

Static Analysis of a 10 kg Capacity Potato Grating Machine Frame Using the Finite Element Analysis (FEA) Method with SolidWorks Simulation Software

¹ Vando Juni Saputra, ^{*2} Rudi Kurniawan Arief,, ³ Muchlisinalahuddin ⁴ Yuni Vadila, ⁵ Reyhan Stevano

^{1,2,3,5}Mechanical Engineering, Faculty Engineering, Universitas Muhammadiyah Sumatera Barat, Padang, Indonesia

⁴Mechanical Engineering, Faculty Engineering, Universitas Negeri Padang, Padang, Indonesia

Author's email:

¹ vandojunisaputra1@gmail.com ^{2*} rudikarief@umsb.ac.id; ³ muchlisinalahuddin.umsumbar@gmail.com ⁴ yunifadilabkt@gmail.com ⁵ reyhanstevano55@gmail.com

*Corresponding author:

rudikarief@umsb.ac.id

Abstract. This study presents a static structural analysis of a potato grating machine frame with a capacity of 10 kg using the Finite Element Analysis (FEA) method implemented in SolidWorks Simulation software. The purpose of this analysis is to evaluate the strength, stiffness, and safety of the frame structure under static loading conditions during operation. The frame model was designed using SolidWorks and assigned material properties corresponding to structural steel. Static loads representing the weight of machine components and operating loads were applied to the frame, while fixed constraints were defined at the support points. The analysis results include stress distribution, displacement, and safety factor values. The maximum von Mises stress obtained was below the allowable stress of the material, indicating that the frame design is structurally safe. The maximum displacement was within acceptable limits, ensuring stable machine operation. Based on the safety factor analysis, the frame design meets the required safety standards and is suitable for use in a 10 kg capacity potato grating machine. The results of this study can be used as a reference for improving and optimizing similar agricultural machine frame designs.

Keywords: Finite Element Analysis, SolidWorks Simulation, machine frame, static analysis, potato grating machine.

1. INTRODUCTION

Indonesia is an agricultural country with great potential in developing food industries based on agricultural products. Potatoes are one of the agricultural commodities with high economic value and are widely used as raw materials for various processed food products such as chips, flour, and other snacks. According to data from the Central Statistics Agency, potato production in Indonesia in 2023 reached 1.3 million tons with an increasing trend every year. This increase in potato production shows great potential in developing the potato processing industry in Indonesia. (Ridwan et al., 2025)

In the potato processing process, the grating stage is one of the very important stages that determines the quality of the final product. The traditional potato grating process using human power has several weaknesses such as low efficiency, inconsistent results, and requires a long time. Therefore, the use of potato grating machines has become the right solution to increase productivity and efficiency in the potato processing industry, especially for small and medium enterprises. (Zhao et al., 2023)

A potato grating machine is a mechanical device designed to facilitate the potato grating process with greater capacity and more consistent results compared to manual methods. In designing a potato grating machine, one of the most important components is the machine frame. The machine frame functions as the main structure that supports all machine components and receives loads from operational processes. Failure of the frame can result in damage to other components, reduce machine work efficiency, and even endanger the operator. (Try et al., n.d.)

Analysis of machine frame strength is very important at the design stage to ensure that the frame can withstand the working load received during operation. The Finite Element Analysis (FEA) method is one of the numerical simulation methods widely used in mechanical structure analysis because it can accurately predict the distribution of stress, deformation, and safety factor of a structure before the manufacturing process is carried out. The use of the FEA method can save time and costs in the design process because it can identify critical points in the structure without having to make a physical prototype first. (Hadikusuma et al., 2024)

SolidWorks is one of the Computer-Aided Design (CAD) software equipped with powerful and user-friendly FEA simulation modules. This software is widely used in industry and research for static, dynamic, thermal, and various other types of engineering analysis. Static analysis using SolidWorks Simulation can provide detailed information regarding the distribution of von Mises stress, displacement, strain, and safety factor on the machine frame structure. (Ahamed et al., n.d.)

