



Journal home page: <http://ajarcde-safe-network.org> ISSN 2581-0405

Finite Element Analysis of the Supporting Frame of a Solar-Powered Maize Sheller for Smallholder Farmers

Tri Satya Ramadhoni, Hendradinata, Toni Okviyanto, Firdaus, Agis Fika Wulandari, Indra Syahputra and Stevan Juliandro Aruan

Department of Mechanical Engineering, Politeknik Negeri Sriwijaya, Palembang 30139, Indonesia

ARTICLE INFO

Article History:

Received: 01 August 2026

Final Revision: 27 August 2026

Accepted: 01 September 2026

Online Publication: 04 September 2026

KEYWORDS

Finite Element Analysis, structural integrity, supporting frame, maize sheller, agricultural machinery

CORRESPONDING AUTHOR

*E-mail: tri.satya.ramadhoni@polsri.ac.id

A B S T R A C T

The structural integrity of agricultural machinery is essential for ensuring safe and reliable operation under service loading conditions. Although the performance of the authors' previously developed solar-powered maize sheller has been experimentally evaluated, the structural behavior of its supporting frame has not yet been investigated. Therefore, this study aims to evaluate the supporting frame's structural performance using the Finite Element Analysis (FEA) method. A three-dimensional numerical model of the frame was developed in SolidWorks Simulation based on the fabricated prototype. The supporting frame was modeled using beam elements, while commercial machine components, including the photovoltaic panel, hopper assembly, shelling cylinder assembly, electric motor, battery, inverter, and solar charge controller, were represented by equivalent static loads applied at their actual mounting locations. The finite element model considered the self-weight of the installed machine components under static operating conditions. The simulation results indicate that the maximum equivalent Von Mises stress developed in the frame is 12 MPa, which is substantially lower than the yield strength of the structural material (221 MPa). The maximum total deformation is 0.20 mm, while the minimum factor of safety reaches 19, indicating that the supporting frame possesses a considerable structural safety margin. These findings demonstrate that the proposed frame design is structurally adequate to support all installed machine components without exceeding the allowable material strength. Furthermore, the developed numerical modeling approach provides an efficient and practical framework for evaluating and optimizing the structural performance of agricultural machinery during the design stage before prototype modification or experimental structural testing.

Contribution to Sustainable Development Goals (SDGs):

SDG 2: Zero Hunger

SDG 7: Sustainable Cities and Communities

SDG 9: Industry, Innovation and Infrastructure

1. INTRODUCTION

1.1. Research Background

The agricultural sector plays a strategic role in supporting food security and economic growth in developing countries. In addition to providing food supplies, agriculture serves as a major

source of raw materials for food processing industries and livestock feed production [1]. One of the most important agricultural commodities in Indonesia is maize (*Zea mays* L.), which is widely utilized for food products, animal feed, and industrial applications. Maize can be processed into various products such as corn flour, corn rice, cooking oil, snacks, and other value-added products [2].



This work is licensed under a Creative Commons Attribution-NonCommercial-ShareAlike 4.0 International License

Published under licence by SAFE-Network

South Sumatra Province is one of Indonesia's major maize-producing regions. According to Statistics Indonesia data, maize production in South Sumatra reached 859,846 tons in 2019. Despite this substantial production, improving post-harvest efficiency remains a major challenge in supporting the national maize self-sufficiency program [1].

Shelling is one of the most critical post-harvest processes in maize production. In many rural areas, shelling is still done manually or with simple equipment that has limited capacity and low productivity. Implementing mechanical maize shellers can increase processing capacity, reduce post-harvest losses, and improve farmers' operational efficiency [3]. However, limited access to electricity in rural and remote areas remains a major challenge for operating electrically driven maize shelling machines. The utilization of solar energy through photovoltaic systems offers a sustainable alternative solution due to Indonesia's high solar irradiation potential throughout the year [4]. Beyond shelling performance and energy efficiency, structural reliability is a key factor affecting machine durability and operational safety. The frame supports all machine components, including the driving motor, shelling mechanism, solar power system, and auxiliary equipment. Structural failure from static or dynamic loading can reduce machine performance and increase maintenance costs [5].

Finite Element Analysis (FEA) is widely used to evaluate structural strength and integrity. This method enables engineers to predict stress distribution, deformation, and safety factors before physical testing, reducing development time and costs [6]. Therefore, this study aims to analyze the structural strength of a solar-powered maize sheller frame using the FEA method to ensure that the designed structure can operate safely and reliably under the intended working conditions.

1.2. Literature Review

Mechanical maize shellers have been widely developed to improve post-harvest efficiency compared with manual shelling methods. Based on their operating mechanisms, maize shelling can be classified as manual or mechanical. Manual shelling relies on human labor, resulting in lower productivity and longer processing times. In contrast, mechanical shelling employs electric motors or internal combustion engines, enabling higher processing capacity while reducing operator workload [6].

