IJAMSET
INTERNATIONAL JOURNAL OF ADVANCED MATERIALS, SMART ENGINEERING AND TECHNOLOGY

IJAMSET | Vol. 1 No. 1 | 2026 ISSN : xxxx-xxxx

Structural Assessment of a Sand-Cast Aluminum Pulley Under Radial Loading Using Finite Element Analysis

Mulyadi Mulyadi[1*], Daddy Budiman[2], Habib Putra Fadel[3], Nofriadi Nofriadi[4], Muh Anhar[5]

[1] Department of Mechanical Engineering, Politeknik Negeri Padang, Indonesia

[2] Department of Mechanical Engineering, Politeknik Negeri Padang, Indonesia

[3] Department of Mechanical Engineering, Politeknik Negeri Padang, Indonesia

[4] Department of Mechanical Engineering, Politeknik Negeri Padang, Indonesia

[5] Department of Mechanical Engineering, Politeknik Negeri Ketapang, Indonesia

**Corresponding Author: Mulyadi@pnp.ac.id*

Received 21 June 2026; Received in revised form 28 June 2026; Accepted 30 June 2026

Abstract

A pulley is a power transmission component that must possess adequate structural strength and stiffness to ensure reliability during operation. This study aims to analyze the structural response of a sand-cast aluminum pulley under variations of radial loading using the Finite Element Analysis (FEA) method. The analysis was conducted based on the actual geometry of the sand-cast pulley, which was modeled in three dimensions and simulated under radial loads of 100 N, 200 N, and 300 N. The evaluated parameters include Von Mises stress, total deformation, and safety factor. The simulation results indicate that increasing the applied load leads to an increase in Von Mises stress from 68.85 MPa to 206.50 MPa and total deformation from 0.01943 mm to 0.05828 mm, with an almost linear trend. The maximum stress concentration consistently occurs in the keyway region of the pulley hub, while the maximum deformation is localized at the outer edge of the flange. The minimum safety factor value of 3.50 indicates that the pulley structure still has an adequate safety margin under loading conditions up to 300 N. The results of this study confirm that the use of actual sand-cast geometry can provide a representative evaluation of the pulley's structural response and can be utilized as a basis for preliminary evaluation in component design development.

Keywords: *Aluminum pulley; sand casting; structural analysis; finite element analysis; Von Mises stress*

1. Introduction

A pulley is a mechanical component that functions to transmit rotational motion and power through a belt system from one shaft to another. This component is widely used in industrial machinery, conveyor systems, agricultural equipment, and various other power transmission systems. During operation, a pulley is subjected to loads that can induce stress and deformation within its structure. If the stress distribution is not uniform, stress concentration may occur in certain regions, potentially leading to crack initiation, permanent deformation, or even component failure. Therefore, the structural capability of a pulley to withstand applied loads must be analyzed to ensure safe operation and extend its service life [1]. Aluminum is one of the most commonly used materials for pulley manufacturing due to its low density, excellent corrosion resistance, and high strength-to-weight ratio. The use of aluminum can reduce the component weight without significantly compromising its mechanical performance. In addition, aluminum is easily formed through casting processes, making it suitable for manufacturing components with relatively complex geometries at a lower production cost compared with several other manufacturing processes [2],[3].

Sand casting is one of the most widely used manufacturing methods for producing aluminum pulleys. This method is extensively applied in the metal casting industry, particularly in small- and medium-scale enterprises, because it is capable of producing components with complex geometries at relatively low production costs. Cast components are generally considered suitable for use once they meet the required dimensional and geometric specifications. However, under these conditions, the evaluation of the structural capability to withstand operational loads is not always performed comprehensively. Consequently, the risk of stress concentration, deformation, and structural failure may increase when the pulley is subjected to loads exceeding the material's load-bearing capacity [4].

Structural analysis is required to evaluate the ability of a pulley to withstand operational loading conditions. The Finite Element Analysis (FEA) method is a numerical approach widely used to predict the structural behavior of engineering components. This method provides information on stress distribution, deformation, and safety factor through numerical simulation without requiring physical testing during the initial design stage. The results of FEA simulations can be used to identify regions experiencing stress concentration and to evaluate the structural integrity of components under the intended loading conditions, thereby enabling potential failures to be anticipated during the design phase [5],[6]. Numerous studies have been conducted on structural analysis using the Finite Element Analysis (FEA) method for various mechanical components. Existing studies generally focus on design optimization or structural evaluation of components manufactured using specific production processes. However, structural analysis of sand-cast aluminum pulleys based on the actual geometry of the manufactured product remains relatively limited [1],[7]. Furthermore, information regarding the distribution of Von Mises stress, total deformation, and safety factor under various radial loading conditions is still scarce in the literature. Such information is essential for evaluating the structural capability of pulleys to withstand operational loads and for supporting the development of safer and more reliable component designs.

