jmesi

Hadad Rafana Mugandy[1], Apip Amrullah[1*], Ma'ruf[1], and Rizal Mahmud[2]

[1] Departmen of Mechanical Engineering, Faculty of Engineering, Lambung Mangkurat University, South Kalimantan, Indonesia.
[2] Teknologi Transportasi, National Research and Innovation Agency (BRIN), Indonesia.

Corresponding author:
Apip Amrullah
Department of Mechanical Engineering, Faculty of Engineering, Lambung Mangkurat University, Indonesia
Email: apip.amrullah@ulm.ac.id

Abstract

The increasing global elderly population highlights the urgent need for safe, ergonomic mobility aids to enhance independent living. This study presents a structural evaluation of the NeoMossa multifunctional cane, which integrates an umbrella mechanism, using Finite Element Method (FEM) simulations in SolidWorks. The cane was assessed under static loading conditions of 300 N, 500 N, and 700 N, with key parameters including Von Mises stress, strain, displacement, and factor of safety analyzed. The results show that the maximum stress reached 87.01 MPa and the minimum factor of safety was 1.022 at the highest load of 700 N, indicating that the structure remains within safe operational limits. Displacement and strain values also increased proportionally with the load but remained within acceptable thresholds without causing structural damage. Although the factor of safety decreased with higher loads, the NeoMossa cane maintained mechanical integrity throughout the tested range. The study demonstrates that integrating multifunctional features into mobility aids is feasible without compromising structural reliability. To ensure full validation, future experimental testing is recommended to complement simulation outcomes and verify real-world performance, providing a comprehensive foundation for the design of innovative, safe mobility aids for elderly users.

Keywords: *Mobility aid, elderly safety, finite element method, structure analysis, multifunctional design*

Received: April 20, 2025; Received in revised: April 29, 2025; Accepted: April 29, 2025
Handling Editor: Naili Saidatin

INTRODUCTION

The aging population is increasing globally, which has driven a surge in research and development of assistive mobility technologies intended to support the independence and quality of life of older adults [1]. Age-related physiological changes particularly in balance, musculoskeletal strength, and coordination contribute to reduced mobility and elevate the risk of falls. Consequently, assistive devices such as walking canes, which redistribute weight and improve stability, are critical for maintaining user confidence during ambulation and for reducing the likelihood of injury [2]. Studies have consistently underscored the importance of ergonomic design principles that consider both user comfort and safety, as evidenced by research exploring digital human modeling and ergonomic validation [3]

In recent years, technological innovations have extended the traditional functionality of walking aids. New developments include multifunctional designs that integrate additional features, such as umbrella mechanisms, to address environmental challenges. The NeoMossa cane prototype serves as a notable example by merging mobility support with weather protection, thereby enhancing overall usability in various conditions[4]. The expansion of assistive technology into multifunctional domains is also backed by the growing body of literature on mobile and wearable technologies that prioritize both physical support and additional utility, such as sensor integration and robotics [1], [5]. This integration not only responds to the evolving needs of an aging demographic but also reflects a broader trend in gerontechnology to provide holistic solutions that combine mobility support with other essential daily living functions [6].

Despite the promise of multifunctional devices like the NeoMossa cane, significant challenges remain in ensuring that additional functionalities do not compromise structural safety. One such challenge is the long-term durability of mechanical joints in assistive devices, which can be prone to failure over time. Mechanical joints, especially in walking aids, experience high-stress concentrations, and their failure could compromise the safety and functionality of the device. Mechanical joints play a crucial role in various structures and systems, ranging from human anatomy to engineered devices. Understanding the behavior, design, and implications of mechanical joints is essential for ensuring structural integrity, especially in systems subjected to dynamic loads and stress conditions. A fundamental aspect of mechanical joints in both biological and artificial structures is their susceptibility to failure under repetitive loading. Joints, particularly those in mechanical systems, experience wear and tear due to factors like impact forces and material fatigue. For example, studies have highlighted that clearances present in kinematic joints can lead to serious impacts, resulting in wear and potential degradation of overall system performance [7], [8]. This phenomenon is echoed in human joints, where excessive mechanical loading can lead to osteoarthritic changes due to altered load distribution within the joint [9]. Therefore, careful consideration of joint design is crucial to mitigate these challenges and ensure longevity and reliability.

Previous studies have highlighted joint failures as a critical issue in the longevity of mobility aids, such as canes and walkers, due to repetitive loading and material fatigue [8]. For example, studies have found that improper joint design or inadequate material selection often leads to catastrophic failures in these devices after prolonged use under typical loading conditions. Therefore, assessing the mechanical integrity of joints in multifunctional devices like the NeoMossa cane is crucial to ensuring its long-term reliability and safety for elderly users.

Thorough structural validation is required to guarantee that devices can withstand the complex, variable loading conditions frequently encountered by elderly users. The Finite Element Method (FEM) has become a cornerstone for such validation efforts. Simulation techniques using FEM, particularly when implemented via SolidWorks

Simulation tools, allow researchers to assess critical parameters such as stress, strain, displacement, and safety factors under realistic user-based loading scenarios [4]. The utility of FEM in engineering analysis is well-documented, with studies demonstrating its efficacy in optimizing the design and predicting failure points in various assistive technologies [7]. Despite the promise of multifunctional devices like the NeoMossa cane, significant challenges remain in ensuring that additional functionalities do not compromise structural safety. Thorough structural validation is required to guarantee that devices can withstand the complex, variable loading conditions frequently encountered by elderly users. The Finite Element Method (FEM) has become a cornerstone for such validation efforts. Simulation techniques using FEM, particularly when implemented via SolidWorks Simulation tools, allow researchers to assess critical parameters such as stress, strain, displacement, and safety factors under realistic user-based loading scenarios [4]. The utility of FEM in engineering analysis is well-documented, with studies demonstrating its efficacy in optimizing the design and predicting failure points in various assistive technologies [10].