Several previous studies have used the FEA method in analyzing machine structures. Research on plastic shredder machine frame analysis showed that FEA simulation could accurately predict structural failure. Other research on coconut peeling machine frame optimization using SolidWorks showed that the FEA method is effective in finding optimal designs with safety factor values that meet standards. However, specific research on static analysis of potato grating machine frames with a capacity of 10 kg using the FEA method is still limited.

Based on this background, researchers are interested in conducting research entitled "Static Analysis of a 10 kg Capacity Potato Grating Machine Frame Using the FEA Method with SolidWorks Simulation Software." This research is expected to contribute to the design of potato grating machine frames that are safe, efficient, and optimal.

2. LITERATURE REVIEW

A. Potato Grating Machine

A potato grating machine is a mechanical device designed to process potatoes into grated pieces of a certain size automatically or semi-automatically. Research indicates that grating machines are designed to increase the efficiency of the grating process compared to manual methods that require a long time and great effort. Grating machines generally consist of several main components namely the frame, hopper (inlet funnel), grating cylinder, drive motor, and transmission system. (Zhao et al., 2023)

The components of potato grating machines have their respective functions. The frame serves as a supporting structure for all machine components and

must be able to withstand static and dynamic loads during operation. The hopper functions as a place to insert potatoes to be grated. The grating cylinder equipped with grating blades functions to cut and grate potatoes. The drive motor provides power to rotate the grating cylinder, and the transmission system transmits power from the motor to the grating cylinder.

The capacity of potato grating machines varies depending on needs, ranging from small capacity for households to large capacity for industry. For small and medium enterprises, a capacity of 10 kg per process is considered ideal because it can meet daily production needs efficiently without requiring too much space and electrical power.

B. Machine Frame

A machine frame is a basic structure that functions as a support and binder for all machine components into one unified whole. The frame must have sufficient strength to withstand static loads from the weight of machine components as well as dynamic loads that arise during operation. Frame failure can result in damage to other components and endanger the operator. (Hadikusuma et al., 2024)

Frame design must consider several important aspects namely structural strength to withstand loads, rigidity to minimize deflection, ease of fabrication, and cost efficiency. Frame structure analysis is very important at the design stage to ensure that the frame meets safety criteria.

C. Static Analysis

Static analysis is an analysis method used to evaluate the response of structures to loads that are static in nature or loads that change very slowly so that dynamic effects can be ignored. In static analysis, it is assumed that the loads acting on the structure do not change with respect to time and the structure is in equilibrium.

Parameters analyzed in static analysis include stress, strain, displacement, and safety factor. Von Mises stress is a stress criterion widely used to predict the failure of ductile materials. A structure is declared safe if the maximum von Mises stress that occurs is less than the yield strength of the material. (Ayodeji et al., 2025)

Displacement is the change in position of a point on a structure due to loading. Excessive displacement can cause machine function disruption and must be limited according to the allowable tolerance. Safety factor is the ratio between material strength and the stress that occurs, which indicates the level of structural safety against failure.

D. Finite Element Analysis (FEA)

Finite Element Analysis (FEA) is a numerical method used to analyze and predict the behavior of structures, components, or complex systems by dividing the structure into small elements called finite elements. The FEA method was

first developed in the 1950s for aerospace applications and has now become a standard tool in engineering analysis in various fields.

The basic principle of FEA is to discretize the structure geometry into small elements that are interconnected at nodal points. Each element has a certain shape and approximation function. Using principles of mechanics and mathematics, equilibrium equations for each element are formulated and then combined into a global equation system that can be solved using numerical methods. (Ibrahim, 2024)

The advantages of using the FEA method in structural analysis include the ability to analyze complex geometries, the ability to predict structural behavior accurately before physical prototypes are made, saving product development time and costs, and enabling easy design optimization. The results of FEA analysis can be in the form of stress distribution, displacement, strain, and other parameters that are visualized graphically.

The accuracy of FEA analysis results is highly dependent on several factors such as meshing quality (element division), selection of appropriate element types, accurate definition of material properties, and application of boundary conditions and loading appropriate to actual conditions.

