

COMPUTATIONAL STUDY OF STATIC LOAD ON FEM BASED ELECTRO-HYDRAULIC OPERATING BED STRUCTURE IN SOLIDWORKS

Ignatius Henry Ismadi¹, Peter Hagios Asa², Doni Raharjo³, Benediktus Indra⁴, Yudha Samodra^{5*}

^{1,2,3,4} Program Studi Teknik Perancangan Mekanik dan Mesin, Politeknik ATMI Surakarta, Surakarta

⁵ Program Studi Rekayasa Teknologi Manufaktur, Politeknik ATMI Surakarta, Surakarta

*Email Corresponding Author: yudha_samodra@atmi.ac.id

ABSTRAK

Rangka struktur tempat tidur operasi berbasis elektro-hidrolik merupakan elemen penting dalam perangkat medis modern, berfungsi menopang tubuh pasien selama tindakan pembedahan. Stabilitas dan kekuatan struktur frame pada unit bed menjadi faktor krusial dalam menjamin keselamatan pasien dan efektivitas prosedur bedah. Oleh karena itu, penelitian ini bertujuan untuk mengevaluasi kekuatan struktur dari komponen *Struktur penopang bagian kepala*, *Rangka punggung*, *Rangka alas tubuh*, dan *Struktur penopang tungkai* pada unit bed meja operasi elektro-hidrolik melalui pendekatan simulasi numerik menggunakan perangkat lunak SOLIDWORKS Simulation. Simulasi dilakukan dengan pembebanan statis yang didasarkan pada distribusi berat tubuh manusia sebesar 280 kg, yang dikonversikan menjadi gaya normal (N). Hasil simulasi menunjukkan bahwa nilai tegangan maksimum yang terjadi adalah: *Struktur penopang bagian kepala* sebesar 14,888 N/mm², *Rangka punggung* 105,621 N/mm², *Rangka alas tubuh* 2,495 N/mm², dan *Struktur penopang tungkai* 81,185 N/mm². Seluruh nilai tegangan tersebut berada di bawah batas luluh material *Aluminium Casting* sebesar 220 N/mm². Berdasarkan hasil tersebut, dapat disimpulkan bahwa desain frame telah memenuhi persyaratan kekuatan struktural dan layak digunakan dalam lingkungan klinis.

ABSTRACT

The electro-hydraulic operating table is a vital component in contemporary surgical systems, responsible for supporting the patient's body throughout various procedures. Ensuring the structural strength and stability of the bed frame is critical for maintaining patient safety and enhancing procedural success. This study investigates the mechanical performance of key substructures within the bed unit, namely the cranial support structure, dorsal frame, pelvic base, and lower limb support frame. A static structural analysis was conducted using SOLIDWORKS Simulation software, incorporating load distribution data based on a 280 kg patient. Results revealed maximum stress values of 14.888 N/mm² (cranial support), 105.621 N/mm² (dorsal frame), 2.495 N/mm² (pelvic base), and 81.185 N/mm² (lower limb support). All stress values were below the yield strength of aluminum casting (220 N/mm²), indicating structural safety. The findings suggest that the current frame design is mechanically reliable and suitable for clinical deployment under specified load conditions.

Keywords: electro-hydraulic operating table, bed frame, SolidWorks Simulation, Von Mises stress, Aluminum Casting.

PENDAHULUAN

The operating table is an essential device in the medical world, especially in surgical procedures, because it functions as the main support of the patient's body during the procedure. Its role is crucial in keeping the patient's body position in line with the needs of the surgery, so that doctors can work effectively. In the context of modern surgery, the operating table is required not only to be flexible in position arrangement, but also to be able to guarantee overall stability and safety (Singh & Kumar, 2020). One type of operating table that is now commonly used is an electro-hydraulic-based operating table. The system integrates electrical and hydraulic technology, which allows for precise movement and positioning of patients quickly and

accurately (Zhang et al., 2021). The stability, strength, and reliability of the mechanical structure of this table are very important because they have a direct effect on patient safety and the effectiveness of medical procedures (Ismail et al., 2019).