Therefore, this study aims to analyze the structural behavior of a sand-cast aluminum pulley using the Finite Element Analysis (FEA) method. Simulations were performed under radial loads of 100 N, 200 N, and 300 N. The structural performance was evaluated based on the distribution of Von Mises stress, total deformation, and safety factor to determine the ability of the pulley to withstand the applied loads. The findings provide valuable information regarding the structural response of sand-cast aluminum pulleys under various radial loading conditions and serve as a reference for design evaluation and the development of similar components in the metal casting industry.

2. Materials and Methods

This study aims to evaluate the structural response of a sand-cast aluminum pulley using the Finite Element Analysis (FEA) method. The research began with a literature review to establish the theoretical foundation regarding the characteristics of pulleys, aluminum materials, the sand casting process, and the application of the FEA method in structural analysis [8]. The sand-cast pulley was then measured to obtain its actual dimensions, which served as the basis for developing a three-dimensional model. The geometric model was subsequently analyzed using finite element simulation under radial loads of 100 N, 200 N, and 300 N. The simulation results were evaluated based on the distribution of Von Mises stress, total deformation, and safety factor to assess the structural capability of the pulley to withstand the applied loads. The research workflow adopted in this study is illustrated in Figure 1.

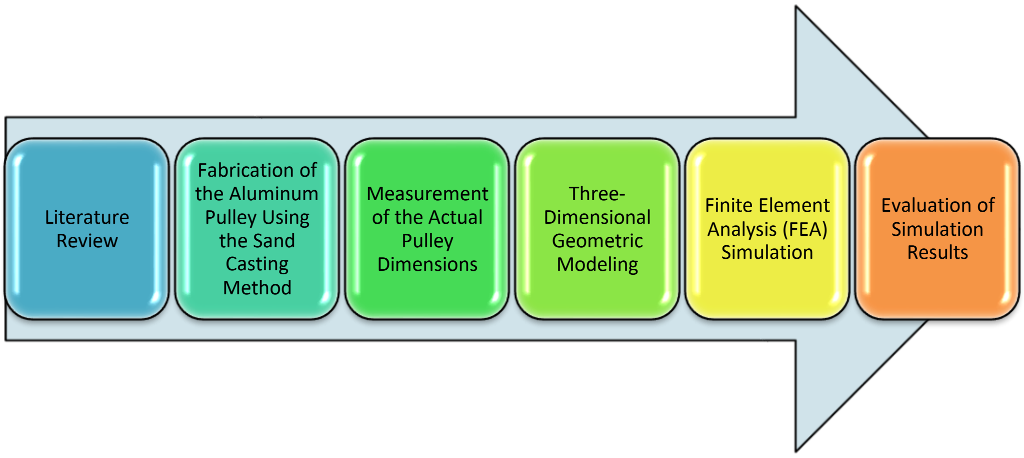


Figure 1. Research Framework

2.1 Literature Review

A literature review was conducted to establish the theoretical foundation regarding pulley design, the characteristics of aluminum materials, the sand casting process, and the application of the Finite Element Analysis (FEA) method in structural analysis. The literature sources included reference books, engineering standards, and relevant scientific articles. The information obtained served as the basis for developing the research methodology, geometric modeling, determining the simulation parameters, and analyzing the simulation results.

2.2 Fabrication of the Aluminum Pulley Using the Sand Casting Method

The aluminum pulley used as the research object was manufactured using the sand casting method. The manufacturing process consisted of sand mold preparation, aluminum melting, pouring molten metal into the mold cavity, solidification, mold removal, and finishing. Observations were carried out at each stage of the manufacturing process to ensure that the produced pulley conformed to the intended geometric design and was suitable for structural simulation [9].

2.3 Measurement of the Actual Pulley Dimensions

Dimensional measurements were performed to obtain the actual geometry of the sand-cast pulley, which served as the basis for developing the three-dimensional model. The manufactured pulley was measured for several key parameters, including the outer diameter of the pulley, groove root diameter, hub diameter, bore diameter, pulley width, and groove width. The measurements were carried out using appropriate measuring instruments to obtain dimensions that accurately represent the actual condition of the sand-cast product. The physical appearance of the pulley and the measured dimensions are presented in Figure 2.

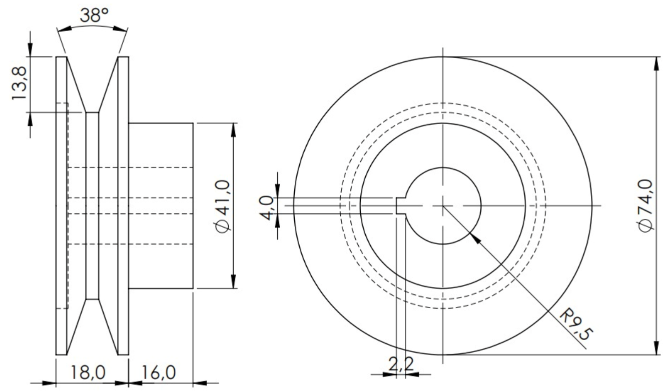


Figure 2. Actual dimensions of the sand-cast aluminum pulley.

