International Journal of Reconfigurable and Embedded Systems (IJRES)
Vol. 14, No. 1, March 2025, pp. 231~242
ISSN: 2089-4864, DOI: 10.11591/ijres.v14.i1.pp231-242

Finite element analysis method as an alternative for furniture prototyping process and product testing

Fesa Putra Kristianto[1], Zain Amarta[2], Nicolas Hutasoit[2], Nuthqy Fariz[2], Fania Putri Herinda[1]
[1]Department of Furniture Industry Business Management, Furniture and Wood Processing Industry Polytechnic, Kendal, Indonesia
[2]Department of Furniture Design, Furniture and Wood Processing Industry Polytechnic, Kendal, Indonesia

Article Info

Article history:

Received Feb 22, 2024
Revised Jun 7, 2024
Accepted Jul 3, 2024

Keywords:

Finite element analysis
Furniture manufacturing process
Furniture test
Production time
Prototype furniture
Software ANSYS 18.1

ABSTRACT

In the current furniture industry, making furniture goes through many steps. There are ordering materials, designing, building a prototype, and testing samples. This process is considered quite complex, requiring significant costs, and lengthy production time. The application of finite element analysis (FEA) can be a solution to simulate the furniture manufacturing process. Objective of this research was to determine FEA could substitute making and test prototype furniture thereby saving costs and time. This method utilizes ANSYS 18.1 software for more accurate and rapid calculations, incorporating load variables of 400 N, 600 N, 800 N, and 1,000 N, along with gravitational acceleration of 10 \frac{m}{s^2}. The research evaluates the difference (expressed as a percentage) between the results obtained from simulations and those obtained directly from experiments, considering maximum equivalent stress, maximum principal stress, and total deformation values. The final step involves comparing the simulation with direct testing in terms of cost and time. The research results show an average error factor of 5% across all aspect. In terms of cost, the method can save 1,807 USD and reduce production time by up to one month. From these findings, it can be concluded that the process of prototyping and sample testing can be replaced using the finite element method.

Corresponding Author:

Fesa Putra Kristianto
Department of Furniture Industry Business Management
Furniture and Wood Processing Industry Polytechnic
Wanamarta Raya Street No. 20, Kendal Industrial Park, Kendal-51371, Central Java, Indonesia
Email: fesa.putra@poltek-furnitur.ac.id

1. INTRODUCTION

Finite element analysis (FEA) has been widely applied as practical method for modelling complex system in various field such as aerospace, renewable energy, automotive, and civil engineering [1]. FEA development involved method and software. Development of method was started by enhancing manipulation variables such as flexibility, force, deformation, and stiffness. Later, shape of element was modified from two dimensions to three dimensions namely tetrahedral and axisymmetric solid. Software development was started by launched general purpose structural analysis (GPSA) from NASA. Currently, Workbench and SpaceClaim program was developed by ANSYS Inc. for FEA computation. In this method, the structure to be analyzed is discretized into small elements (finite elements), interconnected by nodal points (discrete points). These finite elements, which are generally simpler than the actual structure, have a finite size and must represent the properties of the actual structure. Since each finite element has several unknown variables corresponding to the properties of the actual structure, and the structure itself is a combination of multiple finite elements, software is used for analysis [2].

FEA has significantly contributed to solving industrial problems. It was implemented to test collisions, withstand loads, frame strength, and ship construction strength in automotive industry. It was applied to test the strength of buildings, bridges, and tower deflections in civil engineering. From the various application of FEA, this method has not been implemented in furniture industry, especially for strength testing of product samples. This gap can be fulfilled since the FEA method's error factor was typically 0% - 5% and it was commonly used to measure critical strength of building and automotive machinery [3].

In furniture industry, manufacturing process started from customer order, followed by drafters creating designs based on their preferences. After being approved by customer, drafters then created bill of materials (BOM). Prototype was fabricated according to BOM and tested based on standard, then being sent to customer as a sample. After the sample results being approved, the process continued to mass production based on the amount of costumer orders. The finished furniture product was packaged and sent to customer [4]. The furniture production flow process is explained in Figure 1.

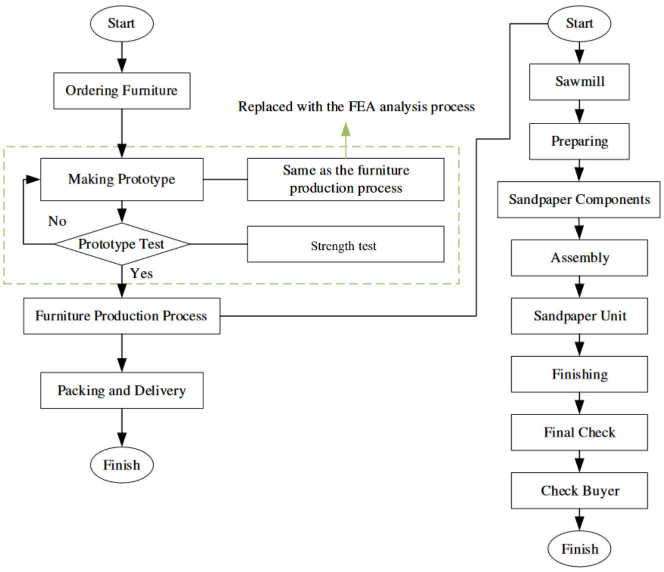


Figure 1. Furniture production process

The FEA method implementation could be conducted on prototype fabrication and product testing. Prototype fabrication costed about 2 times the cost of producing a furniture product. Meanwhile, the product testing costed 40% of the furniture product price. The time for prototype fabrication was 1 month and product testing needed 1 month, so the total time was 2 months. The implementation of FEA method could be completed in one day as an alternative [5]. FEA method would reduce the cost and time production.

Obtaining optimal result, the FEA method was supported by software for accurate and fast computation. The software used was ANSYS Workbench 18.1. The advantage of the ANSYS 18.1 Workbench was produced a lot of and varied engineering data that used for analysis. The process of importing 3D images was very simple and could be conducted from common program that used for 3D images design such as AutoCAD. The computation was more accurate since the settings was very flexible. However, to obtain accurate computation, the mesh size must be standardized according to tested design. The more uniform the mesh shapes such as squares, triangles, and tetrahedral, the smaller the error factor [6].