The main component of the electro-hydraulic operating table is the bed unit which is composed of several frame parts such as the head support structure, the dorsal frame, the body base frame, and the limb support structure. Each of these parts is designed to support the weight of the patient's body with a maximum capacity of up to 280 kg. Thus, the design and development of the frame structure of the bed unit is an important element in guaranteeing the structural integrity and overall performance of the operating table. Technical

challenges in its design and manufacturing process include the selection of the right materials, the use of appropriate machines and production aids, and the efficient and effective arrangement of process sequences (Kalpakjian & Schmid, 2014). Once manufactured, the frame must also go through a functional and performance testing process to ensure that the components can operate optimally under real working conditions (ISO 60601-2-46, 2016). Therefore, an in-depth study of design and manufacturing aspects is needed to ensure the strength, safety, and reliability of the electro-hydraulic operating table structure in support of complex surgical procedures.

RESEARCH METHODS

This study adopts a numerical analysis approach based on computer simulation to assess the structural strength of the frame components of the electro-hydraulic operating table bed unit. The simulation process was carried out using Computer-Aided Engineering (CAE) software, namely the 2020 version of SOLIDWORKS Simulation. The purpose of this method is to determine the prone points and calculate the amount of stress due to static loading that works on the frame geometry model. The simulation data was used to assess structural safety aspects based on the shape configuration and characteristics of the material used. The final assessment is carried out by comparing the highest stress value to the yield strength limit of the material, to determine whether the design is structurally feasible.

The series of research was carried out through several stages, as shown in Figure 1. The initial stage involves the preparation of tools and materials, such as computers/laptops and SolidWorks software, including the model part file (.sldprt) that is the object of analysis. The next stage is data collection and processing, which includes the identification of fixtures, the location of the load, and the value of the load to be applied. Once the model and parameters are complete, the simulation is run using SolidWorks' built-in modules to obtain stress distributions and deformation patterns. The final step includes the interpretation of the simulation results as well as drawing conclusions regarding the feasibility of the frame design based on the voltage value produced.

This study uses hardware in the form of a laptop with a minimum specification of Intel(R) Core(TM) i5-9300H processor @2.40GHz (8 cores) and a RAM capacity of at least 8 GB. This

configuration is considered sufficient to run static simulations on SolidWorks software with medium-sized geometric models. The software used is the 2020 edition of SolidWorks, which has been equipped with an integrated simulation module called SolidWorks Simulation. The analyzed input file is a part file with the extension .sldprt, containing a three-dimensional model of the bed unit frame components.



Figure 1. Research Process Flowchart

Data collection for simulation purposes was carried out through various methods, including direct interviews and consultations with end users (customers), final project supervisors, and internal discussions of the project team. The purpose of this process is to obtain the necessary technical parameters, such as the position of the fixture, the amount of load force, and the location of the load, to match the real operational conditions of the electro-hydraulic operating table. In addition, additional information was obtained from literature studies, related journals, and other supporting technical documents. All the data that has been collected is then used as the basis for input in the simulation setup process in the software.

RESULTS AND DISCUSSION

The analysis of the strength of the skeletal structure in the electro-hydraulic operating bed module begins with a static loading approach based on the distribution of the weight of the human body. As

shown in Table 1, the weight of the human body of 280 kg is distributed to several parts of the body proportionally, with the largest contribution coming from the trunk (43.02%) of 120.5 kg, followed by the thighs (28.94%) of 81.0 kg, the head (6.81%) of 19.1 kg, the hands (9.43%) of 26.4 kg, and the legs (11.80%) of 33.0 kg. This distribution is the basis for determining the actual load acting on each frame of the bed unit, as the position of the patient's body during the surgical procedure will exert a different compressive force on each part of the frame.

Table 1. Proportion of body mass by body region

Body Parts	Percentage (%)	Heavy (kg)
Segmen kranial	6,81	19,1
Segmen torso	43,02	120,5
Top Ekstremitas	9,43	26,4
Segmen femoral	28,94	81,0
Distal lower extremities	11,80	33,0
Berat Total	100	280

Furthermore, Table 2 shows the distribution of body mass in each part of the bed unit, namely the head support segment, the back support segment, the pelvic support, and the lower extremity support segment, with weights of 10.1 kg, 16.2 kg, 50 kg, and 18 kg, respectively. This data is important to take into account the total load that will be received by each frame, because the frame structure bears not only the weight of the patient's body but also the weight of the bed unit itself. The sum between the weight of the body per section and the weight of each section results in the total actual mass received by each frame.