The measured dimensional data were subsequently used as the reference for developing the three-dimensional geometric model, ensuring that the simulation model accurately represented the actual geometry of the sand-cast pulley. The values of each dimensional parameter used in this study are presented in Table 1.

Table 1. Actual Dimensions of the Sand-Cast Aluminum Pulley

Parameter	Symbol	Value
Outer diameter	D_1	74 mm
Groove root diameter	D_2	51 mm
Hub diameter	D_3	41 mm
Bore diameter	D_4	19 mm
Pulley width	W	18 mm
Groove width	G	16 mm

2.4 Three-Dimensional Geometric Modeling

The three-dimensional pulley model was developed using Onshape software based on the actual dimensions obtained from the measurements presented in Table 1. All dimensional parameters were incorporated into the CAD model so that the resulting geometry accurately represented the actual shape of the sand-cast aluminum pulley. The completed three-dimensional model was subsequently exported to the simulation software for structural analysis using the FEA method.

2.5 Finite Element Analysis (FEA) Simulation

The three-dimensional pulley model developed in Onshape was imported into the simulation software for structural analysis using the FEA method. The pulley material was defined as aluminum, while the boundary condition was applied as a fixed constraint on the bore surface. Radial loads were applied to the pulley groove surface with load variations of 100 N, 200 N, and 300 N to evaluate the structural response under different loading conditions. The simulations were performed using tetrahedral elements with a mesh size determined through a mesh independence study. This procedure was conducted to ensure that the simulation results were not affected by variations in mesh size. The analyzed parameters included the distribution of Von Mises stress, total deformation, and the safety factor [10].

2.6 Evaluation of Simulation Results

The simulation results were evaluated based on the distribution of Von Mises stress, total deformation, and the factor of safety obtained from the finite element analysis. These three parameters were used to assess the structural response of the sand-cast aluminum pulley under radial loads of 100 N, 200 N, and 300 N. The evaluation was carried out by comparing the variation of each parameter under the different loading conditions to identify the effect of increasing load on the structural behavior of the pulley and to determine the structural suitability based on the simulation results.

3. Results and Discussion

3.1 Characteristics of the Sand-Cast Aluminum Pulley

The aluminum pulley used in this study was manufactured using the sand casting process in a small-scale metal casting industry and subsequently underwent machining on its functional surfaces to achieve the dimensions required for its intended application. The actual pulley used as the research object is shown in Figure 3.

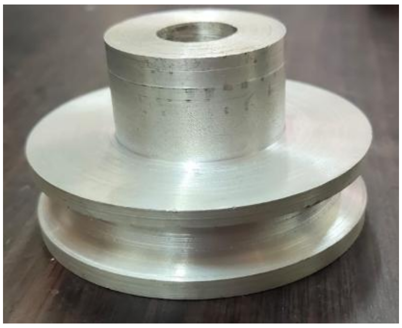


Figure 3. Actual Sand-Cast Aluminum Pulley Used in This Study

As shown in Figure 3. the pulley has a single V-shaped groove that serves as the belt transmission path, a central hub that functions as the shaft seat, and a keyway that locks the pulley to the shaft. These geometric characteristics were preserved during the three-dimensional modeling process so that the finite element simulation was performed using the actual geometry of the sand-cast component. This approach enables the distribution of Von Mises stress, total deformation, and the safety factor to be evaluated based on the actual component geometry, allowing the simulation results to more accurately represent the structural behavior of the pulley than models based on idealized geometry [11],[12].

3.2 Actual Dimensions and CAD Model of the Pulley

The next stage involved measuring the actual dimensions of the pulley to obtain the geometric data required for three-dimensional modeling. The measured dimensions of the pulley are presented in Figure 4, while the three-dimensional model developed using Onshape software is shown in Figure 5.