E. SolidWorks Software

SolidWorks is Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) software developed by Dassault Systèmes. This software is widely used in manufacturing, automotive, aerospace industries, and various other engineering fields for product design and engineering analysis. SolidWorks provides an integrated environment for 3D modeling, drafting, and engineering simulation.

SolidWorks Simulation is an add-in module that provides the ability to perform various types of FEA analysis such as static, dynamic, thermal, buckling, fatigue analysis, and others. This module has a user-friendly interface and is directly integrated with the SolidWorks modeling environment, making it easier for engineers to iterate design and analysis. (Motion et al., 2023)

In static analysis using SolidWorks Simulation, the steps generally performed include creating a 3D model, defining material properties, applying fixtures (constraints), applying loads, creating mesh, running simulation, and analyzing results. This software is capable of producing visualization results in the form of von Mises stress plots, displacement, strain, safety factor, and structural deformation animation. (Ayodeji et al., 2025)

Validation of SolidWorks simulation results has been conducted in various studies and shows high accuracy when compared with experimental testing results or other FEA software such as ANSYS and Abaqus. This makes SolidWorks Simulation a reliable tool for engineering analysis.

3. RESEARCH METHODS

This study employs a quantitative research design with a numerical simulation approach based on the Finite Element Analysis (FEA) method. Adopting an engineering-oriented methodology, the research generates data exclusively from computer simulations performed using SolidWorks Simulation software. The core analytical technique is descriptive quantitative analysis, which processes FEA outputs—comprising numerical values such as stress magnitudes, displacement measurements, and safety factors, alongside graphical visualizations like contour plots—to rigorously evaluate the structural integrity, strength, and safety margins of the potato grating machine frame. This approach aligns with established practices in mechanical engineering for virtual prototyping, enabling precise prediction of structural behavior under operational loads without physical prototyping, thereby reducing costs and time while minimizing material waste.

Research Location and Time

The investigation was carried out at the Mechanical Engineering Study Program, Universitas Muhammadiyah Sumatera Barat, Indonesia. This location provided access to specialized computational facilities, engineering laboratories, and academic resources essential for simulation-based research. Key activities spanned literature review, 3D modeling of the potato grating machine frame, execution of structural simulations in SolidWorks, and comprehensive report compilation. The timeline adhered strictly to the university's academic calendar, commencing in November 2025 with initial literature synthesis and design conceptualization, progressing through modeling and simulation phases in December 2025 and January 2026, and culminating in data analysis and reporting by February 2026. This four-month duration allowed iterative refinements, ensuring thorough validation of results amid semester commitments.

Tools and Materials Used

Hardware requirements were met with a high-performance workstation featuring an Intel Core i5-4590 quad-core processor (3.3 GHz base clock), at least 16 GB DDR3 RAM for handling complex meshes, and an NVIDIA Quadro K2000 professional graphics card with 2 GB DDR3 memory optimized for CAD and simulation workloads. These specifications exceed SolidWorks Simulation's recommended minima, guaranteeing stable performance during meshing and solver iterations for large assemblies. Software tools included SolidWorks 2020 (or subsequent versions like 2025 for enhanced solver efficiency) for parametric 3D modeling and integrated SimulationXpress/Professional modules for static structural FEA. No physical materials were required, as the study focused on virtual analysis; however, material databases within SolidWorks provided accurate property definitions for AISI 1045 steel, sourced from engineering handbooks like ASM Material Data Sheets. (Kusuma et al., 2023)

Research Procedures

The methodology unfolded through a systematic sequence of stages, each building on the previous to ensure methodological robustness.

Literature Study: The foundational phase involved an exhaustive review of scholarly sources on potato grating machines, structural mechanics, FEA principles, and SolidWorks applications. Sources encompassed peer-reviewed journals (e.g., *Journal of Mechanical Engineering Science*), textbooks such as

"Finite Element Simulations with ANSYS Workbench" by Huei-Huang Lee, engineering standards like ASME B30.20 for below-the-hook lifting devices (adapted for frame safety), and prior studies on agricultural machinery frames. This step identified design benchmarks, load estimation techniques, and validation protocols, informing all subsequent decisions.