Table 2. Bed Unit Weight Distribution

Part	Heavy (Kg)
Head-support segment	10,1
Back support segment	16,2
Pelvic mount base	50
Lower extremity support segment	18

The load calculation on each frame is shown in Table 3. The support structure of the head section

receives a combined load from the head and head section of 29.2 kg, which when converted into compressive force (assuming gravitational acceleration

($g = 9.81 \text{ m/s}^2$) resulting in a total load of 286.5 N or 143.25 N per frame (since there are two frames with a symmetrical distribution). The dorsal frame receives the highest load, which is the combined mass of the trunk, arms, and back section of 191.8 kg which is equivalent to a force of 1881.6 N or 940.8 N per frame. This shows that the dorsal frame is the most critical part in receiving structural loads, so it needs to be given special attention in the evaluation of material strength and geometric design. Meanwhile, the base frame received a total load of 1285.0 N, and the limb support structure of 1295.0 N, with a load distribution per frame of 642.5 N and 647.5 N, respectively.

Table 3. Load Distribution on the main frame of the surgical bed unit

Part	Sum	Total Weight	Load (N)	Frame Load (N)
Head support structure	2	29,2	286,5	143,25
Dorsal support structure	2	191,8	1881,6	940,80
Pelvic mount base	2	131	1285	642,50
Lower extremity support segment	2	132	1295	647,50

Referring to the results of the analysis that has been obtained, it can be identified that each frame structure is subjected to varying loads, and simulation using SOLIDWORKS Simulation software is crucial to evaluate the distribution of internal forces as well as structural deformations in the frame unit. The results of this numerical analysis will show the location of the critical region which is a reference in determining the feasibility and safety of the frame structure if used in real conditions in the operating room. With the use of Aluminum Casting material, the next evaluation will focus on the comparison between

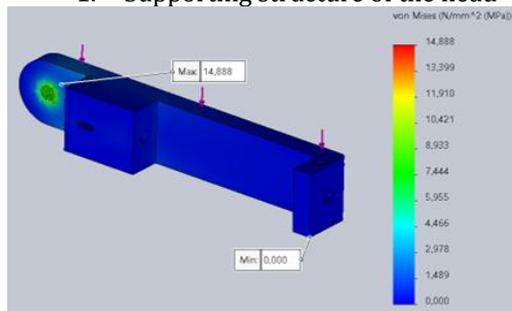
the maximum voltage that occurs and the yield strength of the material to determine whether the frame design is able to withstand the operating load without experiencing failure.

Software Analysis Results:

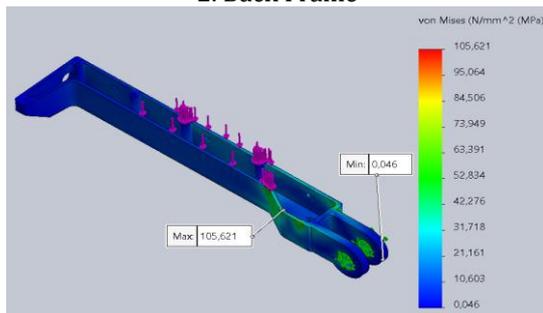
The design of the frame structure on the electro-hydraulic operating table anvil unit is predicted to undergo repeated loading cycles in the long term, along with the intensity of wear during the surgical procedure. Therefore, resistance to material fatigue is one of the crucial aspects that needs to be carefully analyzed. To anticipate the possibility of structural failure, a numerical simulation based on the finite element method (Finite Element Analysis) was performed, using SOLIDWORKS Simulation software, focusing on the maximum static load conditions as listed in Table 3. The material used in the analysis is Aluminum Casting with a yield strength value of 241.275 MPa.