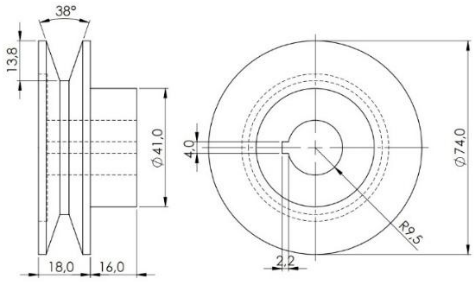


Figure 4. Geometrical dimensions of the sand-cast aluminum pulley

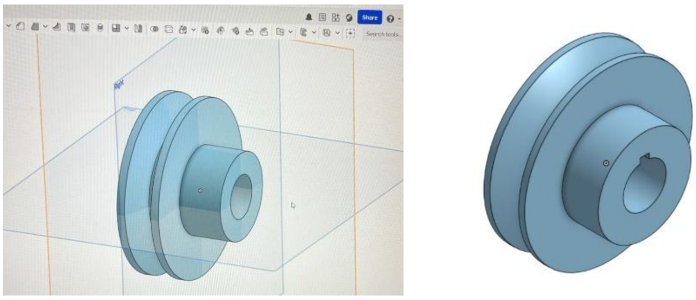


Figure 5. Three-Dimensional CAD Model of the Pulley Developed Using Onshape

The CAD model was developed based on the complete set of measured dimensions so that the geometry used in the simulation accurately represented the actual shape of the sand-cast pulley. This approach was intended to minimize discrepancies between the numerical model and the actual component, considering that the distributions of stress and deformation are highly influenced by geometric characteristics, particularly in regions with cross-sectional changes such as the hub, V-groove, and keyway. Therefore, the use of the actual geometry is expected to provide a more representative prediction of the structural response than a model developed using idealized geometry [13].

3.3 Structural Simulation Results of the Aluminum Pulley

Finite element simulations were performed to evaluate the structural response of the sand-cast aluminum pulley under radial loads of 100 N, 200 N, and 300 N. The analyzed parameters included maximum Von Mises stress, maximum total deformation, and safety factor. These parameters were used to assess the structural capability of the pulley under service loads and to identify critical regions that may be susceptible to failure. A summary of the simulation results for all loading conditions is presented in Table 2.

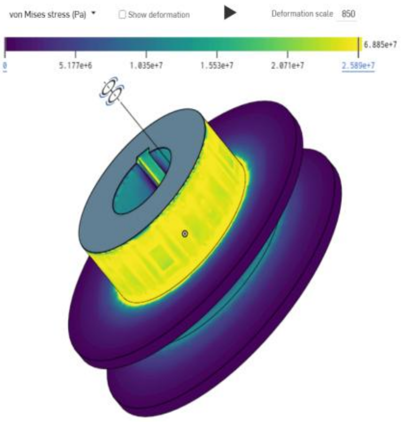
Table 2. Comparison of Simulation Results

Load (N)	Maximum Von Mises Stress (MPa)	Maximum Displacement (mm)	Minimum Safety Factor
100	68.85	0.01943	3.50
200	137.70	0.03885	3.50
300	206.50	0.05828	3.50

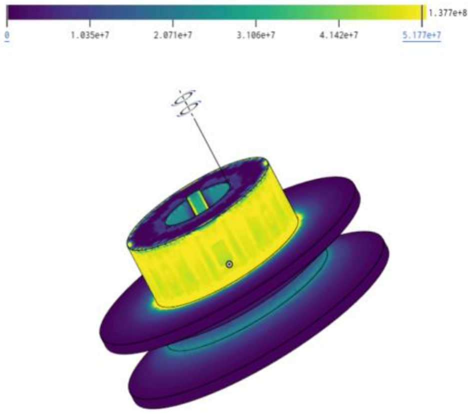
Table 2 shows that the increase in radial loading resulted in a gradual rise in both Von Mises stress and total deformation. The maximum stress increased from 68.85 MPa under a load of 100 N to 137.70 MPa under 200 N and reached 206.50 MPa under 300 N. A similar trend was observed in the maximum deformation, which increased from 0.01943 mm to 0.05828 mm as the applied load increased. In contrast, the safety factor remained within a safe range throughout all simulation conditions. These results indicate a proportional relationship between the applied radial load and the structural response of the pulley.

3.4 Von Mises Stress Distribution

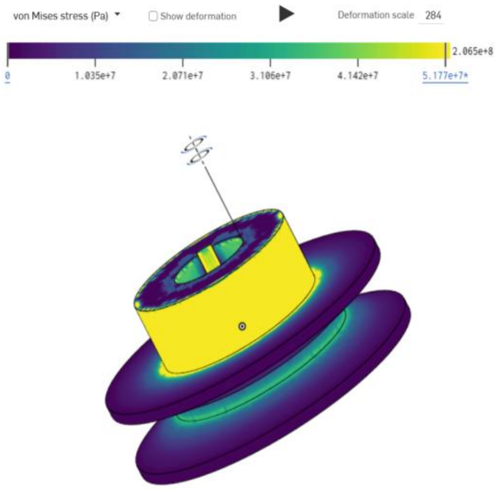
Von Mises stress distribution was used to identify critical regions experiencing the highest stress under radial loading conditions. The stress distributions obtained under radial loads of 100 N, 200 N, and 300 N are presented in Figure 6.



a. Von Mises Stress -100 N (maks. 68.85 MPa)



b. Von Mises Stress-200 N (maks. 137.70 MPa)



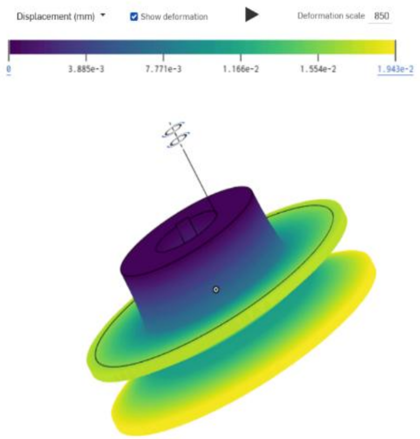
c. Von Mises Stress -300 N (maks. 206.50 MPa)