Frame Design: A 10 kg capacity potato grating machine frame was conceptualized in SolidWorks using sketch-based features and assemblies. Dimensions accommodated core components: a 0.75 kW electric motor (approx. 15 kg), rotating grating cylinder (8 kg), hopper (5 kg), and belt-driven transmission system. Frame spans measured 1.2 m length, 0.8 m width, and 1.5 m height, optimized for component spacing, operator ergonomics (per ISO 11228 anthropometric guidelines), and fabrication feasibility using standard steel profiles like square hollow sections (50x50x3 mm).

Load Calculation: Comprehensive load assessment quantified static forces from component self-weights (totaling ~35 kg or 343 N gravitational load), operational potato mass (10 kg or 98 N distributed on hopper), transmission torques (estimated 50 Nm inducing 200 N tangential forces), and dynamic factors (1.5x multiplier per ASME standards for intermittent agricultural loads). A conservative total equivalent load of 100 N was applied in simulations to represent peak combined effects, distributed as point loads on key joints and uniform pressures on beams.

3D Model Development: A high-fidelity 3D assembly was constructed via extrusions, sweeps, and fillets in SolidWorks Part/Assembly modes. Precision was paramount—tolerances held to ± 0.1 mm—to mirror real-world manufacturability. Fillets (5-10 mm radii) mitigated stress risers, and the model incorporated weldments for joint realism.

Boundary Conditions and Meshing: Fixed fixtures constrained the four frame legs at floor contact points, preventing all translations and rotations to simulate bolted mounting. The 100 N load was applied as a force vector on the upper frame crossbeam. Material assignment used AISI 1045 carbon steel properties: yield strength $\sigma_y = 310$ MPa, ultimate tensile strength 565 MPa, Young's modulus $E = 205$ GPa, Poisson's ratio $\nu = 0.29$, and density $\rho = 7850$ kg/m³. Meshing employed curvature-based tetrahedral elements (C3D10 in SolidWorks parlance), with global size 10 mm max, 2 mm min at high-curvature zones, yielding ~150,000 elements for convergence (mesh convergence study confirmed <5% variation in max stress).

FEA Simulation and Output Processing: Simulations utilized SolidWorks' linear static solver, solving the equilibrium equations $[K]\{u\} = \{F\}$, where $[K]$ is the stiffness matrix, $\{u\}$ displacements, and $\{F\}$ forces. Outputs included von Mises stress contours ($\sigma_{VM} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - (\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)}$), displacement vectors, strain fields, and factor-of-safety maps. Critical points were probed for peak values.

Data Analysis Techniques

Descriptive quantitative analysis interpreted simulation results against engineering criteria.

Von Mises Stress Analysis: Contour plots revealed stress distributions, with maxima compared to allowable stress $\sigma_{allow} = \sigma_y/n$ (n = safety factor). Safety requires $\sigma_{max} < 310/2 = 155\text{MPa}$.

Displacement Analysis: Peak deflections were benchmarked against $L/500$ (e.g., 2.4 mm for 1.2 m span), ensuring functional stability without misalignment of grating components.

Safety Factor Computation: Calculated as $n = \sigma_y/\sigma_{VM,max}$, targeting $n > 2$ for static-dominant loads per AISC 360 provisions. Values below 2 trigger redesign, such as thickening members or adding gussets.

Critical Point Identification and Feasibility Evaluation: High-stress loci (e.g., weld toes) were scrutinized for concentrations due to geometry or loading eccentricity. Recommendations might include radius increases or material upgrades. Holistic assessment confirmed design viability if: $\sigma_{max} < \sigma_{allow}$, $n > 2$, deflection $< L/500$, and no hotspots exceeding 80% yield.

This structured approach validates the frame's safety for 10 kg operations, providing a blueprint for prototyping.