Table 4. FEM Analysis Results of Frame Section Structure

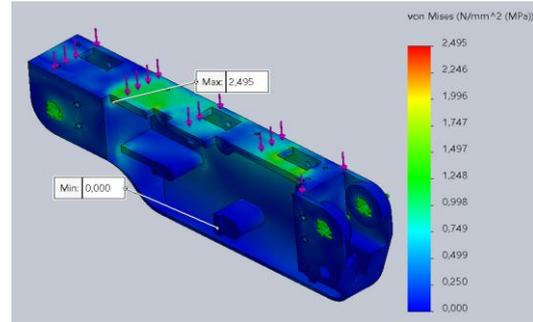
1. Supporting structure of the head



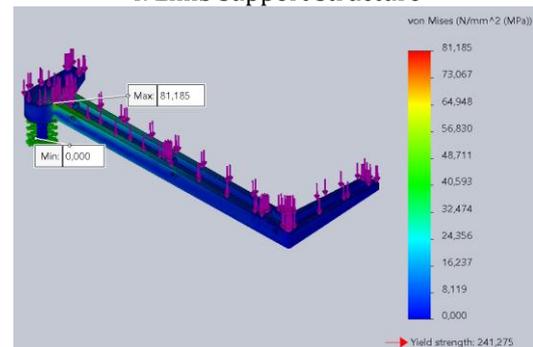
2. Back Frame



3. Body base frame



4. Limb support structure



▪ Supporting structure of the head

Figure 1 and Table 4 show the von Mises voltage distribution in the Supporting structure section of the head. The simulation results show a maximum voltage value of 14.888 MPa, which occurs in the connection area (fixing point), especially at the connection with the shaft. This value is still far below the material yield limit (less than 10%), so it is statically declared safe. However, attention still needs to be paid to the potential stress concentration, as these areas are prone to repetitive load accumulation due to the geometric shape and configuration of the joints.

▪ Dorsal support structure

As per Figure 2 in Table 4, the dorsal skeleton is the component that receives the highest load, especially from the patient's body parts such as the trunk and arms. The maximum voltage was recorded at 105.621 MPa, or about 44% of the yield strength. The stress is distributed fairly evenly across all structural elements, with the highest concentration in the end joint area. Although this value is still within the safe range for static loading, the potential for damage from long-term fatigue is quite high, especially if the joint

design is not reinforced with additional structural elements such as ribs or gussets.

- Pelvic support base structure

The simulation results on the base frame, shown in Figure 3 and Table 4, show a maximum voltage of only 2,495 MPa, which is the lowest value compared to other components. The sturdy frame geometry and even load distribution are factors that cause low voltage. This value is equivalent to about 1% of yield strength, so from a strength and fatigue perspective, the base frame has a very high level of safety and a wide safety margin.

- Structure of the lower extremity support segment

Based on the results in Figure 4 and Table 4, the limb support structure is subjected to a maximum stress of 81.185 MPa, or about 34% of the material yield limit. The voltage is concentrated at the base point and the connection area with the drive actuator. The fairly high voltage values in this area indicate the need for special attention in the aspect of connection design and the selection of local reinforcement methods, especially in the event of dynamic loading due to changes in the patient's position or intense movement of the table legs.

Evaluation of the overall and potential failure of the overall analysis results, it is found that all parts of the frame are still working below the elastic limit of the material (yield strength = 241.275 MPa), so it does not indicate a potential for plastic failure. However, because the actual working conditions of the operating table involve repeated loading, attention to the fatigue factor is very important. Areas with high stress concentrations such as the dorsal frame and limb support structures need to be designed with appropriate arch radius, joint reinforcement (ribs, gussets), and proper joint techniques (e.g., torque-controlled bolting), to minimize the initiation of micro-cracking.

As a follow-up step, it is recommended to perform fatigue life prediction analysis and modal analysis to determine the cycle resistance and vibration response, especially in the back and limb support structures that receive the greatest load. Experimental testing to validate the simulation results is also an important step before the product

enters the mass production stage and medical device certification.

