Biomedical and Mechanical Engineering Journal (BIOMEJ)
e-ISSN: 2776-1983, p-ISSN: 2829-5242
Vol. 3, No.1, May 2023, pp 30-35

Structural Strength Analysis of Walker (Walking Aid Device) Using the Finite Element Method

A K Faizin[1,*], N Hasan[2], L Z Faiza[1], N F R Supramono[1], P Bagas[1], G D Widagda[1], B Y Saputro[1], M Z Purwananda[1].

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Pembangunan Nasional "Veteran" Jawa Timur, 60294, Indonesia
[2] Department of Physics, Faculty of Engineering, Universitas Pembangunan Nasional "Veteran" Jawa Timur, 60294, Indonesia

*Corresponding e-mail: ahmad.khairul.tm@upnjatim.ac.id

Abstract. One of the uses of a walking aid device (walker) is to support walking ability and balance effectively. The walker is a walking aid device that provides support for both weak legs and uses both hands for stability. This research aims to analyze the stainless steel as material in the design of the walker using finite element method through ANSYS. In the modelling process, the walker adopts the design from the website grabcad.com. In the simulation stage, the walker is subjected to a load of 1000N with fixed support conditions on all four legs, and then undergoes a meshing process with a size of 3 mm. The material applied to the walker model is stainless steel. Static structural analysis is performed in Ansys Workbench. The simulation results show a deformation value of 0.011272 mm, and the value of equivalent stress or von Mises stress should not exceed 207 MPa, while the stress occurring in the frame is 5.67 MPa. This is still considered safe as the simulation results are lower than the yield strength of the material, while the strain energy value is 0.00587 mJ.

Keyword: Walker, finite element method, material strength

1. Introduction

The increasing number of elderly individuals has led to a rise in healthcare challenges related to their care. Aging brings about various physical and cognitive changes [1]. One major concern for the elderly is the risk of falls, with 10% of falls resulting in severe injuries or even fatalities [2]. Environmental factors are the primary causes of falls, followed by walking and balance difficulties associated with old age. Underlying medical conditions like heart disease and stroke can further weaken the body, affecting balance. Degenerative conditions such as osteoporosis, post-stroke complications, Parkinson's disease, knee pain, and fractures exacerbate these challenges [3].

To address walking ability and balance issues, walking aids like walkers are valuable tools. Walkers provide support for weak legs and utilize both hands for stability. They come in two main types: standard walkers and rolling walkers [4]. It is crucial for individuals using walkers to undergo regular physical rehabilitation to improve leg strength. However, traditional walking aids have limitations in providing cognitive and sensory support, which is highly important for those with physical limitations [5]. Accurate walker design is essential, and incorporating universal design principles ensures that walkers can be used by people of all ages, abilities, and circumstances. Material selection plays a significant role as it must be capable of bearing the user's weight. Hence, this research aims to analyze the suitability of stainless steel as a material for walker design, utilizing finite element method with ANSYS. Overall, the structural strength analysis using the finite element method enhances the understanding of the walker's mechanical behavior and assists in the development of a safe and reliable walking aid device.

2. Methods

Simulation in ANSYS Workbench encompasses various types, including structural, thermal, fluid mechanics, electromagnetic analysis, and more. In this study, ANSYS Static Structural is utilized to analyze the strength of the walker design. The general process in ANSYS involves three main steps: model generation, solution, and result evaluation. Model generation entails simplification, defining material properties, and creating finite element models. In the solution phase, boundary conditions are determined, and the analysis is executed to obtain a solution. Result evaluation involves plotting or listing the outcomes for review and conclusion.

The research instrument employed in this study is a walking aid device (walker). The walker is made of metal pipe material and is equipped with four legs and two grips that serve as support for the user to hold onto. A three-dimensional finite element model is designed using Solidworks 2020, saved in STL format, as shown in Figure 1.