Recent technological developments have encouraged the adoption of electric motors as the primary driving source for maize shelling machines. Electric motor-driven systems provide more stable rotational speeds, resulting in more consistent shelling performance. However, electrically powered machines remain challenging to use in rural areas with limited access to electrical infrastructure. Consequently, integrating photovoltaic energy systems has become a promising alternative for supporting sustainable agricultural machinery in remote communities [7].

Apart from the power source, the structural frame plays a crucial role in determining machine reliability. The frame supports the driving motor, shelling mechanism, transmission components, photovoltaic system, and other auxiliary equipment. Poor structural design may result in excessive deformation or structural failure during machine operation [5].

Finite Element Analysis (FEA) has been widely used as a numerical approach to evaluate structural performance in engineering applications. This method enables the prediction of

stress distribution, deformation, strain, and safety factors before prototype fabrication or physical testing, thereby reducing product development costs and time. Although finite element analysis has been widely applied in the structural assessment of agricultural machinery, studies specifically investigating the structural behavior of solar-powered maize sheller frames under equivalent static loading conditions remain limited. Furthermore, information on the structural reliability of integrated photovoltaic-powered shelling systems remains scarce in the available literature [8], [14].

Recent studies have increasingly applied Finite Element Analysis (FEA) to evaluate and optimize the structural performance of agricultural machinery frames before prototype fabrication. Meena et al. employed FEA to assess the frame of a tractor-operated maize cobs picker and demonstrated that numerical simulation effectively identified critical stress and deformation regions while confirming adequate structural strength for field operation [10]. More recently, Yu et al. optimized the chassis framework of a self-propelled peanut planter using finite element analysis combined with sensitivity analysis and multi-objective optimization, achieving significant weight reduction while maintaining structural safety [11]. Similarly, recent reviews have highlighted that FEA has become an essential design tool for improving the strength, reliability, and material efficiency of agricultural machinery operating under complex loading conditions [12]. Unlike these optimization-oriented studies, the present work focuses on validating the structural integrity of an existing solar-powered maize sheller prototype using equivalent static loading conditions representative of actual machine operation.

Therefore, this study investigates the structural performance of a solar-powered maize sheller frame using Finite Element Analysis. The results are expected to provide insights into stress distribution, deformation characteristics, and structural safety, thereby supporting the development of reliable agricultural machinery for rural communities.

Beyond its technical contribution, developing a structurally reliable solar-powered maize sheller also supports broader sustainable development objectives. By utilizing renewable energy to operate post-harvest machinery, the proposed system contributes to affordable and clean energy (SDG 7). Improving the structural reliability of agricultural machinery advances technological innovation and sustainable rural infrastructure (SDG 9), while increasing access to efficient maize shelling technology for smallholder farmers supports improved food security and reduced post-harvest losses (SDG 2).

Although previous experimental studies [9] demonstrated satisfactory shelling capacity, shelling efficiency, and energy performance, the prototype's structural reliability has not yet been systematically evaluated. During operation, the supporting frame carries the combined loads of the photovoltaic module, electric motor, transmission system, and shelling mechanism. Excessive deformation or insufficient structural stiffness may cause component misalignment, reduced transmission efficiency, premature fatigue damage, and shortened service life. Therefore, evaluating the structural behavior of the supporting frame through Finite Element Analysis is essential to verify that the prototype possesses sufficient structural strength and safety for long-term operation.

1.3. Research Objective

The objective of this study is to evaluate the structural strength of a solar-powered maize sheller frame using the Finite Element Analysis (FEA) method. The analysis aims to determine the stress distribution, structural deformation, and factor of safety of the frame under equivalent static loads representing the installed machine components. The findings are expected to inform the machine's structural reliability and serve as a reference for developing more reliable agricultural machinery for smallholder farmers.

2. MATERIALS AND METHODS

2.1. Research Workflow

This study employed a numerical simulation approach to evaluate the structural strength of the supporting frame of a solar-powered maize sheller. The research began by identifying the machine specifications based on the prototype developed in the authors' previous study [9]. The authors developed a three-dimensional CAD model of the supporting frame in SolidWorks based on the fabricated prototype and available engineering drawings, then imported it into SolidWorks Simulation for finite element analysis (FEA).

The simulation procedure involved assigning material properties to the structural members, estimating the masses of the mounted machine components, generating a beam-element finite element mesh, defining boundary conditions, applying equivalent static loads representing the self-weight of the installed components, and performing static structural analysis. The structural performance was then evaluated in terms of equivalent stress, total displacement, and factor of safety to assess the frame's structural reliability [8].

2.2. Solar-Powered Maize Sheller

This study focused on a solar-powered maize sheller previously developed and experimentally evaluated by the authors [9]. The machine was designed as a post-harvest technology for smallholder farmers by utilizing electricity generated from a photovoltaic system, allowing operation in rural areas with limited access to the electrical grid.