Figure 6. Von Mises Stress Distribution Under Radial Loads of 100 N (a), 200 N (b), and 300 N (c)

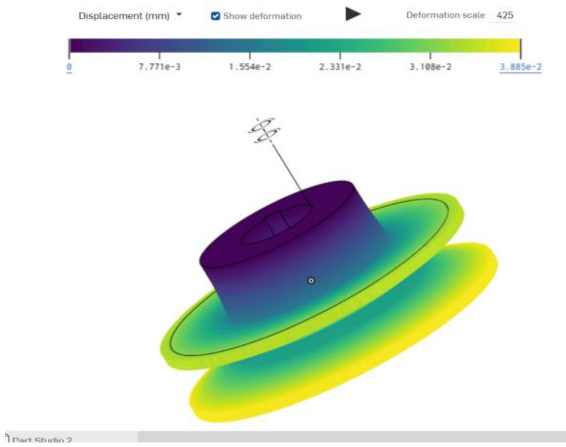
As shown in Figure 6. The stress distribution patterns under the three loading conditions exhibit similar characteristics. The maximum stress is consistently localized in the keyway region of the pulley hub. Under a radial load of 100 N, the maximum stress reaches 68.85 MPa. This value increases to 137.70 MPa under a 200 N load and reaches 206.50 MPa under a 300 N load. The location of the maximum stress remains unchanged despite the increase in the applied load, indicating that the keyway region is the critical area of the pulley structure. The nearly proportional increase in stress with increasing load indicates that the structure continues to behave within the linear elastic region. This phenomenon suggests that the stress distribution follows the principles of mechanics of materials, in which an increase in the applied external load results in a proportional increase in stress as long as the material has not reached its yield condition [14].

3.5 Total Deformation Analysis

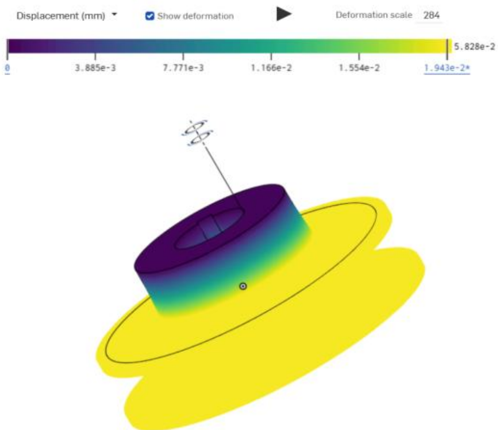
Total deformation was used to evaluate the structural displacement caused by radial loading. The deformation distributions under radial loads of 100 N, 200 N, and 300 N are presented in Figure 7.



a. Displacement-100 N (maks. 0.01943 mm)



b. Displacement-200 N (maks. 0.03885 mm)



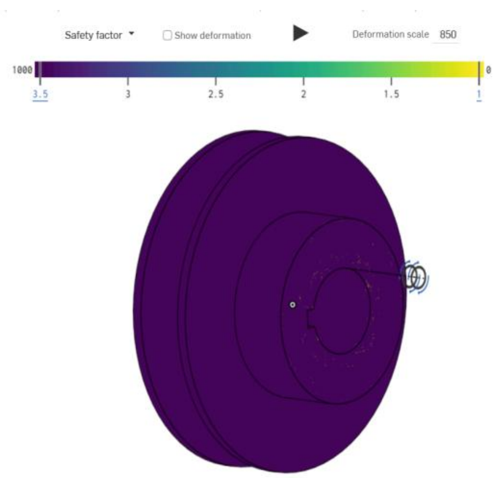
c. Displacement-300 N (maks. 0.05828 mm)

Figure 7. Total Deformation Distribution Under Radial Loads of 100 N (a), 200 N (b), and 300 N (c).

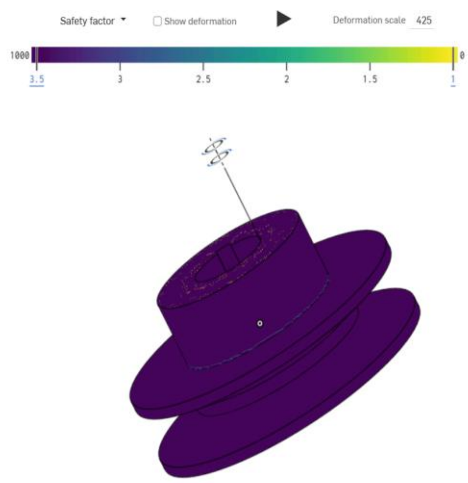
Figure 7. shows that the maximum deformation occurs at the outer edge of the pulley flange under all loading conditions. Under a radial load of 100 N, the maximum deformation is 0.01943 mm. This value increases to 0.03885 mm under a 200 N load and reaches 0.05828 mm under a 300 N load. The increase in deformation with increasing load indicates that the structural response remains within the elastic range. Although the maximum deformation increases by approximately threefold under the highest loading condition, its magnitude remains relatively small compared with the overall dimensions of the pulley. This condition indicates that the pulley structure possesses sufficient stiffness to maintain its geometric stability under operational loading conditions [15].