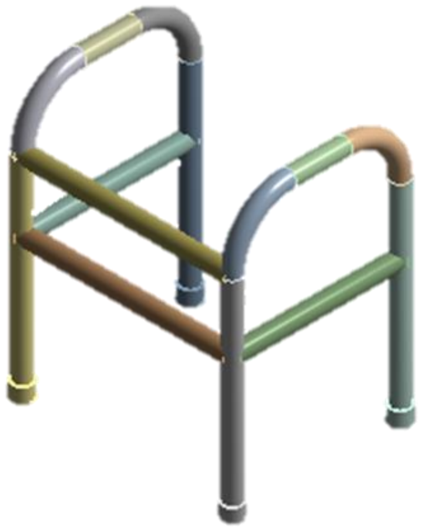

Figure 1. The design of walker (walking aid device)

The model dimensions are adjusted according to the prototype. The analysis is performed using the ANSYS® software (ANSYS, Inc., Pennsylvania, USA), version 19.2. The loading is applied centrally to both grip sections with a magnitude of 1000N. Boundary conditions given to the model include fixed support, represented by zero displacement applied to the four legs of the walker [6]. A meshing size of 3mm is used. Figure 2 illustrates the boundary conditions of the walker in the static structural analysis, where the red area indicates the loading and the blue area represents the fixed support.

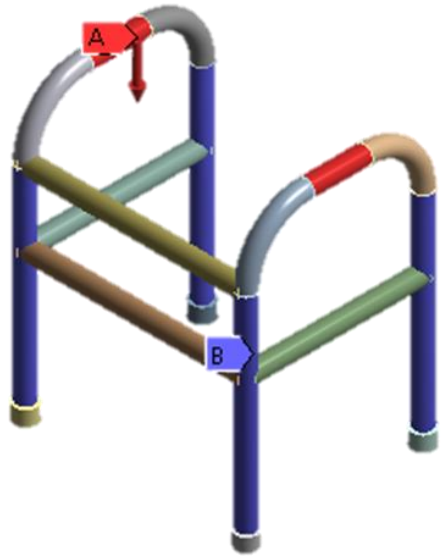

Figure 2. Boundary condition of static structural simulation of walker

Stainless steel is chosen as the material for the walker. The primary criteria for selecting the walker material are its lightweight nature, cost-effectiveness, and moderate strength. By utilizing stainless steel, the aim is to create a walker prototype that is both lightweight and robust, capable of adequately supporting the user while also minimizing the need for complex maintenance procedures [7]. The elastic properties of the material, such as the Young's Modulus (E), Poisson's Ratio (PR), and Density (ρ), are provided in Table 1.

Table 1. Material properties of stainless steel

Material	Density	Young Modulus	Poisson's Ratio	Tensile Yield Strength	Tensile Ultimate Strength
Stainless Steel	7750 kg/m^-3	193000 MPa	0.31	207 MPa	586 MPa

3. Result and Discussion

The stress analysis conducted in ANSYS utilizes the finite element method to assess the behaviour of the walker. Strain-stress analysis involves studying the distribution of stress and strain within the structure. Stress analysis is performed to determine local stress and strain using techniques like the finite element method (FEM) [8]. The simulation process begins by mapping weights for data transfers and interpolating the data from the source mesh to the target mesh for each component [9]. The fundamental principle of the finite element method involves subdividing the complex problem into smaller elements to obtain simplified and manageable solutions. These solutions are then combined to derive the overall solution. Finite element analysis is a numerical mathematical technique used to assess the strength and structural behaviour of engineering components. It involves the meshing process, which divides the object into a grid-like structure. It is essential for researchers to conduct mesh convergence studies to determine the optimal mesh size for accurate modelling, as it can significantly affect the final results. Convergence errors in the mesh allow for control over the numerical solution's accuracy, even in the absence of exact analytical solutions [10].

Several assumptions are made in the analysis, including assuming the material properties to be linearly elastic beyond the yield point, considering low deflection compared to the overall size of the components, assuming the material to be rigid and ductile, and assuming uniform deformation in all directions or isotropic material behaviour. The analysis is performed using linear static analysis, which considers the structural response under static loads [6]. Figure 3 illustrates the simulation results of total deformation or displacement of the material subjected to the applied load. The maximum deformation value of the walker under a 1000 N load is 0.011272 mm, indicating a relatively small deformation.

