ISSN 2087-3336 (Print) | 2721-4729 (Online)

TEKNOSAINS: Jurnal Sains, Teknologi dan Informatika
Vol. 13, No. 1, 2026, hlm. 170-176
http://jurnal.sttmcileungsi.ac.id/index.php/tekno
DOI: 10.37373

Structural evaluation of the textile machine monitoring system support frame using the finite element method

Deni Kurnia[1*], Alif Firizky[1], Nanang Roni Wibowo[1], Raka Pratindy[2]

[1*] Politeknik Enjinering Indorama, Jalan Desa Kembang Kuning, Ubrug, Jatiluhur, Purwakarta, Jawa Barat, Indonesia, 40552
[2] Politeknik Keselamatan Transportasi Jalan, Jalan Perintis Kemerdekaan, Kec. Tegal Tim., Kota Tegal, Jawa Tengah, 52125

* Corresponding Author: deni.kurnia@pei.ac.id

Submitted: 19/10/2025 Revised: 07/12/2025 Accepted: 14/12/2025

Abstract

The development of an Internet of Things (IoT)-based textile machine monitoring system requires a reliable and mechanically safe support structure. A key issue in current practice is the design of control and monitoring frameworks that rely solely on prior models, without adequate testing and validation. This study aims to evaluate the structural strength of the textile machine monitoring system support frame using the Finite Element Method (FEM). The 3D frame model was designed using CAD software and analysed numerically via static simulation with a loading scenario of 100–150 N applied to the upper part of the structure. The material used is mild steel with mechanical characteristics commonly used in industrial applications. The analysis includes evaluation of Von Mises stress, displacement, and safety factor. The simulation results show a maximum stress of 0.5582 MPa, a maximum displacement of 0.001842 mm, and a safety factor exceeding 12. These values indicate that the designed structure has excellent resistance to static loads and is safe for use in industrial environments. This study provides a strong foundation for developing a reliable and sustainable support structure for textile machine monitoring systems.

Keywords: Finite Element Method; IoT; support frame; textile machinery; structural simulation

1. Introduction

The textile industry is a manufacturing sector highly dependent on the continuous operation and efficiency of machines [1][2]. To maintain performance and prevent premature damage, Internet of Things (IoT)- based machine monitoring systems are increasingly being implemented across various production lines [3]–[6]. This system enables real-time monitoring of machine conditions, thereby increasing productivity and reducing maintenance costs [7][8]. However, implementing such a monitoring system does not depend solely on electronic devices and communication networks; it also requires a supporting structure capable of safely and stably bearing the device's load over a long period of time.

In practice, the supporting structures of monitoring systems are often designed conventionally without considering adequate mechanical evaluation [9][10]. This can lead to deformation, structural failure, or endanger operators and disrupt production processes. Therefore, an engineering-based approach is required to assess the structural design's strength and stability [11].

One effective method for evaluating structural strength is the Finite Element Method (FEM) [12]. This method simulates actual loading conditions on a structure, enabling detailed analysis of parameters such as stress, displacement, and safety factor. Several studies have adopted FEM to analyse the application of permanent magnets in generators [13] or in vehicle paddock stands [14]. However, few studies specifically evaluate the supporting structure of textile machine monitoring systems, especially those that address space limitations in factories, the use of lightweight materials, and relatively small but unevenly distributed electronic loads.

Based on this background, this study aims to conduct a structural evaluation of the supporting frame of a textile machine monitoring system using the Finite Element Method. The primary focus is on the analysis of Von Mises stress and displacement under static loads, and on the calculation of safety factors to assess the structure's feasibility in the operational environment of the textile industry. This study is expected to fill a gap in previous research by providing a systematic, quantitative approach to designing a lightweight support structure that remains safe and sturdy to support the IoT-based monitoring system [15].

2. Method

This study uses a numerical simulation approach based on the Finite Element Method (FEM) to evaluate the structural strength of the supporting frame of a textile machine monitoring system. The process is carried out through four main stages, namely: (1) frame model design, (2) material selection, (3) determination of loading and boundary conditions, and (4) implementation of simulations and analysis of the results.