CONCLUSION

Based on the results of numerical simulations carried out with the SOLIDWORKS Simulation software, it can be stated that the frame structure design of the electro-hydraulic operating table unit has met the criteria of strength against a maximum static load of 280 kg. The results of the Von Mises stress analysis show that the maximum stress values for each component are as follows: The support structure of the head reaches 14.888 N/mm², the dorsal frame is 105.621 N/mm², the base frame is 2.495 N/mm², and the support structure of the limbs is 81.185 N/mm². All of these values are still below the yield strength limit of Aluminum Casting material of 241.275 N/mm², so it can be concluded that the structure does not undergo plastic deformation and is safe for mechanical use.

Thus, the frame design of this bed unit is considered to be able to withstand the maximum workload in accordance with the specified operational limit, without showing the potential for structural failure. This shows that the design is technically feasible for use in the medical environment, particularly as a support for the patient's body during the surgical process. However, to ensure long-term reliability, it is recommended to conduct further analysis related to material fatigue analysis, as well as experimental validation tests to ensure the performance of the structure under repeated load cycle conditions throughout the life of the product.

BIBLIOGRAPHY

- [1] Dantes, K. R., & Nugraha, I. N. P. (2019). Analisis tegangan statik pada pengembangan desain cover CVT pada rancangan kendaraan Electric Vehicles Ganesha 1.0 generasi 1 berbantuan software ANSYS 14.5. *Jurnal Ilmiah Teknik Mesin Undiksha*, 5(3). <https://doi.org/10.23887/jjtm.v5i3.20271>
- [2] Firmansyah, F., Hendaryanto, I. A., & M.Eng, S. T. (2021). *Laporan tugas akhir: Perancangan dan pembuatan checking fixture cover Genose C-19 di PT Yogya Presisi Tehnikatama Industri*. Universitas Gadjah Mada.
- [3] Ismail, M. A., Rahman, M. A., & Wahab, A. M. (2019). Design and development of a multipurpose electro-hydraulic surgical table.

- Journal of Medical Devices*, 13(3), 034502. <https://doi.org/10.1115/1.4044337>
- [4] International Organization for Standardization. (2016). *ISO 60601-2-46:2016 – Medical electrical equipment – Part 2-46: Particular requirements for the basic safety and essential performance of operating tables*. <https://www.iso.org/standard/63055.html>
- [5] Kalpakjian, S., & Schmid, S. R. (2014). *Manufacturing engineering and technology* (7th ed.). Pearson Education.
- [6] Li, H., & Chen, J. (2024). Simulation analysis and experimental study of the strength of aluminum alloy suspension structure. *Materials*, 17(1), 128. <https://doi.org/10.3390/ma17010128>
- [7] Prasetyo, E., Hermawan, R., Ridho, M. N. I., Hajar, I. I., Hariri, H., & Pane, E. A. (2020). Analisis kekuatan rangka pada mesin transverse ducting flange (TDF) menggunakan software SolidWorks. *Rekayasa*, 13(3), 299–306.
- [8] Raghavendra, M., & Roy, R. (2024). Fatigue life prediction at mesoscopic scale of samples containing casting defects: A novel energy-based non-local model. *arXiv preprint*. <https://arxiv.org/abs/2407.19519>
- [9] Shiozawa, K., & Tanaka, K. (2014). Effect of micro-porosities on fatigue behavior in aluminum die castings by 3D X-ray tomography inspection. *ISIJ International*, 54(3), 511–518. <https://doi.org/10.2355/isijinternational.54.511>
- [10] Singh, R., & Kumar, A. (2020). Advancements in surgical table mechanisms: A review. *International Journal of Biomedical Engineering*, 12(2), 45–53.
- [11] Wang, X., Wu, Y., & Zhang, H. (2020). Influence of vacuum support on the fatigue life of AlSi9Cu3(Fe) aluminum alloy die castings. *Journal of Materials Engineering and Performance*, 29(11), 7485–7493. <https://doi.org/10.1007/s11665-020-05050-y>
- [12] Zhang, H., Liu, Q., & Wei, Y. (2021). Hydraulic positioning control system for modern operating tables. *Procedia Engineering*, 206, 1202–1210.
- [13] Zhang, L., & Liu, Y. (2022). Effect of surface finishing state on fatigue strength of cast aluminium and steel alloys. *Materials*, 15(13), 4755. <https://doi.org/10.3390/ma15134755>