Table 1. Main Specifications of the Solar-Powered Maize Sheller

No	Parameter	Specification
1	Machine Type	Solar-powered maize sheller
2	Machine Function	Maize shelling
3	Power Source	Photovoltaic (Solar Energy)
4	Solar Panel Capacity	200 Wp
5	Battery	12 V, 40 Ah
6	Inverter	500 W
7	Electric Motor	AC Motor, 125 W
8	Transmission System	Belt and Pulley
9	Frame Material	ST37 Structural Steel
10	Rated Shelling Capacity	65 kg/h



Fig. 1. Three-dimensional assembly model of the solar-powered maize sheller.

The machine consists of a photovoltaic panel, solar charge controller, battery, inverter, AC electric motor, shelling mechanism, belt-pulley transmission system, and a steel frame supporting the major machine assemblies during operation. While the previous study focused on shelling capacity, shelling efficiency, energy consumption, kernel damage, and economic feasibility [9], the present study focuses exclusively on evaluating the supporting frame's structural response under equivalent static loads generated by the installed machine assemblies using Finite Element Analysis.

2.3. Frame Geometry

The main frame serves as the primary structural component supporting all machine elements during operation. A three-dimensional CAD model was developed in SolidWorks based on the fabricated prototype and the available engineering drawings. The numerical model preserves the overall structural configuration, component mounting locations, and principal load-transfer paths of the actual machine.



Fig. 2. Three-dimensional CAD model of the solar-powered maize sheller frame.

Figure 2 presents the three-dimensional CAD model of the solar-powered maize sheller frame developed in SolidWorks for structural analysis. All structural joints were assumed to be perfectly welded. Accordingly, the connections between beam members were considered fully rigid in the beam-element finite element model. Commercial components, including the photovoltaic module, hopper assembly, shelling cylinder assembly, electric motor, battery, inverter, and solar charge controller, were not modeled as deformable bodies. Instead, we represented their self-weights as equivalent static loads applied at their respective mounting locations. This modeling strategy is commonly adopted in structural finite element analysis to reduce

computational complexity while maintaining reliable stress predictions [8], [9].

Minor geometric simplifications were introduced where detailed fabrication dimensions were unavailable or where adjustable structural features did not significantly influence the global structural behavior of the frame. In such cases, we adopted equivalent structural members from the SolidWorks Weldment library to maintain model continuity and numerical stability during the finite element analysis. These simplifications preserve the overall geometry, component mounting positions, and principal load-transfer paths of the machine and therefore have negligible influence on the global structural response. The structural members employed in the finite element model are summarized in Table 2. We adopted standard structural profiles from the SolidWorks Weldment library to construct the numerical model. Where complete fabrication dimensions were unavailable, we selected equivalent structural sections to preserve the overall geometry, structural configuration, and principal load-transfer paths of the fabricated prototype. Consequently, these simplifications have negligible influence on the predicted global structural response.

Table 2. Structural Members Used in the Finite Element Model

No	Structural Member	Section Type	SolidWorks Weldment Profile
1	Main frame	Angle iron	35 × 35 × 5 mm
2	Intermediate frame	Angle iron	35 × 35 × 5 mm
3	Intermediate frame supports	Round pipe	35 × 35 × 5 mm
4	Solar panel supports	Round pipe	Ø21.3 × 2.3 mm

2.4. Material Properties

The mechanical properties assigned in the finite element model are summarized in Table 3. Since ST37 structural steel is not available in the default SolidWorks material library, Plain Carbon Steel was selected to represent the structural frame because its mechanical properties are comparable to those of ST37 structural steel. The remaining materials were assigned according to the standard SolidWorks material database to estimate the masses of the machine components required for the equivalent static load calculations. The material properties were adopted from the SolidWorks material library and standard engineering references [13]–[16].

The masses of commercial components that were not modeled as deformable bodies, namely the photovoltaic panel and the lead-acid battery, were obtained from the technical specifications of commercially available products with equivalent ratings. We selected a 200 Wp monocrystalline photovoltaic module and a 12 V, 20 Ah valve-regulated lead-acid (VRLA) battery to represent the prototype configuration. Based on the selected commercial specifications, the photovoltaic module and battery have approximate masses of 12.0 kg and 5.9 kg, respectively [17], [18].

The assigned material properties served two different purposes in the numerical model. For structural members explicitly included in the finite element model, the material properties were used to calculate the stiffness, stress, and deformation responses during the structural analysis. For

commercial machine components that were not modeled as deformable bodies, the assigned materials were used solely to estimate their masses using the Mass Properties feature in SolidWorks. The calculated component masses were subsequently converted into equivalent static loads and applied to the supporting frame at the corresponding mounting locations during the finite element simulation, as described in Section 2.6.