Furthermore, a dual approach that bridges user-centric ergonomic design with rigorous simulation-based mechanical evaluation is essential for optimizing assistive devices for the elderly. While previous studies have focused primarily on either design aesthetics or user comfort [2], [3], recent research advocates for integrated models that concurrently address structural integrity and ergonomic functionality [4], [11]. Such integrative methods not only enhance the overall safety and performance of devices like multifunctional canes but also pave the way for innovative designs that can be rapidly validated through computational methods prior to full-scale production [12]. This comprehensive strategy ensures that emerging assistive technologies are both reliable and appropriately tailored to the diverse needs of older adults, balancing advanced design features with user safety.

This study aims to evaluate the structural safety of the NeoMossa multifunctional cane for elderly mobility by employing the Finite Element Method via SolidWorks software. The novelty lies in the integration of umbrella functionality into the cane while ensuring its mechanical integrity under typical loading conditions. The research justifies the hypothesis that simulation-based analysis can preemptively identify critical stress points, guiding design improvements and ensuring product safety. The scope encompasses CAD modeling, simulation under varying load conditions, and material specification, ultimately providing a validated design framework for multifunctional assistive devices.

METHODS AND ANALYSIS

Design and Modeling

The research began with the conceptualization and design of the NeoMossa cane (Figure 1), a multifunctional assistive device intended for elderly users, integrating an umbrella mechanism within a traditional walking cane structure.

The design was created using SolidWorks Computer-Aided Design (CAD) software, focusing on both aesthetic and functional aspects, including ease of deployment for the umbrella and structural balance for mobility. The model was constructed to accommodate ergonomic standards and lightweight material usage to enhance user comfort and practicality.

As part of the approach used in this study, it is important to note that the developed design was based on simulations using SolidWorks software and finite element method (FEM) analysis. While these simulations provide valuable insights into the design's performance, it is essential to acknowledge that the results obtained have limitations, particularly due to their reliance on assumed parameters and conditions within the mo-

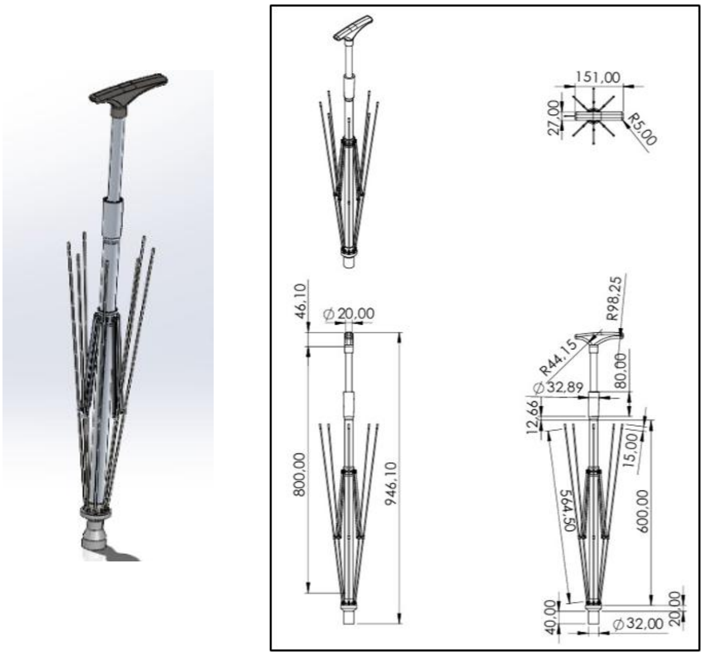

Figure 1. 3D Model and dimensional design of the NeoMossa cane prototype

del. Therefore, further validation of the design through experimental testing is strongly recommended to ensure the accuracy and reliability of the simulation results. Experimental testing will provide more realistic data and allow for the assessment of the design's performance under real-world conditions, which cannot be fully predicted by computer simulations alone.

Material Selection and Specification

Material selection was guided by mechanical requirements, user comfort, and manufacturing feasibility (Table 1-3). The main shaft and umbrella components were designed using Aluminum Alloy 3003, valued for its high strength-to-weight ratio and corrosion resistance. The handle was constructed from polycarbonate (PC) due to its high mechanical durability and tactile comfort. Natural rubber was used for grips and contact surfaces to improve stability and anti-slip properties. Each material's mechanical parameters such as Young's Modulus, Poisson's Ratio, Density, and Tensile Strength were sourced from standardized engineering databases and incorporated into the simulation environment.

The support and grip layers are made of natural rubber, which provides anti-slip grip, vibration absorption, and abrasion resistance, thus enhancing user stability and comfort. The shaft, runner, locking mechanism, and umbrella components are made from Aluminum Alloy 3003 due to its adequate structural strength, corrosion resistance, and lightweight nature, making it cost-effective for manufacturing.

Table 1. Specifications of PC (Polycarbonate) plastic material

Parameter	Specification
Elastic Modulus	2.32×10^9 N/m^2
Poisson's Ratio	0.3912
Shear Modulus	8.291×10^8 N/m^2
Mass Density	1190 kg/m^3
Tensile Strength	6.27×10^7 N/m^2
Thermal Conductivity	0.189 W/m·K
Specific Heat	1535 J/kg·K

Table 2. Specifications of natural rubber material

Parameter	Specification
Elastic Modulus	1.00×10^4 N/m^2
Poisson's Ratio	0.45
Mass Density	960 kg/m^3
Tensile Strength	2.00×10^7 N/m^2
Yield Strength	1.50×10^7 N/m^2

Table 3. Specifications of Aluminum Alloy 3003

Parameter	Specification
Density	2.73 g/cm^3
Mass	8.10 kg/m^3 *(Note: likely an error, needs clarification)*
Area	2,040,280 mm^2
Volume	2,968,600 mm^3
Yield Strength	124 MPa
Ultimate Tensile Strength	131 MPa
Young's Modulus	69 GPa
Poisson's Ratio	0.33
Shear Modulus	25.94 GPa

Table 4. Simulation parameters for modulation type effects

Parameter	Specification
Base Area	314.159 mm^2
Volume	552,346.69 mm^3
Force (F)	500 N
Height	18,111.21 mm
Mass	1,470.64 g
Elongation	0.01 mm

Finite Element Analysis Setup

The structural integrity of the cane was evaluated using the Finite Element Method (FEM) via the SolidWorks Simulation module. The meshing process divided the 3D model into discrete elements to simulate real-world physical behavior under load. Boundary conditions were defined based on realistic use: the base of the cane (ferrule) was fixed, and a vertical load was applied to represent the user's body weight. Three loading conditions were tested 300 N, 500 N, and 700 N representing varying user weights. Key outputs included stress (Von Mises), strain, displacement, and Factor of Safety (FoS). Simulation parameters were standardized for all trials to ensure comparative analysis. Table 4 presents the fixed parameters and specifications used in the simulation.