Frame model design

A 3D model of the support frame was designed in Autodesk Inventor, with dimensions and configurations adjusted to accommodate electronic devices, including a mini PC, a monitor, a vibration sensor, and a microcontroller module. The design took into account ergonomic aspects and installation requirements in a textile industry environment. The structural dimensions refer to space efficiency and stability of equipment placement.

Material Selection: The simulation uses mild steel because it has suitable mechanical properties for lightweight structural applications. The mechanical properties of the material used in the simulation are: a) Modulus of Elasticity (Young's Modulus): 200 GPa, b) Poisson's Ratio: 0.3, c) Yield Strength: ±250 MPa.

Material selection considers strength, availability, cost, and ease of fabrication. Loading and Boundary Condition Determination: Loading scenarios are designed to represent the actual loads the structure will receive during use. A load of 100–150 N is applied to the top of the frame, representing the cumulative weight of the installed electronic devices. The frame legs are given fixed boundary conditions to simulate the structure being rigidly attached to the floor, with no translational or rotational movement.

FEM Simulation and Result Analysis: The simulation was conducted in Autodesk Inventor using the Stress Analysis module and the FEM approach. The main objective of this simulation was to obtain three main mechanical parameters, namely: a) Von Mises Stress, to determine the distribution and maximum stress value that occurs and compare it with the yield strength of the material. b) Displacement, to see the extent of the structure's displacement due to the applied workload. c) Safety Factor, to assess the safety margin of the structure against material failure.

All simulation results are analysed qualitatively and quantitatively to determine the structural feasibility of the proposed design. Feasibility validation is based on mechanical engineering standards and material strength limitations.

3. Results and Discussion

Support frame design

Figure *1* shows the initial design of a support frame structure for a textile machine monitoring system developed using parametric Computer-Aided Design (CAD) software. This frame is designed to support the system's main components, including sensor mounts, a mini PC platform, and modularly interconnected vertical and horizontal support elements.

The overall dimensions of the structure are 1000 mm in height, 300 mm in width, and 225 mm in depth, optimising space efficiency around the textile machine without disrupting production operations.

This design also considers stability factors to withstand static loads from installed electronic devices, such as vibration sensors, thermal cameras, or data processing units.

The parametric-based approach allows flexibility in design modifications, such as resizing or adding components as system development requires. Furthermore, the structure is designed to ensure ease of installation and maintenance access, thus supporting the ongoing implementation of the monitoring system.

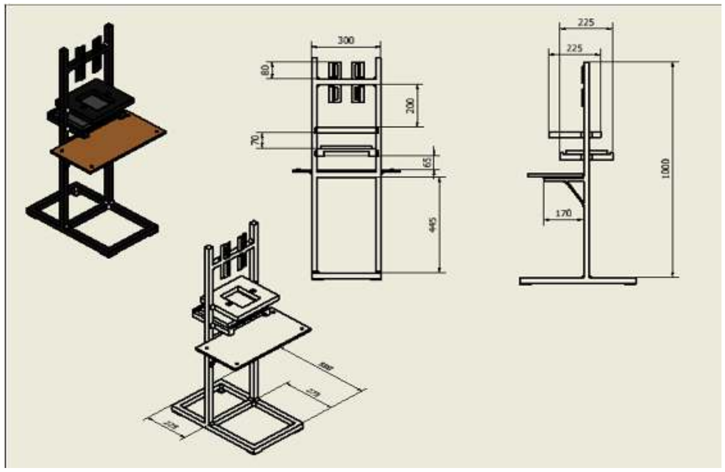

Figure 1**.** Initial 3D design of the support frame and dimensions.

Figure 1 illustrates that CAD visualisation in the drawing helps with the initial analysis of load distribution, critical connection points, and compatibility with the textile industry environment before the fabrication process is carried out.