Table 3. Mechanical Properties of Materials Used in the Finite Element Model

No	Material	Density (kg/m ³)	Young Modulus (GPa)	Poisson's Ratio	Yield Strength (MPa)
1	Plain Carbon Steel	7800	210	0.28	220.59
2	Cast Carbon Steel	7800	200	0.32	248.16
3	Gray Cast Iron	7200	66.17	0.27	-
4	Aluminium Alloy 6061-T6	2700	69	0.33	275
5	ABS	1020	2	0.394	-

2.5. Material Assignment

The materials assigned to each major machine component are summarized in Table 4. The frame, hopper, protective covers, outlet chute, and supporting brackets were modeled using Plain Carbon Steel, which was selected to represent the mechanical characteristics of ST37 structural steel available in the SolidWorks material library. The shelling cylinder and pulleys were assigned Aluminum Alloy 6061-T6, while the pillow block housings were assigned Gray Cast Iron. We assigned ABS Plastic to the inverter and solar charge controller enclosures. These material assignments were primarily intended to estimate the masses of individual machine components using the Mass Properties feature in SolidWorks. We then used the calculated masses to determine the equivalent static loads applied in the finite element simulation.

We assigned representative materials to commercial components, including the electric motor, photovoltaic panel, battery, inverter, and solar charge controller, solely for mass estimation. These components were not modeled as deformable bodies in the finite element analysis. Instead, we converted their calculated weights into equivalent static loads applied at their respective mounting locations on the frame.

The structural members explicitly included in the finite element model, including the main frame, welded supporting members, solar panel supports, and intermediate supporting frame, were assigned their corresponding material properties and subjected only to gravitational body loading. Therefore, the simulation automatically considered their self-weight and did not apply it again as external loads, preventing double counting of structural loads [20]. In contrast, commercial machine components that were not explicitly modeled were represented by equivalent static loads applied directly to the supporting beam members.

Table 4. Material Assignment for Major Machine Components

No	Material	Material (Solidworks Library)
1	Frame	Plain Carbon Steel
2	Outer Cover	Plain Carbon Steel
3	Inner Cover	Plain Carbon Steel
4	Shelling Cylinder	Aluminum Alloy 6061-T6
5	Shaft	Plain Carbon Steel
6	Pulleys	Aluminum Alloy 6061-T6
7	Pillow Block	Gray Cast Iron
8	Inverter Housing	ABS
9	Solar Charge Controller Housing	ABS

2.6. Boundary Conditions and Loading

To represent the machine's actual structural loading conditions, equivalent static loads were applied directly to the supporting beam members at the actual mounting locations of the corresponding machine assemblies. Table 5 presents the calculated assembly masses and their corresponding equivalent static loads. The mass of each assembly was obtained by summing the masses of its individual parts using the Mass Properties feature in SolidWorks after assigning the corresponding materials listed in Table 4. The resulting assembly masses were subsequently converted into equivalent static loads using Equation (1), and the calculated assembly masses and their corresponding equivalent static loads are summarized in Table 5.

$$F = m \cdot g \quad (1)$$

where

F is the applied force (N),

m is the component mass (kg), and

g is the gravitational acceleration (9.81 m/s²).

For loading purposes, individual machine parts were grouped into functional assemblies based on their actual load-transfer paths to the frame. Components rigidly connected and supported by the same mounting structure were treated as a single assembly. For example, the Shelling Cylinder Assembly consists of the shelling cylinder, shaft, pulleys, V-belt, and two pillow block bearings. Since the assembly is supported by two pillow block bearings, its total weight was equally distributed between the left and right bearing supports. Similarly, the Hopper Assembly includes the hopper together with its protective covers because these components are rigidly connected and mounted on the same supporting structure. This grouping simplifies the loading model while preserving the machine's actual structural loading characteristics.

The structural analysis considered only the self-weight of the installed machine components, excluding the transient load generated by maize during shelling. Accordingly, the equivalent static loads were applied vertically downward to the supporting frame members at the actual mounting regions of the corresponding machine components. The applied loads were distributed over the supporting beam surfaces according to each assembly's actual load transfer path. The corresponding assembly masses and equivalent static loads are summarized in Table 5.

Only the primary structural loads acting directly on the frame were considered in the finite element model. Smaller components, including bolts, nuts, washers, electrical cables, switches, and V-belts, were neglected because their masses are relatively insignificant and have little influence on the frame's structural response.

The operational weight of maize was not included in the simulation because the sheller uses a continuous feeding

mechanism in which individual maize ears are fed sequentially into the shelling unit rather than accumulating in the hopper. Consequently, the maize load is transient and considerably smaller than the self-weight of the installed machine components. Therefore, its influence on the global structural response of the frame was considered negligible.

Fixed support constraints were assigned to the four legs of the supporting frame to represent the machine resting on a rigid floor. All translational degrees of freedom at the support locations were constrained

$$U_x = U_y = U_z = 0$$

thereby preventing rigid body motion during the static structural analysis. Since beam elements were employed in the finite element model, the fixed supports were assigned to the beam joints located at the bottom of the four legs. Fig. 3 illustrates the complete boundary conditions adopted in this study.