The implementation of FEA method and ANSYS 18.1 Workbench software were widely used in previous research. Research on shear wall plates testing for earthquake resistant buildings using the FEA method and ANSYS 18.1 Workbench software was conducted. The result of this research was the shear wall plate design made from corrugated plates could increased the strength from earthquakes by 9% and the error factor from FEA analysis method was only 4% [7]. Research on measuring the impact of collisions from fast trains in Indonesia was conducted. In the fast train system, there was an energy absorption system when a collision occurs. This system was tested with FEA method and ANSYS 18.1 Workbench software to evaluate the amount of collision energy. The result of this research was the energy absorption system in high-speed trains could absorb up to 5 MJ of energy from collision incident. The error factor from process data was about 5% [8]. Research on gelam wood for building structure was conducted. The result of this research was gelam wood could be used as a building structure with specifications of a 10-12 cm width and 4 m length. Strength analysis FEA method and ANSYS 18.1 Workbench software was obtained a value of 100.13

MPa [9]. From previous research, FEA method and ANSYS 18.1 Workbench software was very possible to implemented for replacing prototype fabrication and manual product testing become digital simulation.

In this research, the type of furniture tested was a parametric television (TV) table. The testing consisted of two stages. The first stage was the material strength simulation with supported by ANSYS 18.1 Workbench software. The manipulation variables used were loads of 400 N, 600 N, 800 N, and 1,000 N with gravitational acceleration of $10\frac{m}{s^2}$. The second stage was direct testing prototype strength, so that from those stages the error percentage would be determined. The value of error percentage could be considered for deciding whether direct testing of product strength or just simply used ANSYS 18.1 Workbench software.

2. RESEARCH METHOD

2.1. Making prototype

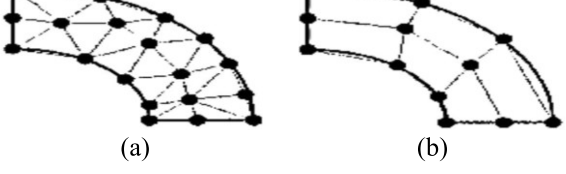
The parametric TV table furniture was made using one of the furniture design methods known as parametric design. It was used for designing and solving structural, form, and aesthetic problems. Parametric furniture was furniture composed of a series of furniture components that have the same distance or arrangement, thus forming a three-dimensional (3D) structure. Its components were made using computer numerical control (CNC) machines and arranged in a specific composition. Its components were joined using glue to form a large lamination. In order to maintain the same distance and assemble them into three dimensions, studs or spacers are used [10].

The parametric TV table furniture was designed using parametric techniques. Its dimensions followed ergonomic norms, measuring 1,498 mm in length, 353 mm in width, and 512 mm in height. The material used was plywood. It was sourced from Kayu Lapis Indonesia Company. Following this, the components were shaped using CNC machining and carving processes. The results from these operations were then utilized in the construction of the table. Examples of furniture products utilizing parametric design methodologies can be seen in Figure 2.

Figure 2. TV table parametric

2.2. Finite element analysis

Finite element constituted a numerical technique predicated on the fundamental paradigm of discretizing a continuum into diminutive and geometrically simplified elements. The finite element method analysis was conducted to derive an approximate solution based on assumed displacement, stress, or a combination of both [11]. The key subject of FEA involves dividing the object under examination into a finite number of subdivisions. The subdivisions were designated as element. It was connected to another element through nodes [12]. Mathematical equations representing the object were then constructed. Meshing involved partitioning the object into a multitude of elements. In calculating the solution per element, it had to meet certain conditions, such as continuity at nodal points and interface elements [13]. Figure 3 illustrates two types of FEA elements, namely triangular elements in Figure 3(a) and quadrilateral elements in Figure 3(b).



(a) (b)

Figure 3. Illustrates two types of FEA elements (a) triangular element and (b) quadrilateral element

2.3. ANSYS 18.1 software simulation

The simulation method used quantitative approach. It had three stage procedure, namely:

2.3.1. Pre-processing

First step of FEA was configuring the ANSYS 18.1 application to prepare it for simulation. The initial configuration for the application involved selecting system analysis. After that, input the 3D design into ANSYS 18.1. The third step was to fill in the engineering data. The fourth step was meshing the 3D design. Finally, set the load and gravity variation. After all these steps have been done, the ANSYS 18.1 FEA testing was ready to proceed.

– Components and structure: this testing was conducted for the structure or design of furniture. It determined the minimum strength limits that the furniture could withstand. While component testing was done to determine the durability of the material used, wood and structural steel frames.
– Selection of analysis system: there were many analysis systems provided by the ANSYS Workbench 18.1 software. In this study, the structural analysis system is utilized [14]. Figure 4 showed an illustration in the application ANSYS 18.1, Figure 4(a) display of analysis system, Figure 4(b) display of structural analysis system simulation.

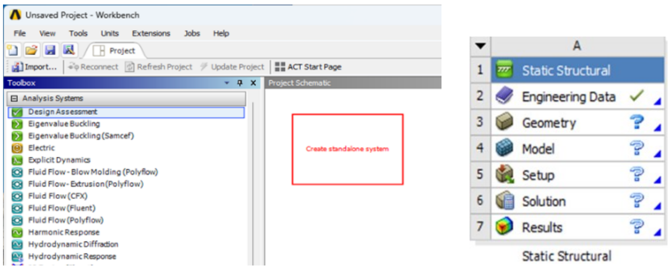


Figure 4. Illustration in the application ANSYS 18.1 (a) selection of analysis system and (b) main part of structural analysis system simulation

– Input geometry: the image 3D model of the parametric TV table was designed using Autodesk AutoCAD 2022. It was converted into initial graphics exchange specification (.IGES) format and then inputted into geometry through import geometry [15]. An example of making geometry in the application could be seen in Figure 5.