3.6 Safety Factor Analysis

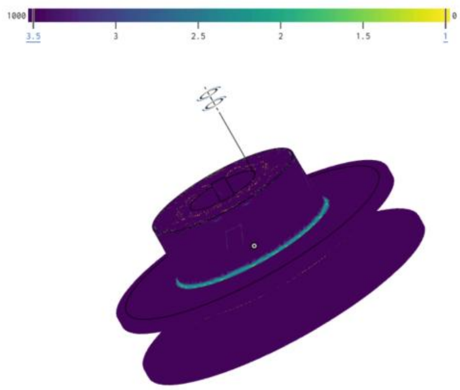
The safety factor was used to evaluate the structural reliability of the pulley against potential material failure. The safety factor distributions under radial loads of 100 N, 200 N, and 300 N are presented in Figure 8.



a. *Safety Factor - 100 N*



b. Safety Factor - 200 N



c. Safety Factor-300 N

Figure 8. Safety Factor Distribution Under Radial Loads of 100 N (a), 200 N (b), and 300 N (c).

The results presented in Figure 8 show that all loading conditions produce a minimum safety factor of 3.5. The region with the lowest safety factor is located around the keyway, which was previously identified as the location of the maximum stress. Under radial loads of 100 N and 200 N, the safety factor distribution indicates a highly safe structural condition. A similar condition is still observed under the 300 N loading condition, although the maximum stress increases to 206.50 MPa. The minimum safety factor remaining above the allowable limit indicates that the pulley structure still possesses an adequate safety margin under the applied radial loading conditions. These results indicate that the sand-cast pulley design remains suitable for use under radial loads of up to 300 N, which is the maximum loading condition investigated in this study.

3.7 Implications of the Results on Pulley Structural Performance

The increase in radial loading from 100 N to 300 N resulted in an almost linear increase in the maximum Von Mises stress and total deformation. This behavior indicates that the pulley structure remained within the elastic region, where an increase in the applied load was accompanied by a proportional increase in the mechanical response. A similar response has also been reported in various FEA-based studies, in which mechanical components subjected to static loading exhibit a linear relationship between load, stress, and deformation as long as the working stress does not exceed the material yield strength [11]. The maximum stress was consistently localized in the keyway region, indicating that the change in cross-sectional geometry in this area is the primary cause of stress concentration [16]. The geometric discontinuity introduced by the keyway results in a non-uniform stress distribution, making this region the critical location under all loading conditions. This phenomenon is consistent with previous studies reporting that the keyway is one of the primary sources of stress concentration in power transmission components due to the abrupt change in cross-sectional geometry [17],[18]. These findings suggest that geometric modifications, such as introducing a fillet radius at the keyway corners, have the potential to reduce stress concentration and improve the structural reliability of the pulley.

The maximum deformation occurred at the outer edge of the pulley flange, with the highest value of 0.05828 mm under the 300 N loading condition. This value is relatively small compared with the overall dimensions of the component and is therefore not expected to affect either the geometric stability or the functional performance of the pulley under radial loading. The ability of the structure to limit deformation indicates that the pulley design possesses sufficient stiffness to maintain its geometric integrity throughout the range of loading conditions analyzed [19]. The minimum safety factor obtained from the simulation indicates that the structure remained within a safe operating condition under all loading variations. The resulting working stress remained below the material yield strength, indicating that the pulley retained an adequate safety margin under radial loads of up to 300 N. Nevertheless, the relatively similar safety factor values obtained for all loading conditions should be interpreted by considering the evaluation methodology implemented in the simulation software. Therefore, the interpretation of the safety factor should not rely solely on its numerical value but should also be supported by the corresponding stress and deformation distributions obtained from the simulation.

This study employed a numerical simulation approach based on the Finite Element Analysis method without validation through experimental testing or analytical calculations. The simulation results exhibit trends that are consistent with previous studies, particularly regarding the location of stress concentration in the keyway region and the linear relationship among applied load, stress, and deformation. However, the simulation results are still influenced by the assumptions that the material is homogeneous, isotropic, and linearly elastic, while the effects of porosity, casting defects, microstructural heterogeneity, and dynamic loading under actual operating conditions have not been considered [20]. Validation through experimental testing, analytical calculations, or simulations under more complex loading conditions is therefore required to improve the reliability of the findings and to extend the applicability of the simulation model to real operating conditions. The use of the actual geometry of the sand-cast pulley represents the primary contribution of this study. Unlike most previous studies that employed idealized geometric models, the simulations in this research were performed using the dimensions of the manufactured component, thereby providing a representation that more closely reflects the actual condition. This approach can serve as a basis for the preliminary evaluation of pulley designs before conducting experimental validation or geometric optimization in regions experiencing stress concentration [21].