Table 5. Assembly Masses and Equivalent Static Loads Applied in the Finite Element Simulation

No	Component	Qty	Mass (kg)	Force (N)	Load Application Location
1	Solar panel	1	12	117,72	Distributed to two upper support beams
2	Hopper Assembly	1	4,9	48,06	Distributed to four hopper supporting beams
3	Shelling Cylinder Assembly	1	1,73	16,67	Applied to left supporting beam
4	Shelling Cylinder Assembly	1	1,73	16,67	Applied to right supporting beam
5	Electric motor	1	3,8	37,27	Distributed to motor support beams
6	Battery	1	5,9	57,87	Applied to battery support frame
7	Inverter	1	0,62	6,08	Applied to inverter support beam
8	Solar Charge Controller	1	0,21	2,06	Applied to SCC support beam

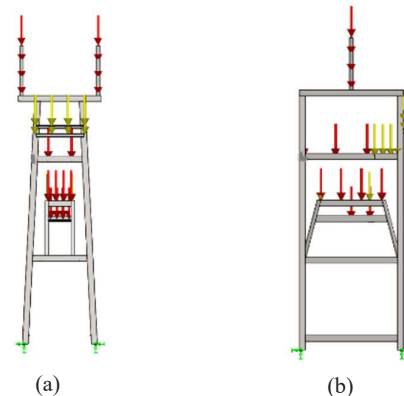
**Fig. 3.** Boundary conditions and equivalent static loads applied to the finite element model: (a) front view and (b) side view.

Figure 3 illustrates the complete boundary conditions adopted in the finite element model, including the locations of the equivalent static loads and the fixed support constraints applied to the four legs of the supporting frame. Front and side views are

presented to clearly illustrate the distribution of the applied loads throughout the structure.

The equivalent static loads were applied directly to the supporting frame beam members at the locations where the corresponding machine components are mounted. Since the finite element model was developed using beam elements, the loads were assigned to the supporting beam members rather than to bolt holes or localized fastening points. This approach provides a realistic representation of the global load transfer mechanism while avoiding artificial stress concentrations associated with localized point loading.

Since the objective of this study was to evaluate the structural behavior of the main supporting frame, commercial machine components including the photovoltaic panel, electric motor, battery, inverter, hopper assembly, and shelling cylinder assembly were represented by equivalent static loads applied directly to the supporting beam members. Secondary supporting details that do not significantly contribute to the global stiffness of the supporting frame were omitted from the finite element model. This simplification preserves the principal load transfer paths while reducing computational complexity and is commonly adopted in structural frame analyses [19].

3. RESULT AND DISCUSSION

3.1. Finite Element Mesh

The finite element model was discretized automatically using beam elements generated by the SolidWorks Simulation weldment mesher. Because beam elements represented all structural members, no manual adjustment of the solid mesh parameters was required. Figure 4 presents the beam-element finite element model generated for the supporting frame. Each structural member was represented by one-dimensional beam elements based on the centerline geometry defined in the SolidWorks Weldment environment.

This modeling approach significantly reduces the computational cost while accurately representing the global structural behavior of welded frame structures subjected to static loading [20], [21]. The automatically generated mesh consisted of 631 nodes and 616 beam elements, which were sufficient to represent the supporting frame configuration. Because the model was developed entirely using beam elements, the weldment mesher automatically generated the finite element discretization according to the beam connectivity and profile definitions.

Consequently, mesh parameters commonly required for solid elements, such as element size, curvature-based refinement, and Jacobian settings, were not applicable to this analysis. The mesh statistics obtained from SolidWorks Simulation are summarized in Table 6, while the corresponding beam-element finite element model is illustrated in Figure 4.

The generated beam mesh adequately represented the supporting frame for evaluating the global structural response under the applied equivalent static loading conditions. The adopted beam discretization was therefore considered appropriate for the objectives of the present structural analysis.

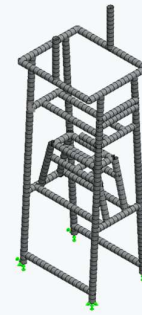


Fig. 4. Beam-element finite element model of the solar-powered maize sheller frame.

Table 6. Mesh Statistics of the Beam Finite Element Model

No	Parameter	Specification
1	Element type	Beam element (Weldment beam)
2	Meshing method	Automatic weldment mesh
3	Total Nodes	631
4	Total beam elements	616

A separate mesh convergence (mesh sensitivity) analysis was not performed in the present study because the finite element model consisted entirely of one-dimensional beam elements generated automatically by the SolidWorks Simulation weldment mesher. Unlike three-dimensional solid element models, beam-element discretization is defined by the structural member connectivity and cross-sectional properties rather than by volumetric element refinement [17], [18]. The adopted mesh, consisting of 631 nodes and 616 beam elements, produced a stable numerical solution without convergence warnings during the static structural analysis and was therefore considered adequate for evaluating the global structural behavior of the supporting frame.

3.2. Stress Distribution

Figure 5 illustrates the stress distribution obtained from the static structural analysis of the supporting frame under the equivalent static loading conditions described in Section 2.6. The stresses shown represent the combined axial and bending stresses calculated by the beam-element formulation in SolidWorks Simulation.