Simulation and Data Collection

Each simulation was executed individually for the three loading conditions. The results were recorded for maximum and minimum values across all stress, strain, and displacement metrics. The Factor of Safety was computed for each scenario to determine the device's structural reliability. Color-coded visual outputs were used to identify areas of concern, such as potential failure zones or excessive deformation. Comparative charts were generated to illustrate the correlation between applied loads and structural responses, providing insight into the safety margin and operational durability of the NeoMossa cane. The findings served as the basis for design validation and future improvements.

RESULTS AND DISCUSSIONS

Structural Response Analysis under Different Load

The structural integrity of the NeoMossa cane was evaluated using Finite Element Method (FEM) simulations in SolidWorks under three different loading conditions: 300 N, 500 N, and 700 N. The maximum Von Mises stress values observed were 32.5 MPa, 62.04 MPa, and 87,01MPa, respectively (Figure 2). Stress concentrations were notably visible at the ferrule and umbrella tip critical points subjected to peak mechanical loads. The simulation results exhibited a dominant color range from blue to green, signifying that the stress levels across most of the cane structure remained safely within the material yield strength.

The finite element analysis (FEA) of the NeoMossa cane was performed to assess its internal stress distribution and identify regions vulnerable to structural failure under varying load conditions. Figure 1 presents the Von Mises stress simulations under three different loading scenarios: 300 N (Fig. 2(a)), 500 N (Fig. 1b), and 700 N (Fig. 1c). These analyses provide insight into the mechanical behavior of the cane and highlight the critical areas where stress is most likely to concentrate. In the 300 N load scenario (Fig. 1a), the maximum Von Mises stress is approximately 3.25×10^7 N/m^2, with the most significant stress concentration observed at the joints connecting the cane's shaft to the supporting elements. While the stress levels are relatively modest, this localized concentration underscores the importance of joint design in controlling stress flow and minimizing potential failure points. Similar patterns have been identified in earlier studies, such as those by Yokoyama et al. [13] and Wang et al. [14], where mechanical joints, particularly bolted connections, were found to be prone to stress accumulation.

At an increased load of 500 N (Fig. 2(b)), the stress concentration intensifies, with the maximum stress reaching approximately 6.20×10^7 N/m^2. Despite the higher load, the structure remains within the elastic deformation range, indicating that the material still performs reliably under standard use conditions. The persistence of stress localization at the joints emphasizes the repeatability of the stress distribution pattern, while the linear increase in stress corresponds with findings from prior research on elastic behavior in load-bearing structures [15], [16]. These results support the conclusion that the current design is sufficient for average user weights without significant risk of structural failure.

Under the high-load condition of 700 N (Fig. 2(c)), the maximum Von Mises stress escalates to approximately 8.70×10^7 N/m^2. This value approaches the yield strength of the polycarbonate material used, thereby narrowing the safety margin. Stress remains concentrated at the mechanical joints and load-bearing interfaces, confirming these as the most critical regions for potential failure. As observed in similar investigations Yokoyama et al., [13] and Wang et al.,[14], such stress localization under repeated or dynamic loading could lead to plastic deformation or fatigue-related issues over time. Thus, the results advocate for structural reinforcement of the joints if the cane is to be used under higher loading conditions.

This trend is further reinforced by the data shown in the accompanying load-stress graph (Figure 3), where maximum Von Mises stress increases linearly with applied loadfrom approximately 32.5 MPa at 300 N to 62.04 MPa at 500 N, and 87,01MPa at 700 N while minimum stress remains nearly constant. This behavior confirms that although the cane safely supports typical user loads, its critical joint regions may require reinforcement for safe use under heavier or dynamic loading conditions.

The analysis of the NeoMossa cane's strain behavior under varying loads offers insights into elastic deformation that are consistent with classical mechanical principles such as Hooke's law [17]. In Fig. 4(a), when a load of 300 N is applied, the strain is primarily localized near the shaft-support junctions where load transfer causes the highest stresses, and the low maximum strain (≈0.018) indicates minor elastic deformation. This behavior is in accordance with the expectation that, under low loads, the elastic response of the material is limited, and the deformation remains within its elastic range [18].