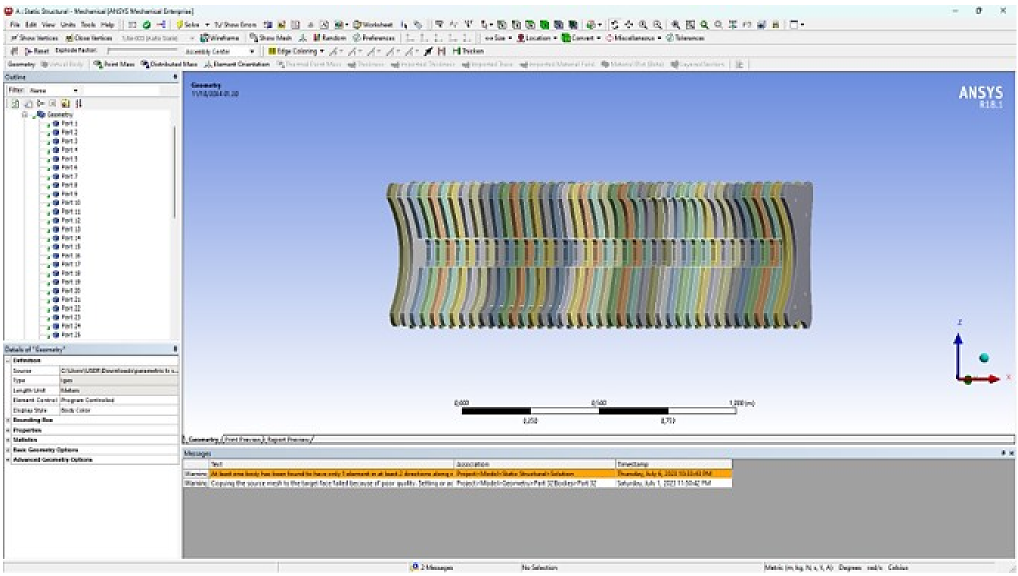


Figure 5. Making geometry in furniture design

- Data engineering was the step where input data for the materials used was collected. It was wood and structural steel. After that, strength testing simulations were conducted using static structural analysis through engineering data source.
- Meshing was performed on each component of the model. Meshing was the process of dividing a geometry into several small objects called meshes with specific shapes and interconnected nodes [16]. In FEA, mesh size was crucial. it was closely related to accuracy. The number of meshes was required for element meshing. Structural analysis types required an appropriate meshing scheme, such as optimal mesh density for static, impact, and frequency analyses. The mesh creation process was carried out using the ANSYS meshing program [17]. The mesh used quadrilaterals with a size of 9 mm, where this mesh type was easy to adapt to complex geometric shapes. The result of meshing process in application for parametric TV table could be seen in Figure 6.
- A simulated strength test was conducted using of ANSYS 18.1, applying loads of 400 N, 600 N, 800 N, and 1,000 N at a gravitational speed of 10 $\frac{m}{s^2}$.

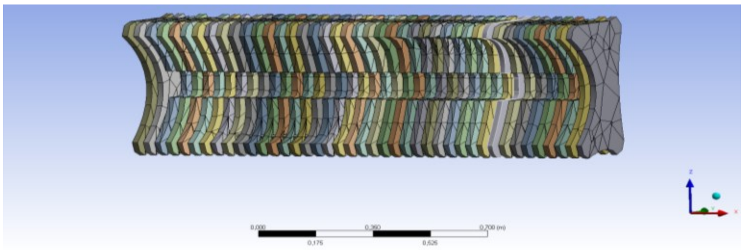


Figure 6. Parametric TV table furniture meshing process

2.3.2. Solution

In the solution process, the calculation data results were displayed after the simulation process. It computed by the computer include equivalent stress, principal stress, and total deformation. In this stage, information about the estimated time for the calculation process (running) was also provided [18]. The calculation data results were visually displayed through contour plots, graphs, and tables. The result data of total deformation, maximum equivalent stress, and maximum principal stress were obtained and subsequently processed using Microsoft Excel. The overall flow of the design testing process with FEA in ANSYS 18.1 was explained in Figure 7.

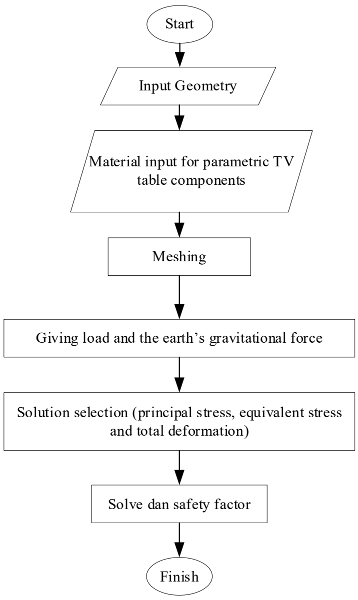


Figure 7. Finite element method analysis flowchart in ANSYS 18.1 software

2.3.3. Evaluation

In the evaluation step, two comparisons were determined. Comparison of calculations were made between the simulation results of the parametric TV table with the test results directly. Then, it was compared in terms of cost and time.

3. RESULTS AND DISCUSSION

The evaluation of the parametric TV table furniture design was undertaken by subjecting the table to loads with magnitudes of 400 N (40 Kg), 600 N (60 Kg), 800 N (80 Kg), and 1,000 N (100 Kg). The meshing size employed was 9 mm. FEA was performed to ascertain the stress and deformation experienced by parametric TV tables under the applied loading conditions, with the maximum scenarios depicted in Figures 8 to 11. In Figures 8 to 11 there are sub figure for sub Figures 8 to 11(a) display of maximum equivalent stress, Figures 8 to 11(b) display of maximal principal stress and Figures 8 to 11(c) display of total deformation.

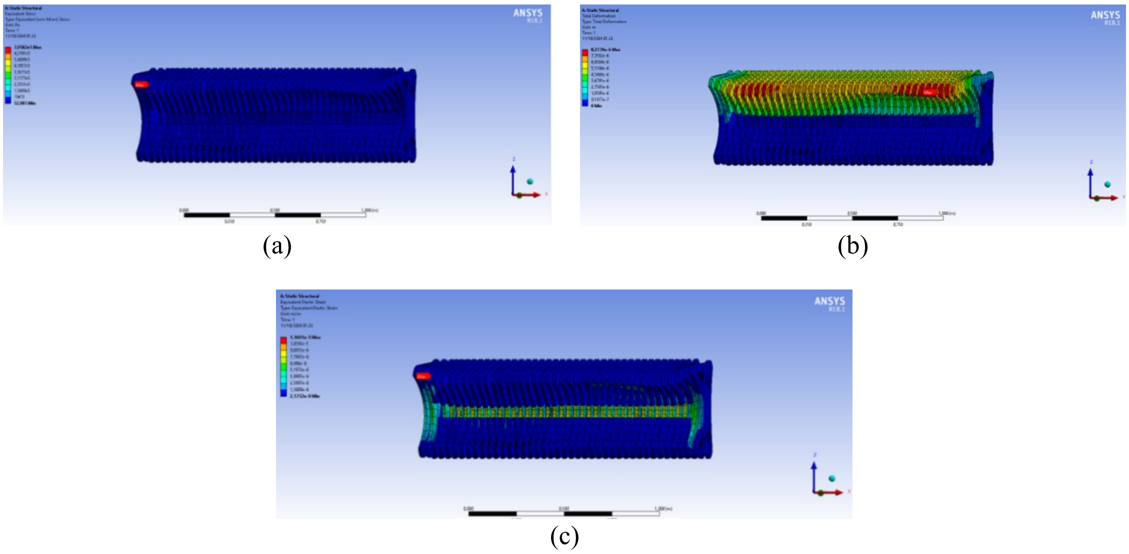


Figure 8. ANSYS 18.1 simulation for parametric TV table with 400 N force (a) maximum equivalent stress, (b) maximal principal stress, and (c) total deformation

