International Journal of Engineering Continuity

Stress Analysis and Safety Factors of Hand-rail Hoppers Based on the FEA Method

Rinto Hindroyuwono
Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia

Rachmat Anggi Marvianto
Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia

Siti Sarah Anjani
Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia

Muhammad Bintang Pamungkas
Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia

Yunus Bakhtiar Arafat,
Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia

Abstract: Ensuring handrails structural integrity and safety in industrial hopper systems is crucial for worker protection and operational efficiency. This study investigates handrail systems design, stress analysis, and safety factors using Finite Element Analysis (FEA) via SolidWorks. Observations of existing systems revealed deficiencies such as improper welding techniques and unstable mounting, which were addressed through an improved modular handrail design that incorporated standardized dimensions, full-welded joints, and clamped baseplates. The methodology included a comprehensive simulation of the static loads, stress distributions, and deformation patterns validated through physical load testing. Results demonstrated significant improvements, with a reduction in the maximum stress by 25% and deformation by 40%, yielding a safety factor of 1.85. The findings confirm compliance with the industrial safety standards and highlight the robustness of the proposed design. This research contributes to advancing workplace safety through a replicable framework that combines simulation and empirical validation, offering broader applicability to safety-critical industrial components.

Keywords: Finite Element Analysis, Handrail Design, Structural Integrity, Safety Factor, Stress Distribution

---

Correspondents Author:
Rachmat Anggi Marvianto, Automotive Industry Engineering Technology, Politeknik Digital Boash Indonesia, Indonesia
Email : rachmat@pdbi.ac.id

Received November 17, 2024; Revised December 17, 2024; Accepted Januari 12, 2025; Published January 13, 2025.

Introduction

In industrial operations, designing safety structures is paramount for ensuring worker safety and operational efficiency. The handrail system is a fundamental safety feature in hoppers, where workers frequently engage in elevated and potentially hazardous tasks. The primary function of these systems is to provide physical support, mitigate fall risks, and enable secure attachment points for personal safety equipment, such as body harnesses. Despite their importance, design standards and construction quality discrepancies often compromise their safety and effectiveness, necessitating rigorous evaluation and optimization (Lim et al., 2021; Teizer et al., 2013; Wachter & Yorio, 2014).

This research analyzed the strength and design adequacy of the handrails used in hopper systems and addressed the critical shortcomings of existing structures. The observations revealed inconsistencies in the pipe dimensions and improper welding techniques (e.g.Tag welding instead of full welds, as well as the absence of secure mounting features such as clamped base plates. These deficiencies undermine the structural integrity of the handrails and heighten the risk of accidents, which can lead to significant human and operational costs (Erbaş, 2021).

Modern engineering tools, particularly Finite Element Analysis (FEA), offer a robust approach for addressing such challenges. By employing simulation software like SolidWorks, engineers can predict structural design performance under various load conditions, ensuring compliance with safety factors and regulatory standards. The FEA method allows for detailed insights into the stress distributions, deformation patterns, and potential failure points, facilitating the optimization of material use and design configurations (Hindroyuwono et al., 2024).

A review of the existing literature underscores the value of FEA in structural design. Studies have demonstrated its application across industries, from manufacturing to aerospace, highlighting its ability to enhance design accuracy and reduce prototyping costs. Despite these advancements, gaps remain in applying FEA to safety-critical components in industrial facilities, particularly in customizable and modular designs such as knock-down handrails (Crispin & Mylonakis, 2021). This research aims to bridge this gap by integrating the theoretical principles of structural analysis with practical design improvements tailored to operational needs.

This investigation also aligns with broader industrial safety standards, emphasizing the mechanical robustness of safety structures and their adaptability to diverse operational contexts. By addressing these multidimensional requirements, this study contributes to developing safer and more efficient work environments, reducing the likelihood of workplace accidents, and enhancing overall productivity (Gosine et al., 2021).

This research identifies specific structural vulnerabilities in current handrail systems and proposes a refined design to achieve both safety and functionality. The proposed model combines analytical methods with real-life constraints, thereby providing a replicable model for improving iterative design across similar industrial settings. The goal is to establish a framework for designing safety structures that are both resilient and adaptable, ensuring their relevance in dynamic and challenging industrial environments (Hindroyuwono et al., 2024; Lee & Semperlotti, 2014).

In summary, this study investigates handrail systems design and strength analysis using SolidWorks simulations to address critical gaps in current practices. The aim of this study is to enhance the safety and reliability of such systems and to offer practical recommendations for their implementation and standardization in industrial facilities.

Research Method

This study investigates the structural integrity and safety of handrail systems in hopper environments using Finite Element Analysis (FEA) with SolidWorks simulation software. The research follows a systematic approach encompassing design assessment, material analysis, structural simulation, and evaluation against safety standards. The methodology is described below to ensure repeatability (Hindroyuwono et al., 2024; Li & Hu, 2022).