Von Mises Stress analysis results

Figure 2 shows the results of the Von Mises Stress simulation of the supporting frame structure for the textile machine monitoring system, expressed in MPa (Megapascals). The simulation was carried out by applying a static load of 100–150 N at the top of the structure to represent the actual load from the sensor components and the mini PC.

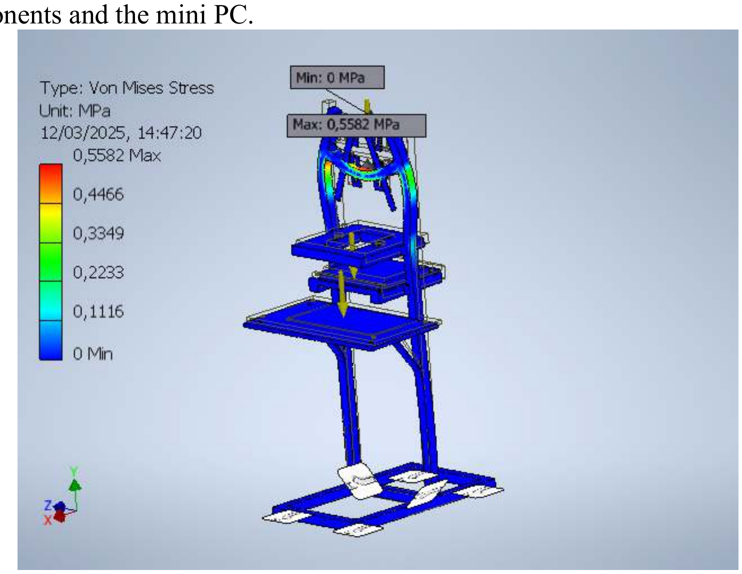

Figure 2. Von Mises stress simulation results with static load.

The analysis results in Figure 2 show that: a) Stress Distribution: The maximum stress recorded was 0.5582 MPa, concentrated in the upper connection area that bears the main load. The minimum value is close to 0 MPa, indicating that the area does not experience significant loading. The stress gradient is distributed gradually (0.1116 MPa–0.4466 MPa), with decreasing values as it moves away from the loading point. b) Structural Safety Evaluation: The maximum stress (0.5582 MPa) is still far below the yield strength limit of mild steel material (±250 MPa), so the structure is considered safe from the risk of plastic deformation or material failure. No critical stress concentrations were detected that could potentially cause cracking or fatigue under normal operational conditions. c) Design Implications: The simulation results validate that the current frame design is capable of withstanding static loads without the need for additional optimisation. The upper connection area can be maintained without further reinforcement, but it is recommended to ensure the quality of the welding or mechanical fastening during production. Visualisation through Finite Element Analysis (FEA) simulation strengthens the argument that the structure meets the reliability and safety factor criteria for textile industry monitoring system applications.

Displacement analysis results

Figure *3* presents the results of the displacement simulation of the supporting frame structure of the textile machine monitoring system under a static load of 100–150 N, with measurements in millimetres (mm).

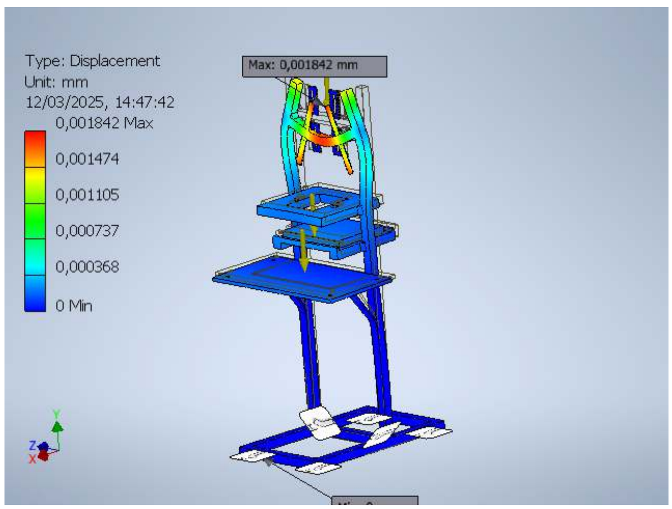

Figure 3. Simulation results of displacement due to static load.