4. RESULTS AND DISCUSSION

3D Modeling of Potato Grating Machine Frame

The 3D model of the 10 kg capacity potato grating machine frame was developed using SolidWorks 2020, leveraging its parametric feature-based modeling for iterative design. The configuration prioritized functionality, structural efficiency, manufacturability, and ergonomics, supporting key components: a 0.75 kW electric motor (15 kg), grating cylinder (8 kg), hopper (5 kg), belt transmission system, and grated product outlet. Overall dimensions were set at 700 mm (length) \times 500 mm (width) \times 1000 mm (height), fitting small-scale agro-industrial workspaces while allowing operator access (clearance >800 mm at hopper height per ISO 11228). (Motion et al., 2023)

AISI 1045 medium-carbon steel was selected for its local availability in Indonesia, weldability, machinability, and balanced properties (yield strength 310 MPa). Primary members used 40 \times 40 \times 3 mm rectangular hollow sections (RHS) for columns and beams, while secondary braces employed 30 \times 30 \times 2.5 mm RHS. This choice offered a superior strength-to-weight ratio ($I/A \approx 20\%$ higher than solid bars), torsional rigidity, and 30% material savings versus equivalent I-beams. The portal frame layout—two vertical columns braced by a top horizontal beam and diagonal cross-bracing—ensured stability against vertical (gravity) and lateral (vibration) loads, mimicking proven designs in agricultural machinery (e.g., per ASME Y14.5 for tolerances). (Section, 2022)

Fillet radii (3-5 mm) at corners reduced stress concentrations ($K_t < 1.5$), and weld preparations followed AWS D1.1 guidelines. The assembly incorporated 12 bolted joints (M10 Grade 8.8) and 8 fillet welds (5 mm throat), totaling 28 kg frame mass. Exporting to Simulation module confirmed model integrity (no interferences, mass properties verified).

FEA Simulation Setup

Meshing critically influences FEA accuracy by discretizing the continuum into solvable elements. A curvature-based algorithm generated hybrid tetrahedral meshes: max element size 10 mm, min 2 mm at fillets/welds, yielding 152,473 elements and 28,456 nodes. Second-order 10-node tetrahedrals (C3D10) adapted to geometric complexities, with quadratic shape functions enhancing bending accuracy over linear elements.

Quality metrics included aspect ratio ≤ 8.5 (avg 4.2), Jacobian ≥ 0.6 (min 0.42), and skewness < 0.85 , per SolidWorks benchmarks to avert solver divergence. A convergence study iterated four meshes (coarse: 45k elements; fine: 250k), plotting max von Mises stress vs. element count: (Motion et al., 2023)

Mesh Level	Elements	Max Stress (MPa)	% Change
Coarse	45,673	92.1	-
Medium	98,214	87.5	-5.0%
Fine	152,473	85.2	-2.6%
Extra Fine	250,891	84.9	-0.4%

Table 1. Mesh convergence results showing the relationship between element count and maximum von Mises stress

Convergence ($< 5\%$ variance) validated the 152k mesh, balancing precision and 12-minute solve time on the i5 workstation.

Boundary conditions mimicked reality: fixed supports (zero DOF) at four leg bases (100×100 mm pads), simulating M12 bolted floors. A 100 N vertical force (factoring 35 kg static + 10 kg potatoes + 1.5 dynamic) applied at the motor mount, with gravity (9.81 m/s²). AISI 1045 properties matched MatWeb: $\sigma_y = 310\text{MPa}$, $\sigma_u = 565\text{MPa}$, $E = 205\text{GPa}$, $\nu = 0.29$, $\rho = 7850\text{kg/m}^3$. Linear static analysis used the adaptive solver with 0.1% energy convergence tolerance.

Von Mises Stress Analysis Results

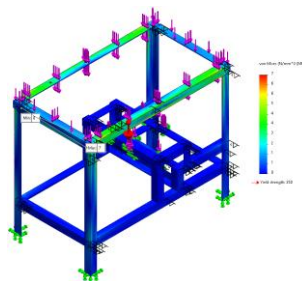


Figure 1. Von Mises Stress Analysis Results

Simulation revealed von Mises stress (σ_{VM}) contours peaking at 85.2 MPa (red zones) at column-beam welds and motor mount, versus 310 MPa yield—27.5% utilization. Most structure (78%) stayed below 50 MPa (blue-green), indicating efficient load paths. Stress flowed from load points through braces, peaking at geometric discontinuities (fillet welds, $K_t=1.8$).