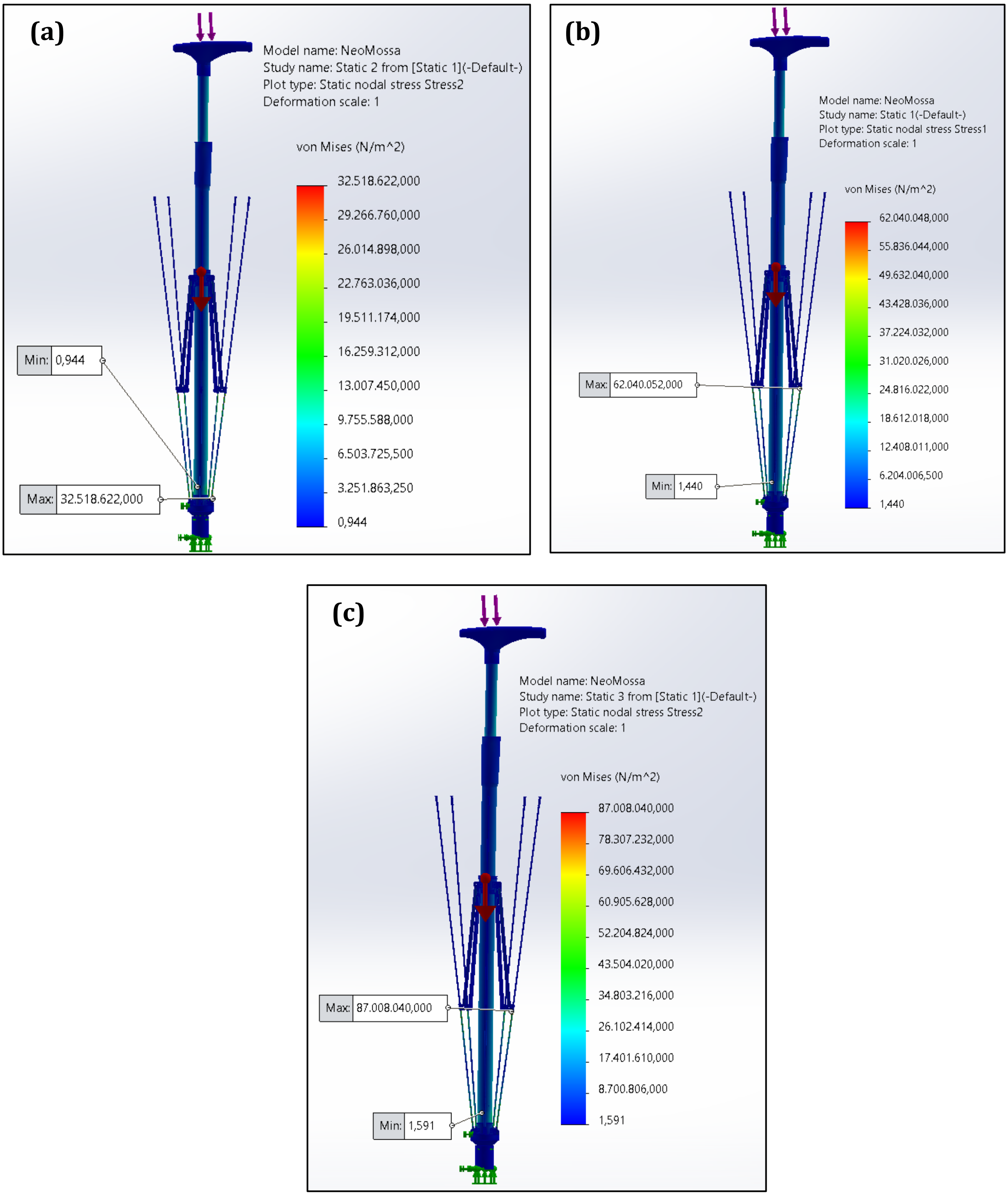

Figure 2. Von Mises stress simulation of NeoMossa cane at (a) 300 N, (b) 500 N, and (c) 700 N

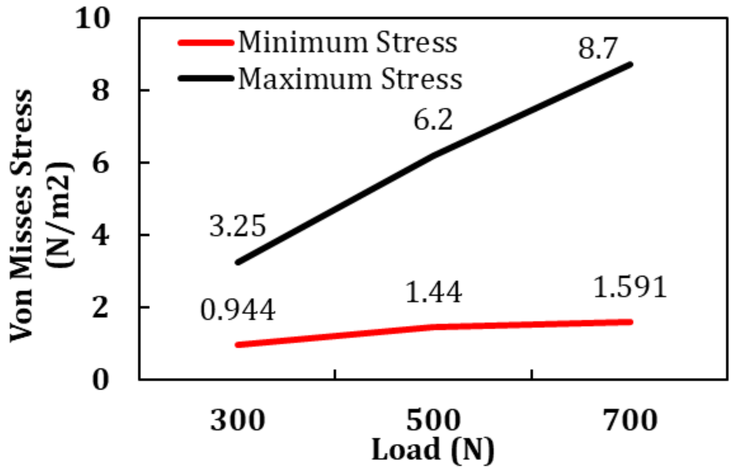

Figure 3. Load vs. Von Mises Stress of the NeoMossa Cane

The main cause of the difference in strain values under different load conditions (300 N, 500 N, and 700 N) is likely due to the redistribution of stress in each loading scenario. At 300 N (0.2), stress tends to concentrate in smaller areas, causing higher strain at those points. At 500 N (0.18), the stress distribution becomes more uniform across the structure, reducing stress concentration and resulting in lower strain. At 700 N (0.45), despite the higher load, the structure may deform in stiffer areas or exhibit changes in material response, leading to higher strain. Other factors, such as variations in local stiffness or imbalances in material response, could also contribute to these differences.

As the applied load increases to 500 N (Fig.4(b)), the evolution of the strain distribution reveals a proportional increase in the strain magnitude, reaching a maximum value (≈0.027) that is attributed to the uniform response of the material as predicted by Hooke's law [17]. The gradual enlargement of the strained regions, particularly near the joints, reflects a clear correlation between the applied load and the resultant elastic deformation [19]. Such progressive changes serve to validate the simulation results, demonstrating that the material behavior adheres to known principles of linear elasticity prior to reaching the plastic deformation regime [17]. At the highest load of 700 N (Fig. 4(c)), the recorded maximum strain of approximately 0.045 suggests an escalation in material elongation in critical areas. The broader regions experiencing higher strain indicate zones of mechanical vulnerability, underscoring the need for structural reinforcement to enhance durability and reduce the risk of fatigue under cyclic or excessive loading [20]. The accompanying graphical trend (Fig. 4(d)), with steadily increasing maximum strain and nearly constant minimum strain, further confirms that the cane's deformation behavior follows predictable elastic principles until its safety margin is compromised [17]. This comprehensive strain analysis reinforces the significance of targeted design optimization in load-bearing assistive devices, ensuring that high-strain regions are adequately supported.