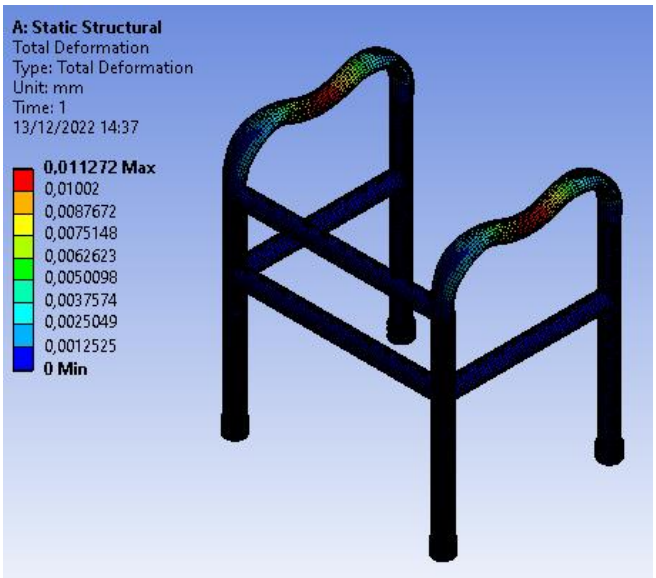

Figure 3. Total deformation

Figure 4 shown the simulation results of the equivalent stress or von Mises stress. The highest value occurred in grip part which has magnitude of 5.67 MPa. This maximum value is far less than the yield strength of stainless steel, thus we can conclude that this design of walker is safe to be used. Meanwhile, the strain energy simulation result is shown in figure 5. The highest energy stored due to deformation occurred in the grip part which has magnitude of 0.00587 mJ.