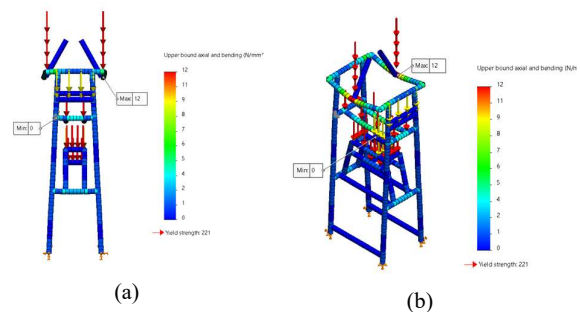


Fig. 5. Stress distribution of the supporting frame under equivalent static loading: (a) front view and (b) isometric view.

The highest stress occurred at the upper frame supporting the photovoltaic panel, particularly near the welded beam intersections where the vertical panel supports transfer the applied loads to the main frame. As shown in Figure 5, the maximum calculated stress reached 12.0 MPa, while most structural members experienced stresses below 5 MPa, indicating that the applied loads were distributed effectively throughout the supporting frame.

The stress concentration in the upper supporting frame is consistent with the machine's actual load-transfer mechanism. The photovoltaic panel produces the largest equivalent static load among all mounted components, causing the upper frame members to carry greater bending moments than the lower structural members. Additional localized stress concentrations were also observed around the supporting members of the hopper assembly and shelling cylinder assembly because these regions receive the combined loads transmitted from several mounted machine components.

The maximum computed stress remains substantially below the yield strength of the structural material (221 MPa), corresponding to only about 5.4% of the material yield strength. This result indicates that the supporting frame operates entirely within the elastic region under the applied static loading condition, and no yielding is expected during normal operation. The numerical results obtained from the static structural analysis are summarized in Table 7.

Table 7. Summary of Static Structural Analysis Results

No	Parameter	Specification
1	Maximum stress (MPa)	12.0
2	Yield strength (MPa)	221
3	Stress ratio ($\sigma_{\max}/\sigma_y$)	0.054

Although the maximum stress remains well below the material yield strength, structural adequacy should also be evaluated in terms of deformation because excessive displacement may adversely affect machine alignment during operation. Therefore, the following section discusses the displacement response of the supporting frame.

3.3. Total Deformation

Figure 6 illustrates the total deformation (resultant displacement) of the supporting frame obtained from the static structural analysis under the equivalent static loading conditions described in Section 2.6. The deformation contours represent the resultant displacement of the beam-element finite element model subjected to the self-weight of the installed machine components.

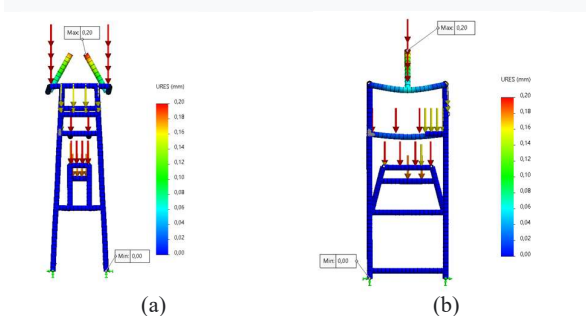


Fig. 6. Total deformation of the supporting frame under equivalent static loading: (a) front view and (b) side view.

As shown in Figure 6, the supporting frame experienced a maximum total deformation of 0.20 mm, while the minimum deformation was 0.00 mm at the fixed support locations. The maximum deformation occurred at the upper photovoltaic panel support, the structural region farthest from the fixed supports and subjected to the largest bending moment from the photovoltaic panel load. The deformation gradually decreased toward the lower structural members, indicating that the applied loads were effectively transferred through the supporting frame to the ground supports.

The deformation pattern in the front and side views shows that the supporting frame behaves as an integral welded structure, with continuous load transfer between the upper supporting members and the main frame. The largest displacement is concentrated only in the upper supporting members, whereas the middle and lower frame members exhibit considerably smaller deformations because of their shorter load paths and higher structural restraint.

The predicted maximum deformation of 0.20 mm is very small compared with the overall dimensions of the supporting frame, indicating that the frame possesses adequate structural stiffness under the applied static loading condition. This small displacement is not expected to affect the alignment of the shelling mechanism, transmission system, or photovoltaic support during normal machine operation.

Furthermore, the deformation results are consistent with the stress distribution discussed in the previous section. Both the maximum stress and maximum deformation occurred in the upper photovoltaic panel support, confirming that this region is the most highly loaded portion of the structure. Nevertheless, the relatively low stress level and very small deformation indicate that the frame remains structurally adequate for the intended operating conditions. The numerical results of the deformation analysis are summarized in Table 8.

Table 8. Summary of Total Deformation Results

No	Parameter	Specification
1	Maximum total deformation	0.20
2	Minimum total deformation	0.00
3	Location of maximum deformation	Upper photovoltaic panel support
4	Structural behavior	Stable elastic deformation

Table 8 summarizes the deformation response obtained from the finite element analysis. The supporting frame exhibited a maximum total deformation of only 0.20 mm, indicating a relatively high structural stiffness under the applied equivalent static loads. The maximum displacement occurred at the upper photovoltaic panel support because this region experiences the greatest bending effect and is farthest from the fixed supports. In contrast, displacement was zero at the fixed support locations, where all translational degrees of freedom were constrained. Overall, the deformation results demonstrate that the supporting frame maintains adequate dimensional stability during operation and is unlikely to experience excessive deflection that could adversely affect the performance of the maize sheller.