While the strain distribution evolves proportionally with increasing load, indicating elastic material behavior as per Hooke's law, the corresponding displacement data (Figure 5) reveals a significant anomaly. The displacement simulation results for the NeoMossa model, as depicted in Figure 5, are consistent with established practices in structural finite element analysis and visualization. In this simulation, three different loading conditions 300 N, 500 N, and 700 N were applied, and the corresponding displacement distributions were rendered using color contours. Warmer hues denote regions of higher displacement, a method that is widely used to visually convey complex spatial variations in displacement fields [21]. Specifically, the three Figures (a), (b), and (c) illustrate that the maximum displacements increase progressively (6.219 mm for 300 N, 10.291 mm for 500 N, and 14.404 mm for 700 N).

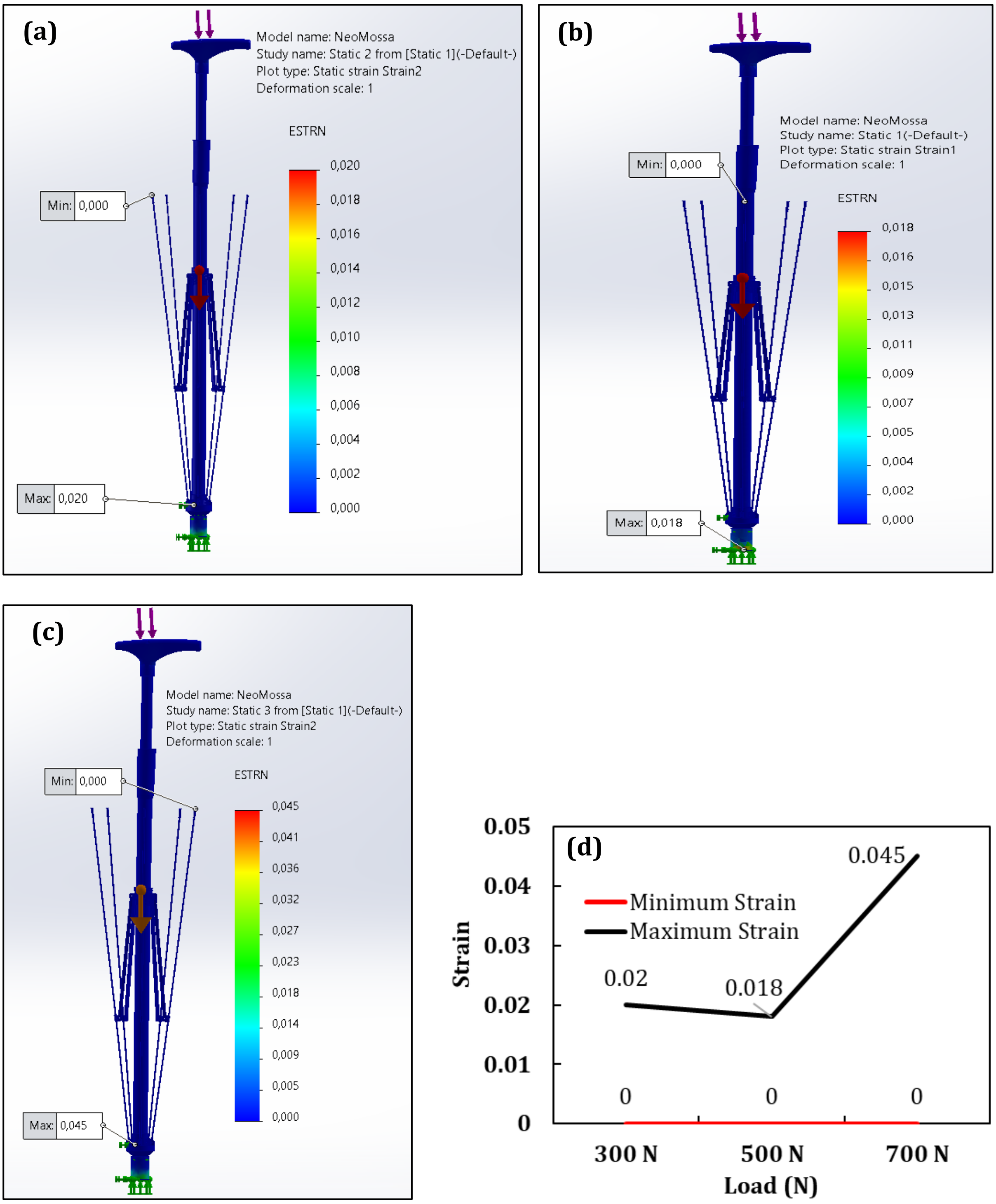

Figure 4. Strain Simulation of *NeoMossa* Stick Under Varying Loads: (a) Strain simulation with 300 N load (b) Strain simulation with 500 N load (c) Strain simulation with 700 N load, and (d) Load variation effect on minimum and maximum strain of *NeoMossa* stick

This monotonic increase in displacement with increased loading is in line with the behavior observed in structural simulations and validated by second-order finite element analyses, where even subtle changes in load can significantly influence displacement [22]. The deformation scale is set to 1, ensuring that the displacements are shown at their true magnitudes for an accurate representation of structural response without additional scaling. Complementing these spatial maps, the graph in Fig. 5(d) plots both minimum and maximum displacement values against the applied load, highlighting the clear load-displacement relationship. This relationship, which shows an increase in maximum displacement up to 14.404 mm under a 700 N load, is indicative of the NeoMossa structure experiencing progressively higher deformations with increased load. Such behavior is critical for assessing the load-bearing capabilities and overall structural integrity, corroborating findings from studies on dynamic analysis and digital twin frameworks that aim to provide real-time insights into structural behavior under varying conditions [23].