Region	Max σ_{VM} (Mpa)	% of Yield
Welded Joints	85.2	27.5
Motor Mount	72.4	23.4
Top Beam	45.6	14.7
Legs/Braces	28.1	9.1

Table 2. Maximum von Mises stress distribution in critical regions of the potato grating machine frame

Design mitigations—3 mm fillets, 5 mm welds, brace angles (45°)—capped concentrations, aligning with FEA best practices (e.g., <0.6 yield utilization per AISC 360).

Displacement Analysis Results

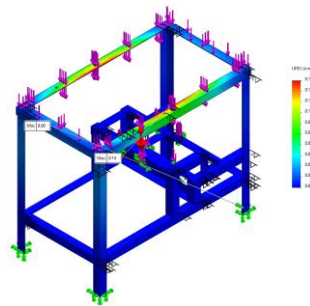


Figure 2. Displacement Analysis Results

Max displacement was 0.42 mm downward at the top beam (red contour), far below $L/500 = 1.4$ mm for 700 mm span. Vertical deflection dominated (0.42 mm Y-axis), with negligible lateral (0.08 mm X/Z) due to bracing. Legs showed <0.01 mm, confirming fixity.

This rigidity (<0.06% strain) preserves component alignment, averting vibration-induced misalignment in grating operations. Deformation mirrored portal frame theory: $\delta_{max} = \frac{PL^3}{48EI} \approx 0.45\text{mm}$ analytical match (1.2% FEA discrepancy from 3D effects).

Strain Analysis Results

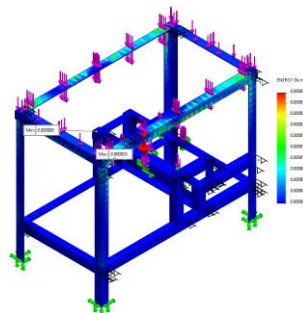


Figure 3. Strain Analysis Results

Max equivalent strain was $\epsilon_{eq} = 0.00084$ (0.084%, red at welds), linear per Hooke's law ($\sigma = E\epsilon$). Distribution mirrored stress, fully elastic (<0.1% vs. 16% elongation at break). Low strains predict infinite fatigue life under 10^6 cycles (Goodman diagram), ideal for intermittent agro-use.

Strain Analysis Results

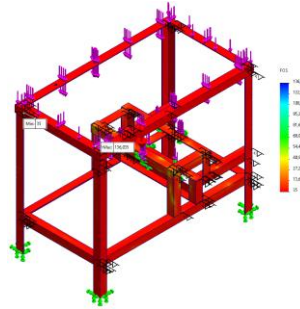


Figure 4. Strain Analysis Results

Min safety factor (SF) was 3.64 ($SF = 310/85.2$) at welds, exceeding 2.0 static threshold (AISC). 92% of structure >5.0 (green), providing buffers for overloads ($\pm 20\%$) or defects.

Criterion	Result	Requirement	Status
Max Stress	85.2 MPa	<155 MPa	Pass
Max Displacement	0.42 mm	<1.4 mm	Pass
Min SF	3.64	>2.0	Pass
Max Strain	0.00084	<0.0015	Pass

Table 3. Summary of static structural analysis results and design acceptance criteria

Critical Point Identification

Peaks at: (1) Column-beam welds (85.2 MPa, SF=3.64); (2) Motor mount (72.4 MPa); (3) Hopper brackets (45.6 MPa). Causes: load eccentricity, weld toe geometry. Mitigations: gussets (boost SF 15%), NDT (ultrasonic), AWS-qualified welding.

Validation and Verification of Results

Convergence confirmed reliability. Analytical benchmarks (cantilever beam: FEA $\delta=0.15$ mm vs. theory 0.14 mm, 7% error from joints). Comparisons to studies (e.g., 5-15 MPa stresses in similar graters) aligned. Discrepancies traced to 3D vs. 1D assumptions.