The simulation results demonstrate that the use of the actual geometry of the sand-cast pulley is capable of representing the structural response of the component under radial loads of up to 300 N. This approach differs from that adopted in most previous studies, which relied on idealized geometric models as the basis for simulation. The use of the actual geometry enables the distributions of Von Mises stress, total deformation, and safety factor to be evaluated according to the characteristics of the manufactured component, resulting in simulation outcomes that more accurately represent the actual structural behavior [22],[24]. These findings indicate that geometry-based modeling using the actual manufactured component provides a more representative approach for evaluating structural integrity and supporting the design development of sand-cast components.

4. Conclusion

This study successfully evaluated the structural response of a sand-cast aluminum pulley under varying radial loading conditions using the Finite Element Analysis (FEA) method. The simulation results showed that increasing the radial load from 100 N to 300 N resulted in an almost linear increase in both the Von Mises stress and total deformation, indicating that the pulley structure remained within the elastic region under the loading conditions analyzed. Stress concentration was consistently localized in the keyway region of the pulley hub, indicating that this area is the most critical location with respect to potential structural failure and should therefore receive particular attention in future design development. The maximum deformation occurred at the outer edge of the pulley flange and remained relatively small, suggesting that it is unlikely to affect the operational performance of the component. The safety factor evaluation indicated that the structure retained an adequate safety margin under radial loads of up to 300 N. This study demonstrates that a simulation approach based on the actual geometry of the sand-cast pulley can provide a representative evaluation of its structural response. Future research should include validation through experimental testing or analytical calculations and investigate the effects of keyway geometry modification and higher loading conditions on the structural integrity of the pulley.

Authors' Contributions

MM: Conceptualization, Methodology, Writing (Original Draft), Investigation. DB: Data Curation, Formal Analysis, Visualization. HPF: Software, Validation, Data Interpretation. NN: Supervision, Writing (Review and Editing), Project Administration. J: Resources, Funding Acquisition, Supervision. All authors have read and approved the final manuscript.

References

[1] H. Peng et al., "Review of Failure Mechanisms of Steel Wire Ropes Under Heavy-Load Conditions and the Anti-Friction Effects of Gel-like Grease," Gels, vol. 11, no. 11, p. 900, Nov. 2025, doi: 10.3390/gels11110900.

[2] E. W. A. Fanani, E. Surojo, A. R. Prabowo, and H. I. Akbar, "Recent Progress in Hybrid Aluminum Composite: Manufacturing and Application," Metals, vol. 11, no. 12, p. 1919, Nov. 2021, doi: 10.3390/met11121919.

[3] P. K. Mallick, Ed., Materials, Design and Manufacturing for Lightweight Vehicles, Second edition. in Woodhead Publishing series in materials. Oxford: Woodhead Publishing, an imprint of Elsevier, 2021.

[4] C. K. Chan et al., "Development and Implementation of an Automated Cattle Feeding System using Cost-Effective Automation Techniques," Eng. Sci., 2026, doi: 10.30919/es2262.

[5] J. M. Pujara, R. L. Patel, J. K. Dadhaniya, K. K. Dave, H. N. Jani, and D. R. Bhatti, "Investigating Deformation and Stress Distribution in Ball Bearings Using Finite Element Analysis," SSRG-IJME, vol. 11, no. 8, pp. 109–121, Aug. 2024, doi: 10.14445/23488360/IJME-V11I8P113.

[6] W. Jiang, Ed., Analysis and Design of Machine Elements, 1st ed. Wiley, 2019. doi: 10.1002/9781119276081.

[7] A. Akhtar Khan et al., "Structural analysis of cotton stalk Puller and Shredder Machine," Alexandria Engineering Journal, vol. 64, pp. 335–347, Feb. 2023, doi: 10.1016/j.aej.2022.09.002.

[8] K. R. Kashyzadeh and N. G. Bogale, "Simulation-based Optimization of Molten Pool Position in Sand Casting of a Complex Steering Knuckle Geometry," Procedia Structural Integrity, vol. 79, pp. 65–72, 2026, doi: 10.1016/j.prostr.2025.12.308.

[9] D. A. Yeshanew, B. Tolessa, and B. Singh, "Experimental Investigation and Optimization of Reclaimed Sand-Casting Parameters for The Aluminum Alloy Easting Process," Discov Mater, vol. 6, no. 1, p. 81, Feb. 2026, doi: 10.1007/s43939-026-00546-1.

[10] F. Ferroudji, C. Khelifi, F. Meguellati, and K. Koussa, "Design and Static Structural Analysis of a 2.5 kW Combined Darrieus-Savonius Wind Turbine," JERA, vol. 30, pp. 94–99, May 2017, doi: 10.4028/www.scientific.net/JERA.30.94.