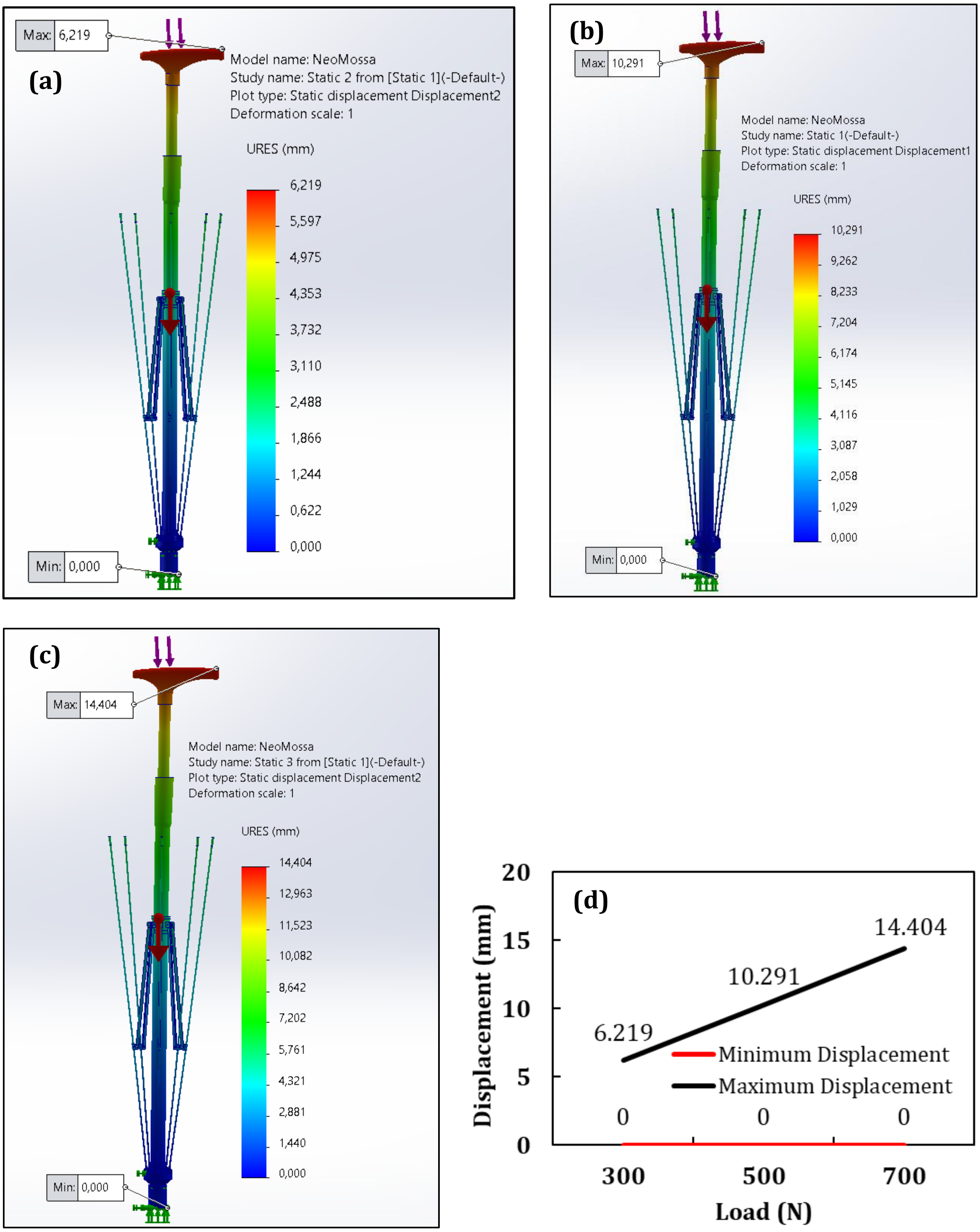

Figure 5. (a) Displacement simulation with 300 N load (b) Displacement simulation with 500 N load (c) Displacement simulation with 700 N load, and (d) Load Variation Effect of *NeoMossa* Stick Displacement

Safety Assessment and Design Validation

The Factor of Safety (FoS) analysis further affirmed the structural resilience of the NeoMossa cane. Values recorded were 4.15 for 300 N, 2.48 for 500 N, and 1.72 for 700 N (Figure 6).