Based on Figure 3, the analysis results can be explained as follows: a) Distribution and Displacement Value: The maximum displacement of the structure was recorded at 0.001842 mm, occurring at the upper end point of the frame, which is the main load-bearing area of the sensor and mini PC. The displacement decreased gradually towards the bottom of the structure (from 0.000368 mm to 0.001474 mm), indicating an even distribution of load across the vertical and horizontal support elements. The minimum displacement approached 0 mm at the base area of the frame, indicating stable fixation and the absence of significant sliding or rotation. b) Stiffness Evaluation of the Structure: The maximum displacement value (0.001842 mm) is minimal (in the order of micrometres), proving that the structure has sufficient stiffness to withstand the design load without significant deformation. In industrial applications, displacements of this magnitude do not significantly affect the operational stability of the monitoring system, given the general tolerances of sensor devices and supporting electronics. c) Design Validation: The simulation results are consistent with the previous stress analysis (Von Mises Stress), where the structure was proven to be mechanically safe with a high safety factor. No design modifications to increase stiffness are required, unless dynamic loading (such as machine vibration) is an additional consideration. The results of the displacement simulation (Figure 3) and Von Mises Stress (Figure 2) complement each other in proving the reliability of the design. Both exhibit a linear, safe structural response under static loading, with a high safety factor relative to the material's yield strength. As for the recommendation at the production stage, ensure the material has a modulus of elasticity consistent with the simulation assumptions (e.g., E = 200 GPa for mild steel) to maintain structural performance.

Results of safety factor analysis

Figure 4 presents the results of the simulation of the safety factor (Safety Factor) of the supporting frame structure of the textile machine monitoring system in unitless units (ul).

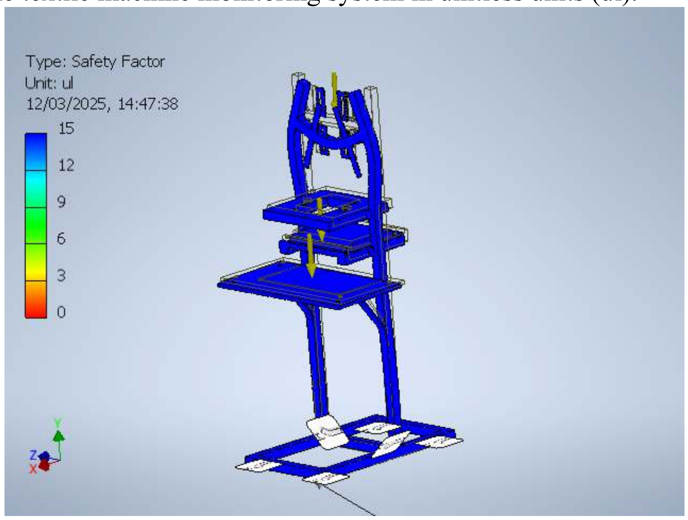

Figure 4. Distribution of safety factor values in frame structures.

This analysis complements the previous evaluation related to Von Mises Stress and Displacement, with the following results: a) Distribution and Safety Factor Value: The minimum value of the safety factor recorded is >12, indicating a very high margin of safety against potential material failure. Then, the distribution of safety factor values ranges from 3 to 15, with the lowest values concentrated in the connection area that receives the main load. There are no areas with safety factor values close to 1, indicating a critical point. b) Evaluation.

Structural reliability

Compared to the industry standard for static applications (minimum SF=2), this design has a safety margin 6 times greater than the minimum requirement. This significant safety margin allows the structure to, among other things, withstand additional dynamic loads (e.g., machine vibration), accommodate material variations during fabrication, and withstand unexpected loading conditions. These results validate that the structure has sufficient mechanical redundancy for long-term operation. Despite the high safety margin, it is recommended to maintain material quality and fabrication processes, conduct regular inspections of critical joints, and consider material optimisation for cost efficiency without compromising safety.