[11] S. O. Afolabi, B. I. Oladapo, C. O. Ijagbemi, A. O. M. Adeoye, and J. F. Kayode, "Design and Finite Element Analysis of a Fatigue Life Prediction for Safe and Economical Machine Shaft," Journal of Materials Research and Technology, vol. 8, no. 1, pp. 105–111, Jan. 2019, doi: 10.1016/j.jmrt.2017.10.007.

[12] X. Lai, S. Wang, Z. Guo, C. Zhang, W. Sun, and X. Song, "Designing a Shape–Performance Integrated Digital Twin Based on Multiple Models and Dynamic Data: A Boom Crane Example," Journal of Mechanical Design, vol. 143, no. 7, p. 071703, Jul. 2021, doi: 10.1115/1.4049861.

[13] H. A. Musa and M. Q. Abdullah, "Influence of Using Asymmetric Teeth Profile on Stress Concentrations in Gear Key-way," IOP Conf. Ser.: Mater. Sci. Eng., vol. 1094, no. 1, p. 012067, Feb. 2021, doi: 10.1088/1757-899X/1094/1/012067.

[14] I. T. Haider, M. Lee, R. Page, D. Smith, and W. B. Edwards, "Mechanical Fatigue of Whole Rabbit-tibiae Under Combined Compression-T Loading is Better Explained by Strained Volume than Peak Strain Magnitude," Journal of Biomechanics, vol. 122, p. 110434, Jun. 2021, doi: 10.1016/j.jbiomech.2021.110434.

[15] S. Coemert, B. Yalvac, V. Bott, Y. Sun, and T. C. Lueth, "Development and Validation of an Automated FEM-Based Design Optimization Tool for Continuum Compliant Structures," Int J Mech Mater Des, vol. 17, no. 2, pp. 245–269, Jun. 2021, doi: 10.1007/s10999-020-09506-w.

[16] Cipto, K. A. Rahangmetan, F. Sariman, Hariyanto, S. B. Bahar, and M. T. Sapsal, "Stress Analysis on The Frame of Peanut Skin Peeler Using Autodesk Inventor Software," IOP Conf. Ser.: Earth Environ. Sci., vol. 343, no. 1, p. 012164, Oct. 2019, doi: 10.1088/1755-1315/343/1/012164.

[17] Z. Tian et al., "A New Method for Calculating the Setting Stress of Downhole Packer's Inner Sleeve," Processes, vol. 13, no. 5, p. 1593, May 2025, doi: 10.3390/pr13051593.

[18] W. Reffeor, "Using FEA as a Pedagogical Tool for Teaching Machine Component Design," in 2018 ASEE Annual Conference & Exposition Proceedings, Salt Lake City, Utah: ASEE Conferences, Jun. 2018, p. 31205. doi: 10.18260/1-2--31205.

[19] A. Berardi, C. D. Dellisanti, D. Tarantino, K. S. Leheche Ouette, A. Leone, and A. Tamborrino, "FEA for Optimizing Design and Fabrication of Frame Structure of Elevating Work Platforms," Applied Sciences, vol. 15, no. 13, p. 7356, Jun. 2025, doi: 10.3390/app15137356.

[20] L. I. W. Arnouts, T. J. Massart, N. De Temmerman, and P. Z. Berke, "Computational Design of Bistable Deployable Scissor Structures: Trends and Challenges," Journal of the International Association for Shell and Spatial Structures, vol. 60, no. 1, pp. 19–34, Mar. 2019, doi: 10.20898/j.iass.2019.199.031.

[21] V. Tuninetti, D. Martínez, S. Narayan, B. Menacer, and A. Oñate, "Design Optimization of a Marine Propeller Shaft for Enhanced Fatigue Life: An Integrated Computational Approach," JMSE, vol. 12, no. 12, p. 2227, Dec. 2024, doi: 10.3390/jmse12122227.
[22] D. G. Ntritsos, A. D. Tsolakis, and K. I. Giannakopoulos, "Experimental and Analytical Approach of Fatigue Behavior of Stepped CK45 Shaft with Adjacent Key Groove," Procedia Structural Integrity, vol. 10, pp. 288–294, 2018, doi: 10.1016/j.prostr.2018.09.040.
[23] K. Ntarladima, M. Pieber, and J. Gerstmayr, "A Model for Contact and Friction Between Beams Under Large Deformation and Sheaves," Nonlinear Dyn, vol. 111, no. 22, pp. 20643–20660, Nov. 2023, doi: 10.1007/s11071-023-08973-y.
[24] Y. Vadila, R. Stevano, and R. P. Putra, "Numerical Analysis of Pressure and Velocity Distribution on NACA 0012 Airfoil Using Two CFD Computational Domains," International Journal of Mechanical Engineering and Smart Technology, vol. 1, no. 1, pp. 41–52, 2026.