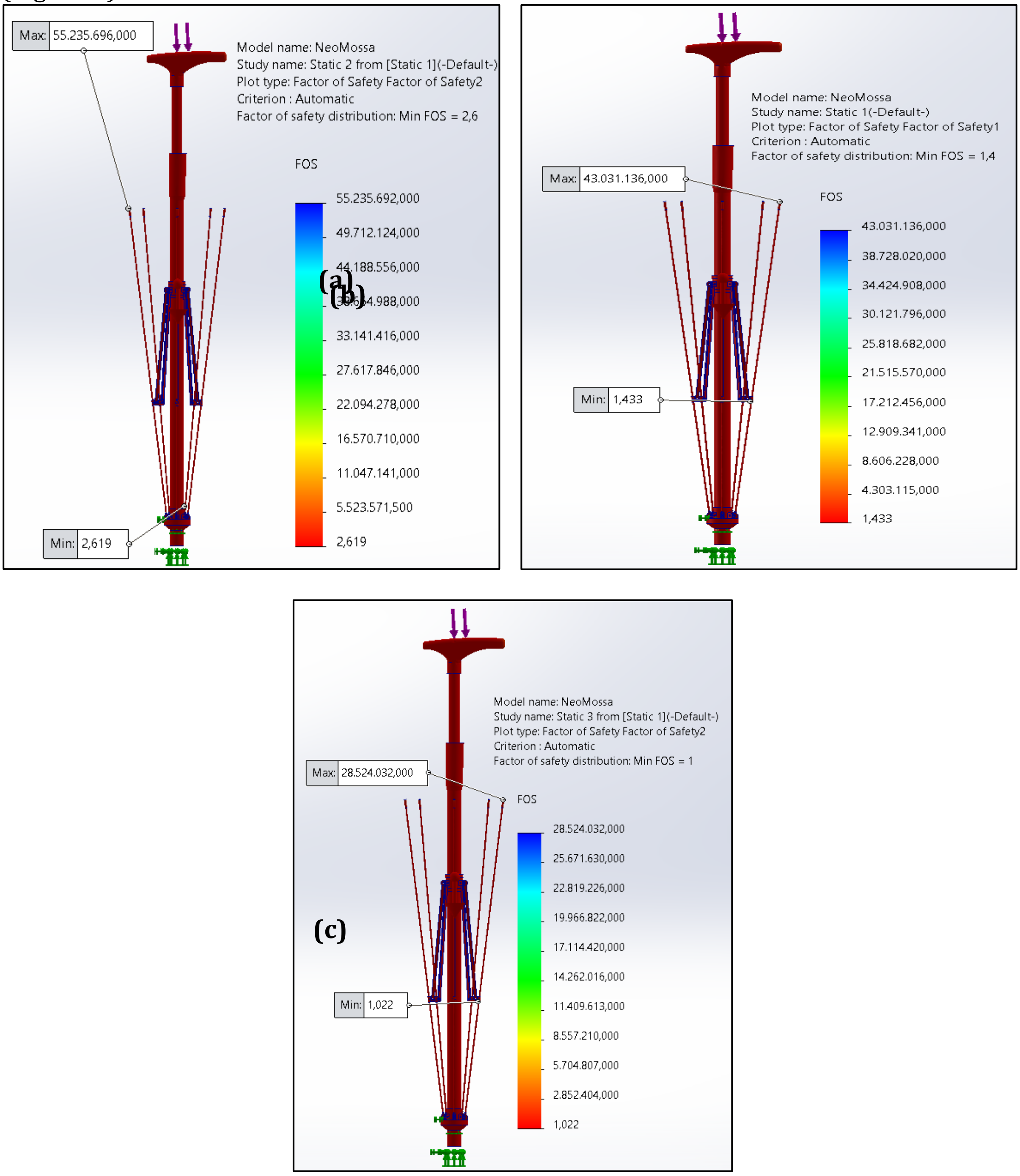

Figure 6. Factor of Safety simulation for NeoMossa cane at (a) 300 N, (b) 500 N, and (c) 700 N

The simulation results illustrated in Figure 6 are consistent with advanced analyses reported in the literature regarding the evaluation of structural safety under varied load conditions. In the NeoMossa cane study, the distinct decrease in the Factor of Safety (FOS) from 2.6 at a 300 N load to 1 at a 700 N load emphasizes the sensitivity of the structure to increasing loads, a phenomenon also observed in reliability-based assessments (Wang et al., [14]. Such analyses typically adopt probabilistic frameworks and finite element methodologies that account for uncertainties in load and material properties, thereby providing a more comprehensive safety evaluation than traditional deterministic methods.

Several studies have successfully employed the Finite Element Method (FEM) to capture stress distributions and predict safety factors in components subject to dynamic and static loads. For example, Purnomo et al., [24] applied FEM to predict stress and safety factors in shock absorbers under cyclic loading, demonstrating that meshing strategies and convergence tests are crucial to obtaining accurate results. Similarly, Alardhi et al., [25], utilized a simulation-based design approach to analyze a lightweight vehicle chassis, emphasizing the importance of optimizing material distribution and structural geometry to maintain high levels of safety across varying load conditions. These studies affirm that the trends observed in the FOS for the NeoMossa cane-specifically the localized vulnerability around connection joints, indicated by the red color gradients in the simulation are consistent with established practices in structural engineering.

Additionally, research on shakedown analysis has shown that structures under multiple load combinations might experience a decrease in safety margins, which necessitates a careful evaluation of the design, particularly under near-critical loading conditions [26]. The reduction in the minimum FOS with increased applied loads in the NeoMossa cane is an indicator that, although the structure remains ostensibly safe, there is a need for potential reinforcement, especially when considering cyclic or higher-than-anticipated loads. Such reinforcement strategies are fundamental to ensuring long-term durability and preventing progressive failure, as reinforced by analogous studies in the structural safety field.

CONCLUSIONS

Based on the results of the Von Mises stress, strain, displacement, and factor of safety analysis of the NeoMossa cane using SolidWorks, it can be concluded that the device is capable of withstanding loads up to 700 N under safe conditions, although the factor of safety decreases at higher loadings. The analysis of Von Mises stress, strain, displacement, and the factor of safety provides a comprehensive understanding of the device's structural performance under various loading conditions. In the stress analysis, the stress value increases as the load increases, with the maximum value reaching 87,008,040 N/m^2 at a load of 700 N, which is still within the safe limit for the device's use. The maximum strain recorded is 0.045 at 700 N, indicating deformation that remains within the acceptable tolerance for the material. Displacement increases with load, reaching a maximum value of 14.404 mm at 700 N, which is still acceptable and does not cause structural damage. The factors of safety for loads of 300 N, 500 N, and 700 N are 2.619, 1.433, and 1.022, respectively, indicating that the device remains in a safe condition at 300 N and 500 N but approaches the minimum acceptable safety limit at 700 N. The decrease in the factor of safety with increasing load suggests a higher risk of structural failure under heavier loading. Although the simulation results provide a useful overview of the device’s performance, it is recommended to conduct experimental testing to validate the simulation outcomes, as experimental tests will yield more accurate insights into the device’s real-world behavior under actual loading conditions and help identify potential risks not detected in simulations, thereby ensuring the reliability and safety of the design in practical applications.

ACKNOWLEDGEMENTS

The authors would like to express their sincere gratitude to the Department of Mechanical Engineering, Faculty of Engineering, Universitas Lambung Mangkurat, for providing laboratory support for this study

DECLARATION OF CONFLICTING INTERESTS

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

FUNDING

The authors declare that no specific funding was received for this research.

REFERENCES

[1] S. Malwade *et al.*, "Mobile and Wearable Technologies in Healthcare for the Ageing Population," *Comput. Methods Programs Biomed.*, vol. 161, pp. 233–237, 2018, doi: 10.1016/j.cmpb.2018.04.026.

[2] A. Woodcock, L. Moody, D. McDonagh, A. Jain, and L. C. Jain, "Design of Assistive Technology for Ageing Populations," 2020, doi: 10.1007/978-3-030-26292-1.

[3] K. Halicka and D. Surel, "Gerontechnology — New Opportunities in the Service of Older Adults," *Eng. Manag. Prod. Serv.*, vol. 13, no. 3, pp. 114–126, 2021, doi: 10.2478/emj-2021-0025.