Overall, the results of the safety factor analysis (Figure 4) are consistent with the previous stress and displacement simulations. The combination of the three analyses shows that: a) The working stress (0.5582 MPa) is far below the material capacity (250 MPa), b) The maximum displacement (0.001842 mm) is not operationally significant. c) The minimum safety factor (>12) provides a very adequate margin.

Thus, the analysis results shown in Figure 4 not only confirm the structure's safety under current loading conditions but also demonstrate the potential for further development in terms of material efficiency and adaptation to more complex operating conditions.

Discussion

In summary, the simulation results in Autodesk Inventor demonstrate that the support frame design meets all mechanical performance criteria required for a textile machine monitoring system application. Material Strength Aspect: The maximum stress of 0.5582 MPa at the top mount connection is only 0.22% of the yield strength of the mild steel (250 MPa). Furthermore, no critical stress concentration that can trigger material failure was detected, and the even stress distribution indicates an optimal connection design. Structural Stiffness Aspect: The minimal maximum displacement of the structure

(0.001842 mm) proves adequate stiffness. This value practically does not affect the monitoring system's accuracy. Structural Safety Aspect: Theoretical calculations of the safety factor show an extremely high value (SF = 447.86). The simulation results provide a minimum SF >12, far exceeding the industry standard for static applications (SF≥2). This condition indicates an overdesigned structure and has the potential for material optimisation.

This finding is in line with research on the nyamplung bean peeling machine, which used a similar methodology [12]. The main similarity lies in the effectiveness of the combined analysis of von Mises stress, displacement, and safety factor. Although the current design meets safety requirements, several aspects need to be considered for further development, including material optimisation through component dimension reduction, the use of alternative materials such as aluminium, and the application of topology optimisation.

4. Conclusion

This study evaluates the structural strength of the supporting frame of a textile machine monitoring system using the Finite Element Method (FEM). The simulation and analysis results show that the parametric frame design successfully accommodates monitoring devices such as sensors, mini PCs, and monitors, with good space efficiency and stability. Von Mises stress simulations yielded a maximum value of only 0.5582 MPa, well below the yield limit of mild steel (±250 MPa), indicating the structure is safe under the applied loads. In addition, the minimum maximum displacement (0.001842 mm) indicates high stiffness and minimal deformation during regular operation, while a safety factor above 12 enhances the structure's resistance to failure. Thus, this frame is considered to meet the safety and strength requirements for application in a textile machine monitoring system. However, further validation through experimental tests and dynamic analysis is needed to ensure its performance under real-world conditions.

Acknowledment

Thanks are extended to the Indorama Education Foundation (YPI) and the LPPM Indorama Engineering Polytechnic (PEI) for their financial support for this research.

Reference

[1] S. Utomo and N. Setiastuti, "Industri 4.0: Pengukuran Tingkat Kesiapan Industri Tekstil Dengan Metode Singapore Smart Industry Readiness Index," *J. Techno Nusa Mandiri*, vol. 16, no. 1, pp. 29–36, 2019, doi: 10.33480/techno.v16i1.114. https://doi.org/10.33480/techno.v16i1.114

[2] H. T. Anaam K I and P. A. Y. W. Pranata R Y, Abdillah h, "Pengaruh Trend Otomasi Dalam Dunia Manufaktur dan Industri," *Vocat. Educ. Natl. Semin.*, vol. 1, no. 1, pp. 46–50, 2022.

[3] S. Elkateb, A. Métwalli, A. Shendy, and A. E. B. Abu-Elanien, "Machine learning and IoT–based predictive maintenance approach for industrial applications," *Alexandria Eng. J.*, vol. 88, no. January, pp. 298–309, 2024, doi: 10.1016/j.aej.2023.12.065. https://doi.org/10.1016/j.aej.2023.12.065

[4] T. M. Fernández-Caramés and P. Fraga-Lamas, "Towards the internet-of-smart-clothing: A review on IoT wearables and garments for creating intelligent connected E-textiles," *Electron.*, vol. 7, no. 12, 2018, doi: 10.3390/electronics7120405. https://doi.org/10.3390/electronics7120405

[5] K. B. Nihilesh, A. K. V, and R. Kowshikh, "Remote Monitoring and Data Analysis for Textile Machinery," 2021, doi: 10.4108/eai. 7-12-2021.2314724.