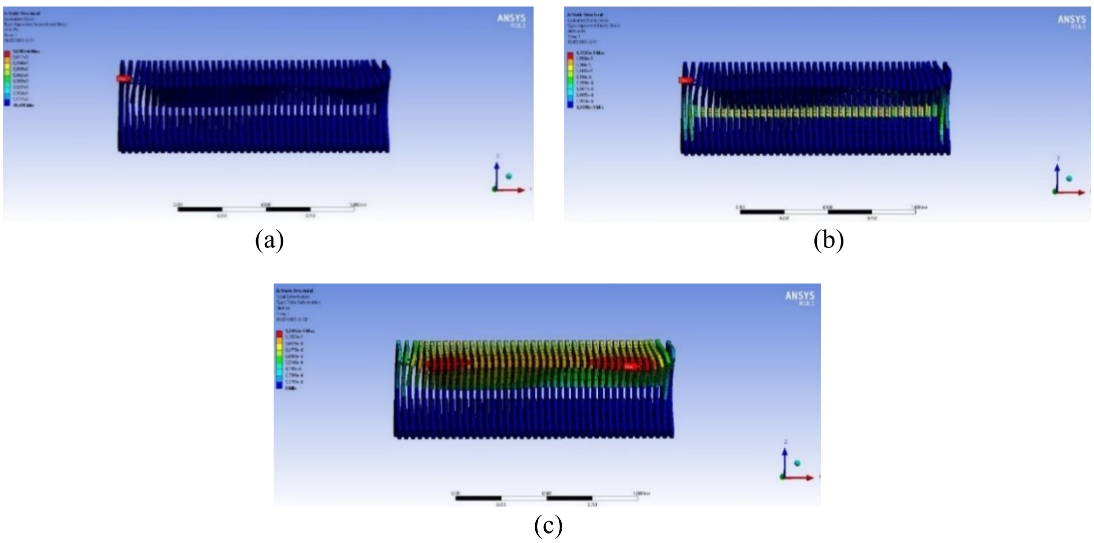
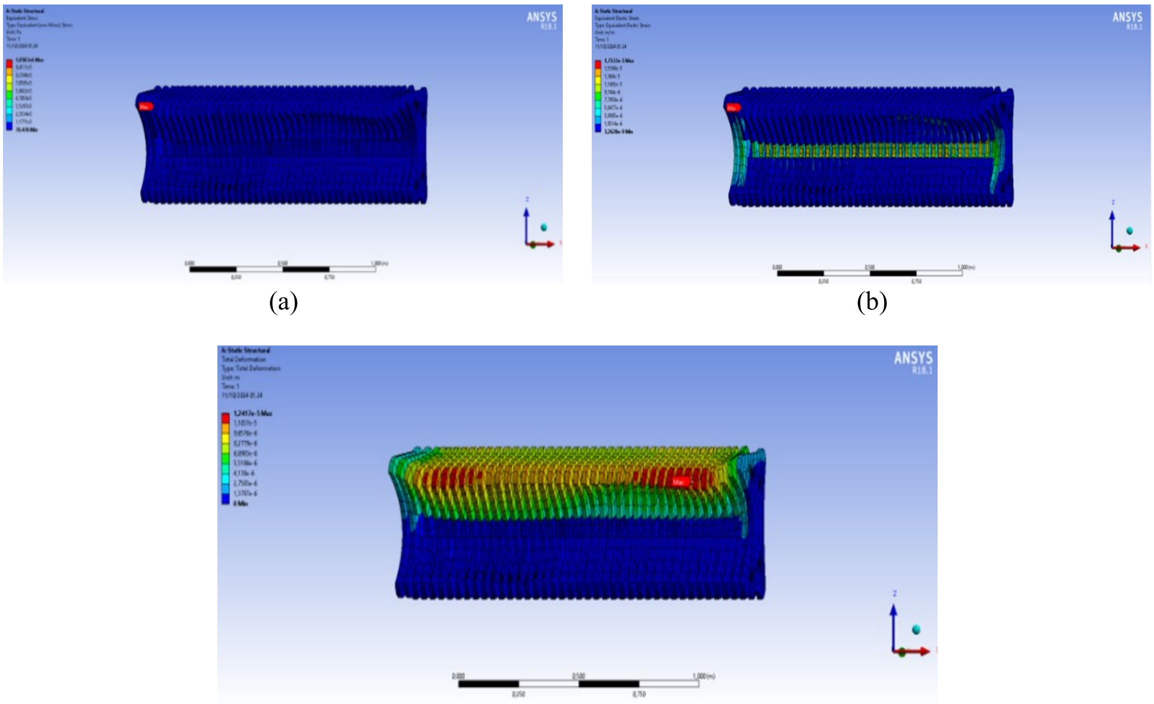
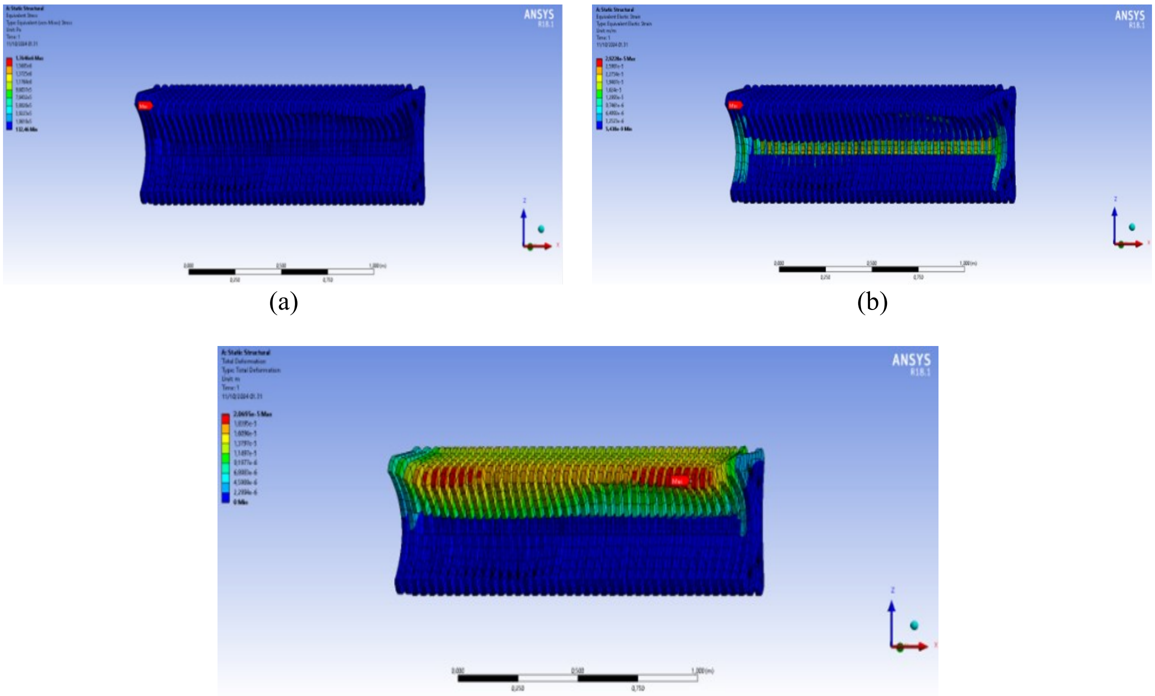