[4] Y. Morsi, A. Shukla, and C. P. Rathore, "Optimizing Assistive Technologies for Aging Populations," 2016, doi: 10.4018/978-1-4666-9530-6.

[5] P. Asgharian, A. M. Panchea, and F. Ferland, "A Review on the Use of Mobile Service Robots in Elderly Care," *Robotics*, vol. 11, no. 6, p. 127, 2022, doi: 10.3390/robotics11060127.

[6] S. Sarcar *et al.*, "Designing Mobile Interactions for the Ageing Populations," 2017, doi: 10.1145/3027063.3027074.

[7] P. Flores and J. Ambrósio, "Revolute Joints With Clearance in Multibody Systems," *Comput. Struct.*, vol. 82, no. 17–19, pp. 1359–1369, 2004, doi: 10.1016/j.compstruc.2004.03.031.

[8] P. Flores, J. Ambrósio, and J. C. P. Claro, "Dynamic Analysis for Planar Multibody Mechanical Systems With Lubricated Joints," *Multibody Syst. Dyn.*, vol. 12, no. 1, pp. 47–74, 2004, doi: 10.1023/b:mubo.0000042901.74498.3a.

[9] K. E. Reuther *et al.*, "Glenoid Cartilage Mechanical Properties Decrease After Rotator Cuff Tears in a Rat Model," *J. Orthop. Res.*, vol. 30, no. 9, pp. 1435–1439, 2012, doi: 10.1002/jor.22100.

[10] K. K. Santhanaraj, R. M.M., and D. Dinakaran, "A Survey of Assistive Robots and Systems for Elderly Care," *J. Enabling Technol.*, vol. 15, no. 1, pp. 66–72, 2021, doi: 10.1108/jet-10-2020-0043.

[11] N. Yein and S. Pal, "Technological Assistance for Fall Among Aging Population: A Review," pp. 409–419, 2017, doi: 10.1007/978-981-10-3518-0_36.

[12] J. d. Armas, J. Rodríguez-Pereira, B. Vieira, and H. Ramalhinho, "Optimizing Assistive Technology Operations for Aging Populations," *Sustainability*, vol. 13, no. 12, p. 6925, 2021, doi: 10.3390/su13126925.

[13] T. YOKOYAMA, S. IZUMI, and S. Sakai, "Analytical Modelling of the Mechanical Behavior of Bolted Joint Subjected to Transverse Loading," *J. Solid Mech. Mater. Eng.*, vol. 4, no. 9, pp. 1427–1443, 2010, doi: 10.1299/jmmp.4.1427.

[14] C. Y. Wang, M. Du, M. Zheng, and Q. Sun, "Study on the Stress Evolution at Threads of Bolted Joint Under Transverse Loads," *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.*, vol. 239, no. 6, pp. 2106–2116, 2024, doi: 10.1177/09544062241292858.

[15] D. Y. H. Kumarajati, "Design and Analysis of 3D Printable Prosthetic Foot With Honeycomb Structure," *Appl. Sci. Technol. Reaserch J.*, vol. 2, no. 2, pp. 92–99, 2023, doi: 10.31316/astro.v2i2.5628.

[16] A. M. Halahla, "Study the Behavior of Reinforced Concrete Beam Using Finite Element Analysis," 2018, doi: 10.11159/icsenm18.103.

[17] D. D. Kée and K. F. Wissbrun, "Polymer Rheology," *Phys. Today*, vol. 51, no. 6, pp. 24–29, 1998, doi: 10.1063/1.882283.

[18] W. P. Davey, "The Plasticity of Solids," *J. Rheol.*, vol. 1, no. 1, pp. 45–48, 1929, doi: 10.1122/1.2116292.

[19] A. V. Orekhov, "Statistical Criteria for the Limits of Application of Hooke's Law," *Vestn. St. Petersburg Univ. Appl. Math. Comput. Sci. Control Process.*, vol. 16, no. 4, pp. 391–401, 2020, doi: 10.21638/11701/spbu10.2020.404.

[20] D. Hodžić and E. Veladžić, "Experimental and Analytical Deformation Analysis of Truss Members," *Iop Conf. Ser. Mater. Sci. Eng.*, vol. 1298, no. 1, p. 012033, 2023, doi: 10.1088/1757-899x/1298/1/012033.

[21] B. Nelson, E. Liu, R. M. Kirby, and R. Haimes, "ElVis: A System for the Accurate and Interactive Visualization of High-Order Finite Element Solutions," *Ieee Trans. Vis. Comput. Graph.*, vol. 18, no. 12, pp. 2325–2334, 2012, doi: 10.1109/tvcg.2012.218.

[22] C. K. Iu and M. A. Bradford, "Second-Order Elastic Finite Element Analysis of Steel Structures Using a Single Element Per Member," *Eng. Struct.*, vol. 32, no. 9, pp. 2606–2616, 2010, doi: 10.1016/j.engstruct.2010.04.033.

[23] T. Okuda, T. Saida, G. Matono, and M. Nishio, "Digital Twin Framework for Real-Time Dynamic Analysis Visualization With Detecting Dynamic Changes in Structures Properties Using PINN," 2023, doi: 10.1117/12.2658640.

[24] E. D. Purnomo *et al.*, "The Prediction of Stress and Safety Factor of Shock Absorber Based on Cyclic Loading Using Finite Element Method," *Evergreen*, vol. 10, no. 4, pp. 2456–2463, 2023, doi: 10.5109/7162006.

[25] M. Alardhi, F. Almaskari, M. Fahed, and J. Alrajhi, "Analysis of a Lightweight Aluminum Vehicle Chassis in a Simulation-Based Design Approach," *Glob. J. Res. Eng.*, pp. 1–10, 2020, doi: 10.34257/gjrejvol20is1pg1.

[26] D. Magisano, L. Leonetti, and G. Garcea, "Shakedown Analysis of 3D Frames Under Multiple Load Combinations Using Mixed Fiber Beam Elements," 2021, doi: 10.23967/complas.2021.058.