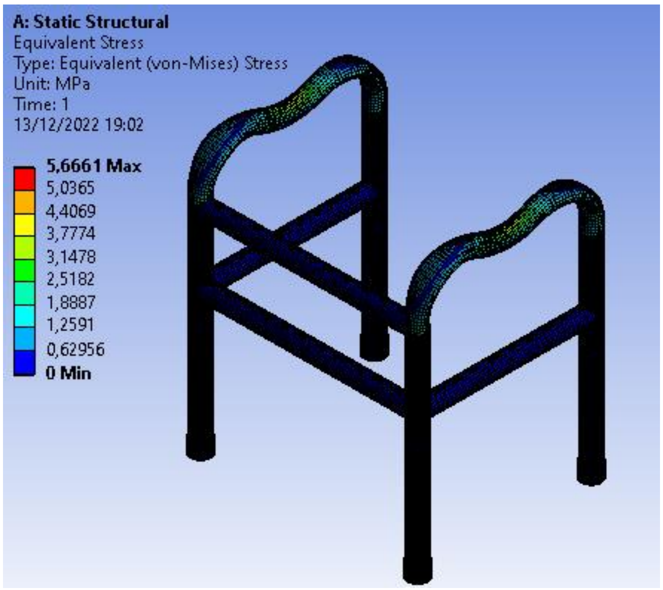

Figure 4. von Mises stresses

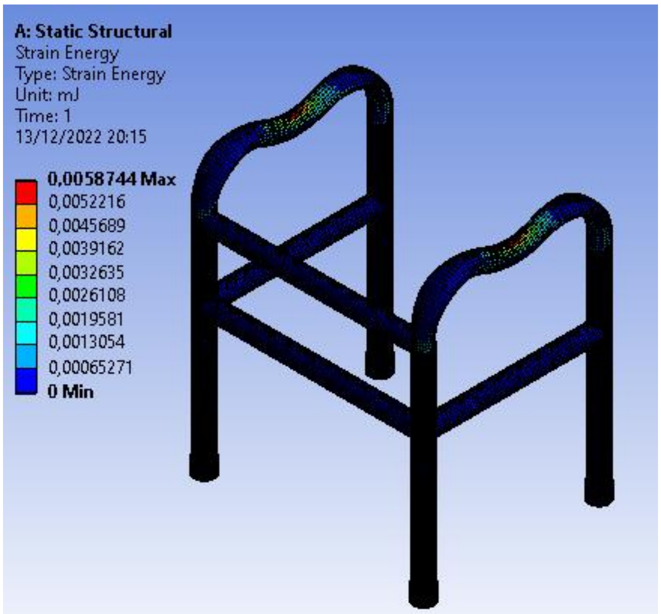

Figure 5. Strain energy

4. Conclusion

The results of the static stress simulation indicate that the designed model of the walking aid device is capable of withstanding a load of 1000N, with the load centered on both grips of the walker. The deformation value is measured at 0.011272 mm, and it is ensured that the equivalent stress or von Mises stress does not exceed 207 MPa, while the stress experienced by the frame is 5.67 MPa. These findings suggest that the design is within safe limits as the simulation results are lower than the material's yield strength. Furthermore, the strain energy is determined to be 0.00587 mJ.

The critical region, where the maximum von Mises stress is concentrated, is primarily located in the grip area. Hence, future research should consider incorporating additional materials in the grip section to enhance safety and improve the comfort for elderly users, without significantly increasing the overall mass. This approach would effectively strengthen the frame of the walking aid device.

For future studies, it would be beneficial to conduct a structural strength analysis of the walker using transient structural simulation approaches. Additionally, validating the simulation results through experimental testing, and even considering direct patient cohort studies, would further enhance the design and ensure its readiness for practical implementation and use by patients.

References

[1] Sadeli Jokowiyono, & Santoso Mulyadi. 2012. Analisa Tegangan Von Mises Pada Alat Bantu Jalan (Walker). *Analisa Tegangan Von Mises Pada Alat Bantu Jalan (Walker)*, *5*, 34–41.

[2] Muammar Bin Md Yasin, A., Wei Liong, L., S.K. Chua, P., & Jianxin, Z. 2016. Design of an Assistive Walking Device with Special Rehabilitation Capabilities. *Universal Journal of Mechanical Engineering*, *4*(6), 147–152. https://doi.org/10.13189/ujme.2016.040603

[3] Djumhariyanto, D. 2016. Pengembangan Alat Bantu Jalan (Walker) Dengan Metode Quality Function Deployment (QFD). *Jurnal Flywheel*, *7*(1), 35–44.

[4] Haryanto, L. T. 2015. The Redesign Of Walker For Pasca- Stroke Patient Use A Value Engineering Method. *Universitas Sebelas Maret*, 61.

[5] Sierra M, S. D., Garcia A, D. E., Otálora, S., Arias-Castro, M. C., Gómez-Rodas, A., Múnera, M., & Cifuentes, C. A. 2021. Assessment of a Robotic Walker in Older Adults With Parkinson's Disease in Daily Living Activities. *Frontiers in Neurorobotics*, *15*(December), 1–14. https://doi.org/10.3389/fnbot.2021.742281

[6] Wibawa, L. A. N. 2019. Desain Dan Analisis Kekuatan Rangka Meja Kerja (Workbench) Balai Lapan Garut Menggunakan Metode Elemen Hingga. *Jurnal Teknik Mesin ITI*, *3*(1), 13. https://doi.org/10.31543/jtm.v3i1.216

[7] Lestari, N. T., Susmartini, S., & Herdiman, L. 2020. Redesign paediatric walker for children with spastic cerebral palsy using TRIZ Method. *Journal of Physics: Conference Series*, *1450*(1), 8–14. https://doi.org/10.1088/1742-6596/1450/1/012117

[8] Köksal, N. S., Kayapunar, A., & Cevik, M. 2013. Fatigue Analysis of a Notched Cantilever Beam. *Proceedings of the Fourth International Conference on Mathematical and Computational Applications*, (June), 111–118.

[9] Chimakurthi, S. K., Reuss, S., Tooley, M., & Scampoli, S. 2018. ANSYS Workbench System Coupling: a state-of-the-art computational framework for analyzing multiphysics problems. *Engineering with Computers*, *34*(2), 385–411. https://doi.org/10.1007/s00366-017-0548-4

[10] Patil, H., & Jeyakarthikeyan, P. V. 2018. Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS. *IOP Conference Series: Materials Science and Engineering*, *402*(1). https://doi.org/10.1088/1757-899X/402/1/012065