Figure 9. ANSYS 18.1 simulation for parametric TV desk with 600 N force (a) maximum equivalent stress, (b) maximal principal stress, and (c) total deformation



(a) (b) (c)

Figure 10. ANSYS 18.1 simulation for parametric TV table with 800 N force (a) maximum equivalent stress, (b) maximal principal stress, and (c) total deformation



(a) (b) (c)

Figure 11. ANSYS 18.1 simulation for parametric TV desk with 1,000 N force (a) maximum equivalent stress, (b) maximal principal stress, and (c) total deformation

The simulation obtained from the ANSYS 18.1, conducted with applied forces of 400 N, 600 N, 800 N, and 1,000 N, demonstrated that the parametric TV furniture design could adequately fulfill its intended function under load of 1,000 N. The load distribution across the top surface of the parametric TV furniture was uniform. However, the corner regions exhibited a need for design adjustments to accommodate the observed pressure distribution patterns. Such modifications to the corner sections of the parametric TV furniture design could be accomplished by implementing a more compact spatial configuration.

Strength testing on parametric TV furniture is carried out in two stages, namely the ANSYS 18.1 simulation stage and the direct testing stage. The outputs of both the ANSYS 18.1 simulation stage and the direct testing stage will provide values for maximum equivalent stress, maximum principal stress, and total deformation. From Figures 8(a), 9(a), 10(a), and 11(a), maximum equivalent stress values can be obtained from ANSYS 18.1 simulation. The result of its simulation with forces of 400 N (40 Kg), 600 N (60 Kg), 800 N (80 Kg), and 1,000 N (100 Kg) are 7.05×10^{-5} Pa, 1.05×10^{-6} Pa, 1.41×10^{-6} Pa, and 1.76×10^{-6} Pa, respectively. While the maximum equivalent stress values at the direct test stage with forces of 400 N, 600 N, 800 N, and 1,000 N are 7.19×10^{-5} Pa, 1×10^{-6} Pa, 1.5×10^{-6} Pa, and 1.6×10^{-6} Pa.

The maximum equivalent stress represent the total load experience, which is calculated as the sum of elastic load and any additional applied loads [19]. Figures 8 to 11 show that the most severe loading occurs at the corner edges of the parametric TV furniture model, indicated by red, while blue denotes lower stress levels, highlighting the corners as the areas of highest stress. The maximum equivalent stress values in Figure 12, both in the simulation stage and direct testing, peak at a load of 400 N and then stabilize from 600 N to 1,000 N. This is because the highest elasticity in the design occurs at a load of 600 N and remains stable up to 1,000 N. Supplementary loads do not increase the elastic behavior of the design, as the parametric design inherently tends toward rigidity and structural robustness [20].

The Figures 8(b), 9(b), 10(b), and 11(b) show maximum principal stress results from the simulation process. Maximum principal stress is the stress that a material or object in resisting a load [21]. The values of the maximum principal stress in the ANSYS 18.1 simulation stage with forces of 400 N, 600 N, 800 N, and 1,000 N are $1.16\times10^{-5}\,\frac{m}{m}$, $1.75\times10^{-5}\,\frac{m}{m}$, $2.33\times10^{-5}\,\frac{m}{m}$, and $2.92\times10^{-5}\,\frac{m}{m}$, respectively. Meanwhile, the values of maximum equivalent stress in the direct testing stage with forces of 400 N, 600 N, 800 N, and 1,000 N are $1.1\times10^{-5}\,\frac{m}{m}$, $1.82\times10^{-5}\,\frac{m}{m}$, $2.52\times10^{-5}\,\frac{m}{m}$, and $2.83\times10^{-5}\,\frac{m}{m}$. From the overall results obtained, a graph was made in Figure 13.

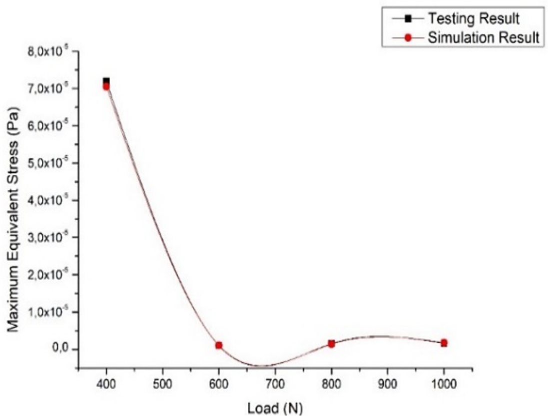


Figure 12. Maximum equivalent stress results of parametric TV furniture

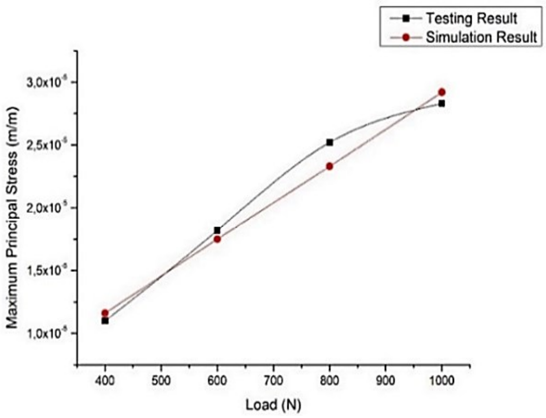


Figure 13. Maximum principal stress results of parametric TV furniture

The maximum principal stress peaked in the columnar components of the parametric TV furniture design due to these columns (*fulcurms*) bearing the highest load within the overall configuration. The maximum principal stress result in both simulation and direct testing stages show that the highest value is at the load of 1,000 N. This is because the heavier the supported load, the higher the value of maximum principal stress. The findings lead to the conclusion that the maximum principal stress exhibits a direct proportional relationship with the magnitude of supported load [5].