Although the supporting frame exhibited low stress and very small deformation under the applied loading conditions, structural safety should also be evaluated by considering the

available strength margin. Therefore, the factor of safety of the supporting frame is discussed in the following section.

3.4. Factor of Safety

The factor of safety (FOS) distribution obtained from the finite element analysis is presented in Fig. 7. The factor of safety was calculated as the ratio between the yield strength of the frame material and the equivalent stress developed in each structural member. Consequently, regions subjected to higher equivalent stresses exhibit lower safety factors, whereas members carrying relatively smaller loads possess larger safety margins.

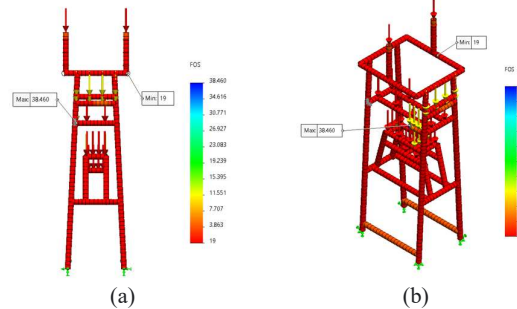


Fig. 7. Factor of safety distribution of the supporting frame: (a) front view and (b) isometric view.

As illustrated in Fig. 7, the minimum factor of safety obtained from the simulation is 19, while the maximum factor of safety reaches 38.46. The critical location is in the upper supporting members beneath the photovoltaic panel support structure, corresponding to the region where the stress distribution analysis in Section 3.2 identified the maximum equivalent stress. This relationship confirms that the factor-of-safety distribution is consistent with the stress distribution predicted by the finite element model.

The remaining structural members exhibit considerably higher safety factors because the induced stresses are substantially lower than the material yield strength. Overall, the factor-of-safety distribution indicates that the applied loads are effectively transferred through the supporting frame without producing critical stress concentrations that could compromise the machine's structural integrity. The minimum and maximum factor of safety obtained from the simulation are summarized in Table 9.

The relatively high minimum factor of safety indicates that the supporting frame possesses a substantial structural reserve under the applied equivalent static loading conditions. Although this suggests that the current frame configuration may provide a greater safety margin than required for purely static loading, the present study does not recommend immediately reducing structural member dimensions. The supporting frame is intended for practical agricultural applications where additional loading may arise from transportation, accidental impacts, operational vibration, and long-term fatigue, which the present static analysis did not consider. Nevertheless, the obtained factor of safety indicates that future studies may investigate structural optimization through profile size reduction or lightweight design while maintaining acceptable structural safety.

Table 9. Factor of Safety Obtained from the Finite Element Analysis

No	Parameter	Value
1	Minimum Factor of Safety	19.0
2	Maximum Factor of Safety	38.46
3	Critical Location	Upper photovoltaic support frame
5	Structural Assessment	Safe

The minimum factor of safety of 19 indicates that the maximum equivalent stress developed in the supporting frame remains far below the yield strength of the structural material. According to common engineering design practice, structures subjected to static loading are generally considered safe when the factor of safety exceeds 2. Therefore, the supporting frame has a substantial safety margin and can support all installed machine components under the specified loading conditions. Although the frame demonstrates excellent structural reliability, the relatively high factor of safety also suggests that certain structural members may be conservatively designed. Consequently, future studies may investigate structural optimization through material reduction or profile modification while maintaining an adequate safety margin.

From a practical perspective, the present study demonstrates that a locally fabricated supporting frame made from ST37 structural steel has sufficient strength to safely support the integration of renewable energy components, including the photovoltaic module, battery, inverter, and electric motor. These findings provide engineering evidence that low-cost fabrication methods commonly adopted by local workshops can achieve adequate structural reliability for solar-powered agricultural machinery. Consequently, the proposed design offers a practical reference for developing affordable and sustainable post-harvest technologies for smallholder farmers.

4. CONCLUSION

This study evaluated the structural performance of the supporting frame of a solar-powered maize sheller previously developed by the authors using Finite Element Analysis (FEA). A three-dimensional numerical model was established in SolidWorks Simulation by representing the commercial machine components as equivalent static loads while explicitly modeling the supporting frame using beam elements. This modeling approach enabled the structural response of the frame to be evaluated efficiently while preserving the actual load transfer paths between the mounted components and the supporting structure. The simulation results showed that the maximum equivalent Von Mises stress developed in the frame was 12 MPa, which is considerably lower than the yield strength of the assigned structural material (221 MPa). The maximum total deformation was 0.20 mm, indicating that the supporting frame possesses adequate stiffness under the specified static loading condition. Furthermore, the minimum factor of safety from the analysis was 19, showing that the frame has a substantial safety margin and can support all installed machine components under normal operating conditions. The adopted finite element modeling strategy proved effective for evaluating the structural integrity of the supporting frame without requiring repeated prototype modifications or destructive structural testing. The findings confirm that the proposed frame design satisfies the structural requirements for the

intended application and can serve as a reference for the structural design of similar agricultural machinery. Future work may focus on structural optimization through profile size reduction or alternative material selection to reduce the overall frame weight while maintaining an adequate factor of safety. Experimental validation of the numerical model is also recommended to further verify the simulation results.