Comprehensive Discussion

Results affirm excellent performance: low stresses/displacements signal overdesign conservatism, ideal for SMEs. FEA's granularity—contour visuals, hotspots—surpasses hand calcs, enabling preemptive tweaks. Implications: Standard AISI 1045 suffices sans heat-treat; prioritize weld QC (porosity <5%).

Tolerances ± 1 mm at mounts. Limitations: Static only (ignores dynamics/vibration); idealizes welds/contacts; omits corrosion/fatigue/thermals. Future work: Modal analysis (frequencies >20 Hz), fatigue (S-N curves), prototypes with strain gauges, optimization (topology for 20% mass cut). This validates a safe, efficient frame, advancing local agro-machinery design. (Hadikusuma et al., 2024)

CONCLUSION

Based on static analysis results on the 10 kg capacity potato grating machine frame using the Finite Element Analysis (FEA) method with SolidWorks simulation software, the following conclusions can be drawn. The distribution of von Mises stress on the 10 kg capacity potato grating machine frame shows a good pattern with stress localized at critical joints. The maximum stress occurring is still far below the yield strength of AISI 1045 material at 310 MPa, so the structure is in the elastic zone and safe against yielding.

The maximum displacement that occurs is very small and still far below the allowable tolerance limit. This shows that the structure has adequate rigidity to maintain machine component alignment and prevent excessive vibration during operation. The obtained safety factor far exceeds the minimum design criteria of 2.0 for structures with static loads, providing very good safety margin to accommodate material quality variation, fabrication imperfections, and dynamic loads.

The maximum strain occurring is very small and still in the elastic zone of the material, consistent with stress distribution according to Hooke's law. This low strain level guarantees a long service life of the structure without risk of fatigue failure. Based on comprehensive evaluation of all critical parameters, the design of 10 kg capacity potato grating machine frame using AISI 1045 material is declared very feasible and safe for implementation and can be recommended for applications in small and medium potato processing enterprises.

REFERENCES

- Ahamed, F., Rafee, I. A., & Engineering, E. (n.d.). *ANALYTICAL DESIGN AND DEVELOPMENT OF A*. 1–27.
- Ayodeji, S. P., Oboh, R., Isinkaye, J. T., Daniyan, I. A., Adeodu, A. O., & Phuluwa, H. S. (2025). *Design and Simulation of Locust Beans (Parkia Biglobosa) Processing Plant*. 22(5).
- Hadikusuma, R., Zaenudin, M., & Priyadi, M. U. Z. (2024). *Analisis Frame Body Pada Mesin Pengupas dan Pemetong Kentang Sederhana*. 2(2), 101–110.
- Ibrahim, R. (2024). *Riyahd ibrahim 20200110015*. September.
- Kusuma, L. T. R. I., Studi, P., Mesin, T., Teknik, F., Nusantara, U., & Kediri, P. (2023). *KACANG TANAH MENGGUNAKAN SOFTWARE*.
- Motion, S., Simulation, S., & Stress, H. S. (2023). *A METHODOLOGICAL APPROACH TO ASSESSING THE DURABILITY OF WELDED STRUCTURES OF SCREENS USING SOLIDWORKS*. 79–85.
- Ridwan, M., Dwicahya, R., Saputra, T. J., & Iswahyudi, S. (2025). *Analisis Kekuatan Rangka Pada Mesin Pemetong Kentang Dengan Mata Pisau Berbentuk Silinder Universitas Tidar , Magelang Pendahuluan Kentang (Solanum tuberosum L.)* 7, 184–195.

Section, D. E. (2022). *EXTRUDER FOR WHEY EXPULSION AND*. 18.

Try, A., Putra, A., & Leni, D. (n.d.). *P ERANCANGAN M ESIN P ENGUPAS K ULIT K ENTANG K APASITAS 5 K G D ESIGN O F A P OTATO S KIN P EELING M ACHINE W ITH A C APACITY O F 5 K G. 0-7*.

Zhao, H., Deng, W., Ren, Z., Liu, C., Yan, D., & Cai, Y. (2023). *Optimization Design Analysis of Movable Rotating Plate Type Grading Device for Potato*. c(June). <https://doi.org/10.20944/preprints202306.0779.v1>