[6] R. Hudec, S. Matúška, P. Kamencay, and M. Benco, "A smart IoT system for detecting the position of a lying person using a novel textile pressure sensor," *Sensors (Switzerland)*, vol. 21, no. 1, pp. 1–21, 2021, doi: 10.3390/s21010206. https://doi.org/10.3390/s21010206

[7] M. Y. Upadhye, P. B. Borole, and A. K. Sharma, "Real-time wireless vibration

monitoring system using LabVIEW," *2015 Int. Conf. Ind. Instrum. Control. ICIC 2015*, no. Icic, pp. 925–928, 2015, doi: 10.1109/IIC.2015.7150876. https://doi.org/10.1109/IIC.2015.7150876

[8] X. Li and Y. Zhu, "A real-time and accurate convolutional neural network for fabric defect detection," *Complex Intell. Syst.*, vol. 10, no. 3, pp. 3371–3387, 2024, doi: 10.1007/s40747-023-01317-8. https://doi.org/10.1007/s40747-023-01317-8

[9] A. Wijianto, M. Ma, M. G. Hadi, and R. Novison, "Experimental study of the strength of straw rope as a material for making mattresses," vol. 6, no. 2, pp. 159–165, 2025. https://doi.org/10.37373/jttm.v6i2.1376

[10] D. Octavianto and R. B. Jakaria, "Implementation Design for Assembly in Two-Wheel Motorcycle Chassis Design (Case Study at Hummeroad Workshop)," *TEKNOSAINS J. Sains, Teknol. dan Inform.*, vol. 12, no. 2, pp. 171–178, 2025. https://doi.org/10.37373/tekno.v12i2.1266

[11] A. Kharis Surnadi and Subekti Subekti, "Material analysis of tool feed on scrap machines using the vibration method," *JTTM J. Terap. Tek. Mesin*, vol. 5, no. 1, pp. 14–20, 2024, doi: 10.37373/jttm.v5i1.565. https://doi.org/10.37373/jttm.v5i1.565

[12] A. Dharmanto, A. H. Muzakki, H. Sholih, and A. Domodite, "Analysis of the strength of the frame construction on the Nyamplung bean peeling machine using the finite element method," vol. 12, no. 1, pp. 1–7, 2025. https://doi.org/10.37373/tekno.v12i1.1027

[13] A. Ramadhan and M. T. Tamam, “Perancangan Permanent Magnet Synchronous Generator Kapasitas 22 KVA Menggunakan Metode Finite Element Method,” *J. Ris. Rekayasa Elektro*, vol. 3, no. 2, pp. 83–90, 2021, doi: 10.30595/jrre.v3i2.11516. https://doi.org/10.30595/jrre.v3i2.11516

[14] S. Susastro, A. F. H. Muhammad, A. Lostari, and Y. A. Fakhrudi, “Optimasi Desain Paddock Stand Sebagai Sistem Statis Dengan Menggunakan Finite Element Method,” *JRST (Jurnal Ris. Sains dan Teknol.*, vol. 5, no. 1, p. 9, 2021, doi: 10.30595/jrst.v5i1.6023. https://doi.org/10.30595/jrst.v5i1.6023

[15] Afzeri, D. Kurnia, M. N. Hakim, and I. S. Nurmala, "Implementation of FFT in Industrial Induction Motor Monitoring via Mobile Phone," *J. Appl. Sci. Adv. Eng.*, vol. 1, no. 1, pp. 5–10, 2023, doi: 10.59097/jasae.v1i1.8. https://doi.org/10.59097/jasae.v1i1.8