Total deformation value from simulation is shown in Figures 8(c), 9(c), 10(c), and 11(c). The total deformation is the net deflection of a system quantified by taking the vector sum of all individual displacements in different orientations [22]. Total deformation at the ANSYS 18.1 simulation stage with forces of 400 N, 600 N, 800 N, and 1,000 N was 8.27×10^{-6} M, 1.24×10^{-5} M, 1.65×10^{-5} M, and 2.06×10^{-5} M, respectively. While the total deformation in the direct test stage with forces of 400 N, 600 N, 800 N, and 1,000 N was 7.9×10^{-6} M, 1.23×10^{-5} M, 1.7×10^{-5} M, and 2.18×10^{-5} M.

From Figure 14, the greatest total deformation occurs along the Y-axis, increasing with greater gravitational force and loading, due to the direct proportionality between deformation and loading on parametric TV furniture [23]. The simulation results indicate that the largest total deformation occurred at the loading of 1,000 N, measuring 2.06×10^{-5} M. Similarly, direct testing results show that the largest total deformation, measuring 2.18×10^{-5} M, occurred at a loading 1,000 N.

In (1) demonstrates the calculation of error factor. The error factor represents the percentage of error that arise in strength test, affecting the discrepancy between the simulation result and the direct testing result. The calculation of the error percentage employs the concept of absolute value. Thus, if a negative result is obtained, it is necessary to determine its absolute value since the purpose of calculating the error percentage is to determine the magnitude of the difference between the simulation value and the direct testing value [24]. The formula for calculating the percentage of errors is as (1).

$$\%\ Error = \frac{Simulation\ Value - Testing\ Value}{Testing\ Value}\ x\ 100\% \quad (1)$$

Based on this formula, it is evident that the maximum equivalent stress in the simulation stage and the test stage, with forces of 400 N (40 Kg), 600 N (60 Kg), 800 N (80 Kg), and 1,000 N (100 Kg), exhibit error percentages of -2%, 5%, -6%, and 10% respectively. Consequently, the average error percentage in maximum equivalent stress is 6%. Similarly, the maximum principal stress with forces of 400 N (40 Kg), 600 N (60 Kg), 800 N (80 Kg), and 1,000 N (100 Kg) in both the simulation stage and test stage demonstrates error percentages of 5%, -4%, -8%, and 3%. Therefore, the average error percentage in maximum principal stress is 5%. Furthermore, the total deformation with forces of 400 N (40 Kg), 600 N (60 Kg), 800 N (80 Kg), and 1,000 N (100 Kg) in the simulation stage and test stage exhibits error percentages of 5%, 1%, -3%, and -6%. Consequently, the average error percentage in total deformation is 3%. Considering all aspects, the average error factor is only 5%, indicating that the simulation results adequately represent the direct test results [25]. Figure 15 illustrates the comparison results of the error percentages across all aspects.

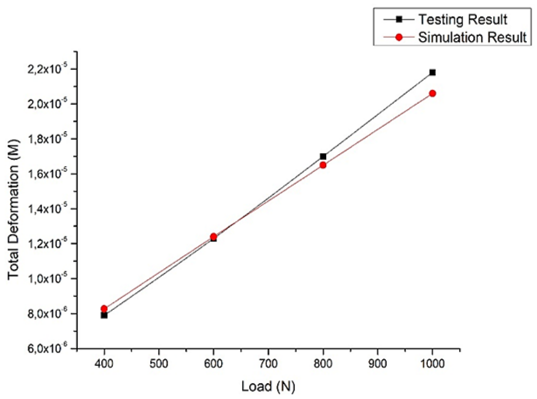


Figure 14. Results of total deformation of parametric TV furniture

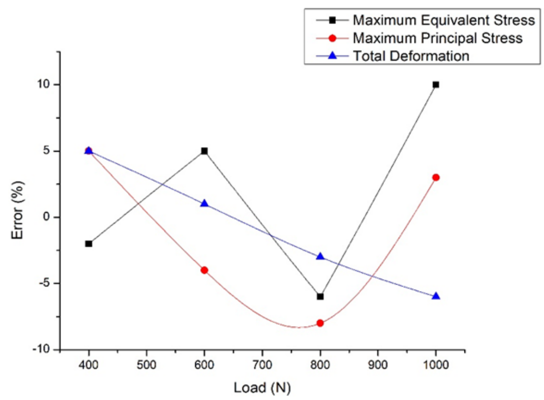


Figure 15. Error percentage comparison

The safety factor is a crucial parameter in construction or structures, indicating safety if its value is greater than 1. It is calculated by comparing the allowable stress of the material with the maximum stress that occurs. If the maximum stress is smaller than the allowable stress, the material considered safe [26]. Given the yield strength of the material is 140 MPa and the safety factor is 1.8, the allowable stress is calculated to be 77.77 Mpa. Since the maximum stress (70.5 Mpa) is less than the allowable stress, it can be concluded that the material and dimensions used are safe under the applied load.

Based on the test results, the average error factor is only 5% for both direct testing and simulation results. According to research [27], the minimum error from FEA simulations is 5%, which already indicates good meshing quality. With a 5% error and a safety factor greater than 1, the process of creating furniture prototypes and testing furniture can be replaced with the FEA method. Utilizing the FEA method can reduce costs and processing time. Table 1 illustrates the advantages of replacing the current furniture industry processes with the FEA method and simulation.

Table 1. Comparison of simulation and experimental results

Description	Current furniture industry process	Proposed furniture industry process (simulation)
Building prototype	1,327 USD	0 USD
Test	480 USD	0 USD
Total cost	1,807 USD	0 USD
Required time	1 month	1 day

Based on research conducted on parametric TV table furniture, a comparison was made between direct testing (experimental) and simulation using ANSYS software with the finite element method. This comparison was made in terms of cost and time. The stage of creating prototypes and conducting direct testing incurred a cost of 1,327 USD, and the testing cost alone was 480 USD, resulting in a total cost of 1,807 USD and a duration of 1 month. In contrast, when using FEA in ANSYS 18.1 software for prototyping and testing furniture, the cost is free (0 USD). Furthermore, in terms of time, the process of making and testing furniture through ANSYS 18.1 software simulation using the finite element method only takes 1 day. Therefore, both in terms of cost and time, utilizing ANSYS 18.1 software simulation for furniture making and testing proves to be superior to direct testing. Consequently, it is recommended to replace the prototyping process and prototype testing in the furniture manufacturing industry with the FEA method using ANSYS 18.1 software simulation. Figure 16 illustrates the recommended new process for furniture making in the furniture industry. In Figure 16(a) furniture production process before reconfigurable process and Figure 16(b) furniture production process after reconfigurable process.