REFERENCE

- [1] Badan Pusat Statistik, Provinsi Sumatera Selatan Dalam Angka 2020. Palembang, Indonesia: BPS Sumatera Selatan, 2020.
- [2] Food and Agriculture Organization (FAO), Maize in Human Nutrition. Rome, Italy: FAO, 1992.
- [3] Kementerian Pertanian Republik Indonesia, Teknologi Pascapanen Jagung. Jakarta, Indonesia: Ministry of Agriculture, 2020.
- [4] International Renewable Energy Agency (IRENA), Renewable Energy Prospects: Indonesia. Abu Dhabi, UAE: IRENA, 2017.
- [5] R. G. Budynas and J. K. Nisbett, Shigley's Mechanical Engineering Design, 11th ed. New York, NY, USA: McGraw-Hill Education, 2020.
- [6] M. Raharjo and Kisdiyani, Teknologi Pascapanen Jagung. Bogor, Indonesia: Balai Penelitian Pascapanen Pertanian, 1996.
- [7] A. Sai'in, E. Saputra, W. I. Nugroho, and R. T. Hardian, "Rancang Bangun Mesin Pemipil Jagung Dua Silinder Menggunakan Motor Listrik," Jurnal Rekayasa Mesin, vol. 16, no. 3, pp. 391–400, 2021, doi: 10.32497/jrm.v16i3.3027.
- [8] O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu, The Finite Element Method: Its Basis and Fundamentals, 7th ed. Oxford, U.K.: Butterworth-Heinemann, 2013.
- [9] T. S. Ramadhoni, H. Hendradinata, A. M. Z. Fuadi, et al., "Performance analysis of a solar-powered maize sheller: Production capacity and shelling efficiency," Jurnal Polimesin, vol. 23, no. 6, pp. 820–830, 2025, doi:10.30811/jpl.v23i6.7880.
- [10] B. S. Meena, A. K. Shrivastava, and K. Bhelave, "A Finite-Element Method of Solution for Optimization of Frame of Tractor Operated Single Row Maize Cobs Picker," International Journal of Plant & Soil Science, vol. 35, no. 3, pp. 154–159, 2023, doi: 10.9734/IJPSS/2023/v35i32787.
- [11] Y. Yu, D. Yi, J. Wang, X. Tan, X. Wang, W. Dong, et al., "Lightweight Design of the Chassis Framework for a Self-Propelled Peanut Planter in Hilly Areas Based on Finite Element Analysis," International Journal of Agricultural and Biological Engineering, vol. 18, no. 5, pp. 117–126, 2025, doi: 10.25165/ijabe.20251805.9447.
- [12] Y. S. Saygılı and B. Çakmak, "Use of Finite Element Method in Agricultural Machinery and Application Examples," Journal of Agriculture Faculty of Ege University, vol. 62, no. 3, pp. 407–419, 2025, doi: 10.20289/zfdergi.1638358.
- [13] DIN Deutsches Institut für Normung, DIN 17100: Structural Steels—Quality Standards for General Structural Steels (ST37). Berlin, Germany: DIN, 1980.
- [14] D. L. Logan, A First Course in the Finite Element Method, 6th ed. Boston, MA, USA: Cengage Learning, 2017.
- [15] W. D. Callister Jr. and D. G. Rethwisch, Materials Science and Engineering: An Introduction, 10th ed. Hoboken, NJ, USA: John Wiley & Sons, 2018.
- [16] Dassault Systèmes, SOLIDWORKS Simulation User Guide. Waltham, MA, USA: Dassault Systèmes SolidWorks Corporation, 2023.
- [17] Yuasa Battery Inc., NP20-12 Valve Regulated Lead-Acid Battery Technical Manual. Reading, U.K.: GS Yuasa Corporation, 2023.
- [18] Canadian Solar Inc., CS6K-200MS Monocrystalline Photovoltaic Module Datasheet. Ontario, Canada.
- [19] K. J. Bathe, Finite Element Procedures, 2nd ed. Watertown, MA, USA: Klaus-Jürgen Bathe, 2014.
- [20] Dassault Systèmes, SOLIDWORKS Simulation Validation Manual. Waltham, MA, USA: Dassault Systèmes SolidWorks Corporation, 2023.
- [21] P. M. Kurowski, Engineering Analysis with SOLIDWORKS Simulation 2023. Mission, KS, USA: SDC Publications, 2023.