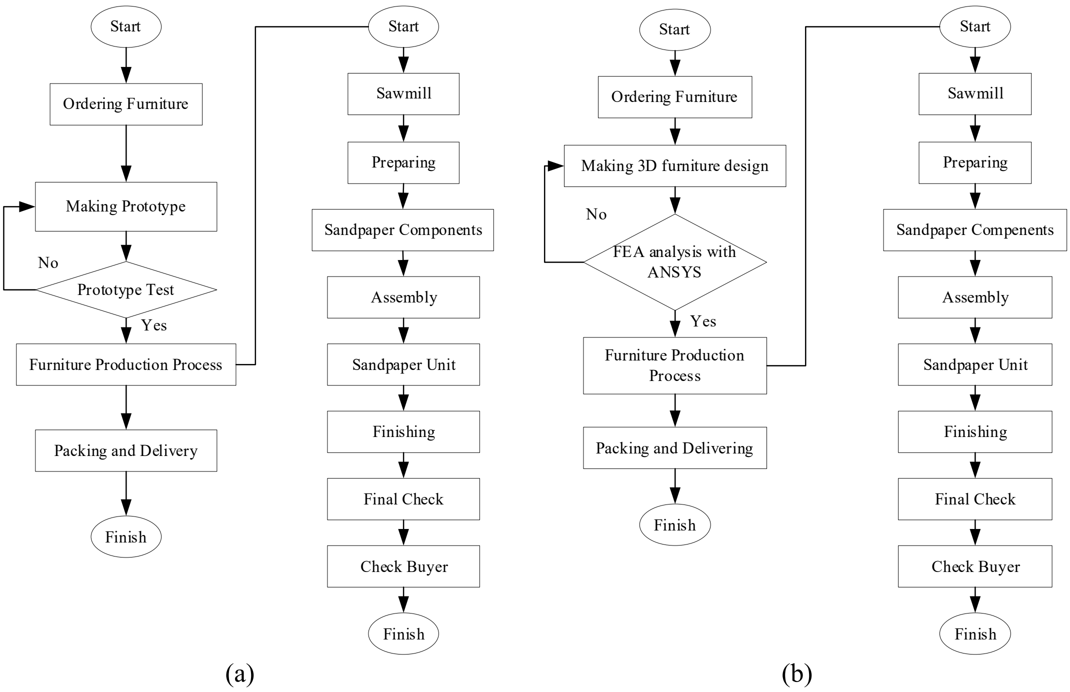


Figure 16. Changes in furniture production process (a) before using FEA metod and (b) after using FEA method

4. CONCLUSION

Implementation of FEA method on furniture prototyping process was proper simulation method. This method could replace the current inefficient furniture prototyping, which costs quite a lot and takes a long production time. This research developed FEA method with ANSYS 18.1 Workbench software for prototyping process of parametric TV table furniture through computation analysis of maximum equivalent stress, maximum principal stress, and total deformation. Load variables were 400 N, 600 N, 800 N, and 1,000 N with gravitational acceleration of 10 $\frac{m}{s^2}$. The percentage of error factor was determined between simulation result and direct strength testing of furniture prototype.

A comparison of costs and time between digital simulation and direct testing has been conducted. The fabrication and direct strength testing of furniture prototype that currently implemented costs greater amount of 1,807 USD within 1 month production time. When compared to the process of furniture prototyping and strength furniture testing using FEA method and ANSYS 18.1 Workbench software was free of cost (0 USD) and only takes 1 day production time. So that the implementation of FEA method and ANSYS 18.1 Workbench software could be used as an alternative for furniture prototyping process and product testing.

REFERENCES

[1] D. Hutton, *Fundamentals of finite element analysis,* McGrawHill Higher Education International ed., 2004.
[2] M. K. Thompson and J. M. Thompson, *ANSYS Mechanical APDL for finite element analysis*, Butterworth-Heinemann, 2017.
[3] N. S. Gokhale, S. S. Deshpande, S. V. Bedekar, and A. N. Thite, *Practical finite element analysis*, Finite to infinite, 2008.
[4] R. G. Kyle and C. R. Ludka, "Simulating the furniture industry," in *Winter Simulation Conference Proceedings*, IEEE, 2000, pp. 1347–1350. doi: 10.1109/WSC.2000.899107.
[5] N. Fariz, F. P. K. Line, Z. Amarta, N. Hutasoit, and D. Amalia, "Analysis of stress and deformation in parametric furniture using the finite element method," *E3S Web of Conferences*, vol. 465, p. 02032, Dec. 2023, doi: 10.1051/e3sconf/202346502032.
[6] A. Kumar, A. Sharma, R. Kumar, R. Vaish, and V. S. Chauhan, "Finite element analysis of vibration energy harvesting using lead-free piezoelectric materials: A comparative study," *Journal of Asian Ceramic Societies*, vol. 2, no. 2, pp. 138–143, Jun. 2014, doi: 10.1016/j.jascer.2014.02.001.
[7] A. Muhammad, M. A. H. Ali, and I. H. Shanono, "Finite element analysis of a connecting rod in ANSYS: an overview," *IOP Conference Series: Materials Science and Engineering*, vol. 736, no. 2, Jan. 2020, doi: 10.1088/1757-899X/736/2/022119.
[8] K. Rizal and A. Syaifudin, "Evaluation of crash energy management of the first-developed high-speed train in Indonesia," *Journal of Engineering and Technological Sciences*, vol. 55, no. 3, pp. 235–246, Aug. 2023, doi: 10.5614/j.eng.technol.sci.2023.55.3.2.
[9] A. Triarso and B. Dwipurwanto, "Stability analysis of gabion pair project for beach embankment in Kenjeran Residential area (in Indonesian: *Analisis stabilitas pada proyek pasangan bronjong untuk tanggul pantai di area permukiman Kenjeran*)," *Jurnal Ilmiah MITSU*, vol. 9, no. 2, pp. 73–82, Oct. 2021, doi: 10.24929/ft.v9i2.1084.
[10] S. Ö. Felek, "Parametric modelling in furniture design a case study: two door wardrope," *The European Journal of Research and Development*, vol. 2, no. 2, pp. 62–74, Jun. 2022, doi: 10.56038/ejrnd.v2i2.29.
[11] M. E. Bechly and P. D. Clausen, "Structural design of a composite wind turbine blade using finite element analysis," *Computers and Structures*, vol. 63, no. 3, pp. 639–646, May 1997, doi: 10.1016/S0045-7949(96)00387-2.
[12] N. A. Fallah, C. Bailey, M. Cross, and G. A. Taylor, "Comparison of finite element and finite volume methods application in geometrically nonlinear stress analysis," *Applied Mathematical Modelling*, vol. 24, no. 7, pp. 439–455, Jun. 2000, doi: 10.1016/S0307-904X(99)00047-5.
[13] H. Patil and P. V. Jeyakarthikeyan, "Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS," *IOP Conference Series: Materials Science and Engineering*, vol. 402, no. 1, pp. 1–12, Oct. 2018, doi: 10.1088/1757-899X/402/1/012065.
[14] S. Saha and A. Roychowdhury, "Application of the finite element method in orthopedic implant design," *Journal of Long-Term Effects of Medical Implants*, vol. 19, no. 1, pp. 55–82, 2009, doi: 10.1615/jlongtermeffmedimplants.v19.i1.70.
[15] Z. Wen, X. Tin, and W. Zhang, "Contact-impact stress analysis of rail joint region using the dynamic finite element method," *Wear*, vol. 258, no. 7–8, pp. 1301–1309, Mar. 2005, doi: 10.1016/j.wear.2004.03.040.
[16] J. P. Srivastava, P. K. Sarkar, and V. Ranjan, "Contact stress analysis in wheel–rail by hertzian method and finite element method," *Journal of The Institution of Engineers (India): Series C*, vol. 95, no. 4, pp. 319–325, Oct. 2014, doi: 10.1007/s40032-014-0145-x.
[17] A. Pratama and D. Agusman, "Analysis of frame construction strength in belt conveyor design using ANSYS Workbench," *Sainteks: Jurnal Sains dan Teknik*, vol. 5, no. 1, pp. 19–28, Mar. 2023, doi: 10.37577/sainteks.v5i1.527.
[18] E. Gardie, V. Paramasivam, H. Dubale, E. Tefera Chekol, and S. K. Selvaraj, "Numerical analysis of reinforced carbon fiber composite material for lightweight automotive wheel application," *Materials Today: Proceedings*, vol. 46, pp. 7369–7374, 2021, doi: 10.1016/j.matpr.2020.12.1047.
[19] L. Himmlová, T. Dostálová, A. Kácovský, and S. Konvičková, "Influence of implant length and diameter on stress distribution: a finite element analysis," *Journal of Prosthetic Dentistry*, vol. 91, no. 1, pp. 20–25, Jan. 2004, doi: 10.1016/j.prosdent.2003.08.008.
[20] A. Robau-Porrua, Y. Pérez-Rodríguez, L. M. Soris-Rodríguez, O. Pérez-Acosta, and J. E. González, "The effect of diameter, length and elastic modulus of a dental implant on stress and strain levels in peri-implant bone: a 3D finite element analysis," *Bio-Medical Materials and Engineering*, vol. 30, no. 5–6, pp. 541–558, Jan. 2020, doi: 10.3233/BME-191073.
[21] N. Kaleli, D. Sarac, S. Külünk, and Ö. Öztürk, "Effect of different restorative crown and customized abutment materials on stress distribution in single implants and peripheral bone: a three-dimensional finite element analysis study," *Journal of Prosthetic Dentistry*, vol. 119, no. 3, pp. 437–445, Mar. 2018, doi: 10.1016/j.prosdent.2017.03.008.
[22] Y. Li, W. J. T. Daniel, and P. A. Meehan, "Deformation analysis in single-point incremental forming through finite element simulation," *International Journal of Advanced Manufacturing Technology*, vol. 88, no. 1–4, pp. 255–267, Jan. 2017, doi: 10.1007/s00170-016-8727-9.
[23] J. Dahlberg, B. M. Phares, and Z. Liu, "Evaluation of the load distribution of timber slab bridge and the efficiency of the retrofit methods," *Engineering Structures*, vol. 291, p. 116502, Sep. 2023, doi: 10.1016/j.engstruct.2023.116502.
[24] A. A. Pratikto, "Simulation of control and monitoring of electric power of household appliances based on ESP32 (in Indonesian: *Simulasi kendali dan monitoring daya listrik peralatan rumah tangga berbasis ESP32*)," *ALINIER: Journal of Artificial Intelligence & Applications*, vol. 3, no. 1, pp. 38–48, Jul. 2022, doi: 10.36040/alinier.v3i1.4855.
[25] Y. Chen, Y. Huang, and N. Yi, "A SCR-based error estimation and adaptive finite element method for the Allen–Cahn equation," *Computers and Mathematics with Applications*, vol. 78, no. 1, pp. 204–223, Jul. 2019, doi: 10.1016/j.camwa.2019.02.022.
[26] N. Pereira and X. Romão, "Material strength safety factors for the seismic safety assessment of existing RC buildings," *Construction and Building Materials*, vol. 119, pp. 319–328, Aug. 2016, doi: 10.1016/j.conbuildmat.2016.05.055.
[27] G. Bao and P. Li, "Finite element methods," in *Maxwell's Equations in Periodic Structures*, Springer, 2021, pp. 87–161, doi: 10.1007/978-981-16-0061-6_4.

BIOGRAPHIES OF AUTHORS

Fesa Putra Kristianto received S.T. and M.MT. at the Sepuluh Nopember Institute of Technology. He is a full-time lecturer at the Furniture and Wood Processing Industry Polytechnic. He has also worked as coworker with Sepuluh Nopember Institute of Technology. He has published several research papers in various international journal and international conference. His research areas are operations management, quality management and industrial management. He can be contacted at email: fesa.putra@poltek-furnitur.ac.id.

Zain Amarta received his bachelor degree in engineering physics at the Sepuluh Nopember Institute of Technology and his master degree in industrial management at Sepuluh Nopember Institute of Technology. He is full time lecturer at the Furniture and Wood Processing Industry Polytechnic. His research lines are industrial and engineering management. He can be contacted at email: zain.amarta@poltek-furnitur.ac.id.

Nicolas Hutasoit received his S.Pi. degree in fisheries product technology at the Bogor Agricultural University, and his M.T. in industrial engineering at the University of Indonesia. He is full-time lecture at the Department of Furniture Design, Furniture and Wood Processing Industry Polytechnic. His research lines are product development, factory work system, automatic control and energy efficiency audit. He can be contacted at email: nicolas.hutasoit@poltek-furnitur.ac.id.

Nuthqy Fariz is a graduate of a bachelor of engineering and has an S.T. degree from architecture education (UNDIP) in 2015, and earned an M.Ars. degree in 2018 with a concentration in advanced architectural science. He has conducted research in the field of advanced architectural design and also deepened the design of furniture design. He has also conducted scientific publications in accredited national journals and international seminars. His research fields are design, architectural design, and furniture design. He can be contacted at email: nuthqy.fariz@poltek-furnitur.ac.id.

Fania Putri Herinda is a student of the Department of Furniture Industry Business Management, Furniture and Wood Processing Industry Polytechnic. She is now in 3rd semester. She is studying business management, marketing, and quality control. She has achieved a bronze medal in the International Festival of Engineering, Science, and Technology in Tunisia (IFEST2) competition and a silver medal in the Indonesia National Science Enterprise Challenge (INASEC) competition. She can be contacted at email: fputriherinda@student.poltek-furnitur.ac.id.