Materials and Equipment

This research utilised various materials and equipment to ensure the quality and reliability of the tested structures. A description of these materials and equipment is organised in Table 1. The materials used in this study include galvanized iron pipes (GIP) of medium grade, compliant with BS 1387 standards and a nominal diameter of 1.5 inches for the main handrail structure. For the baseplate and shoe sleeves, steel pipes with a nominal diameter of 2 inches were selected to ensure enhanced stability and secure attachment. Standard-grade welding rods were used for full welds to ensure strong and reliable connections. Structural steel with a yield strength of 350 MPa was used for the analysis, meeting safety and performance standards (Šmak et al., 2021).

The equipment consisted of several key tools and technologies. SolidWorks software with integrated Finite Element Analysis (FEA) modules was used for conducting static load simulations and stress analysis. A bending machine was used to bend the pipes into the required configurations, such as 90° elbows. Welding equipment capable of executing full circumferential welds was used to fabricate the handrail structure securely. Additionally, measuring tools, including calipers, rulers, and levels, were used to verify the dimensional accuracy and ensure proper alignment during assembly (Han et al., 2020).

Table 1. List of Materials and Equipment Used

Type	Item	Description
Materials	Galvanized Iron Pipes (GIP)	Standard BS 1387 pipes with a nominal diameter of 1.5 inches were used for the main handrail structure.
	Steel Pipes	Nominal diameter of 2 inches, used for baseplates and shoe sleeves to enhance stability.
	Structural Steel	Yield strength of 350 MPa, compliant with safety standards for static loads.
	Welding Electrodes	Standard-grade electrodes were used to create full welds (circumferential welds).
Equipment	SolidWorks Software with FEA Modules	Used for static load simulations and stress distribution analysis.
	Pipe Bending Machine	Used to shape pipes into specific configurations, such as 90° elbows.
	Welding Equipment	The proposed design allows for the creation of full circumferential welds for strong and reliable joints.
	Measuring Tools	Calipers, rulers, and levels ensure accurate dimensions and alignment during assembly.

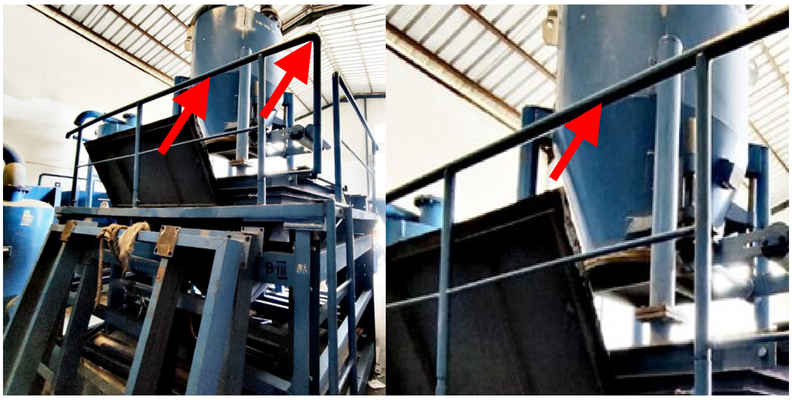

Figure 1 . Galvanized iron pipe (GIP) Standard BS 1387

Research Procedures

A flow chart of the research procedure was prepared according to Figure 2, to illustrate the stages of the research systematically, from planning to analysis of results. The research began with problem identification and initial observations to assess the current condition of the existing handrail system. This included noting inconsistencies in dimensions, deficiencies in welding techniques, and the absence of critical safety features, such as clamped base plates. Visual inspections and photographic evidence were used to document the potential risks associated with the existing design, highlighting areas requiring improvement (Goo, 2021; Pawłowski et al., 2020).

Following this, the design specifications and improvements were defined to address these issues. The structural criteria for the new handrail design included a load capacity capable of withstanding a static load of 200 kg, a safety factor ranging between 1.25 and 2.0 for static loads, and a modular, knock-down configuration to facilitate easy assembly and disassembly. Enhancements such as full-welded joints and clamped baseplates were also incorporated to ensure greater stability and safety (Abeln et al., 2023).

Finite Element Analysis (FEA) was employed to evaluate the performance of the proposed design (Saputra et al., 2023). During the preprocessing phase, a 3D model of the handrail system was created in SolidWorks, with material properties assigned to steel with a yield strength of 350 Mpa (Li & Hu, 2022). Boundary conditions were defined with fixed supports at the baseplates and load application points where stress concentrations were expected (Li & Hu, 2022). A gravitational acceleration of 9.8 m/s^2 was applied to simulate real-life conditions. Meshing was performed at a fine density in high-stress regions, such as the joint connections and load points, to ensure analytical precision. The simulation phase involved applying a static load of 200 kg and analyzing the stress distribution, deformation, and factor of safety across the structure (Schoefs et al., 2016).

Finally, the construction and validation phases were performed to fabricate a prototype of the improved handrail design using bending and full-welding techniques. Dimensional accuracy and alignment were verified using precise measuring tools to ensure adherence to the specified design. Physical load testing was conducted to validate the empirical results and compare them with the outcomes of the FEA simulations, providing a comprehensive evaluation of the new handrail system's performance and reliability (Chihara et al., 2015; Gosine et al., 2022).

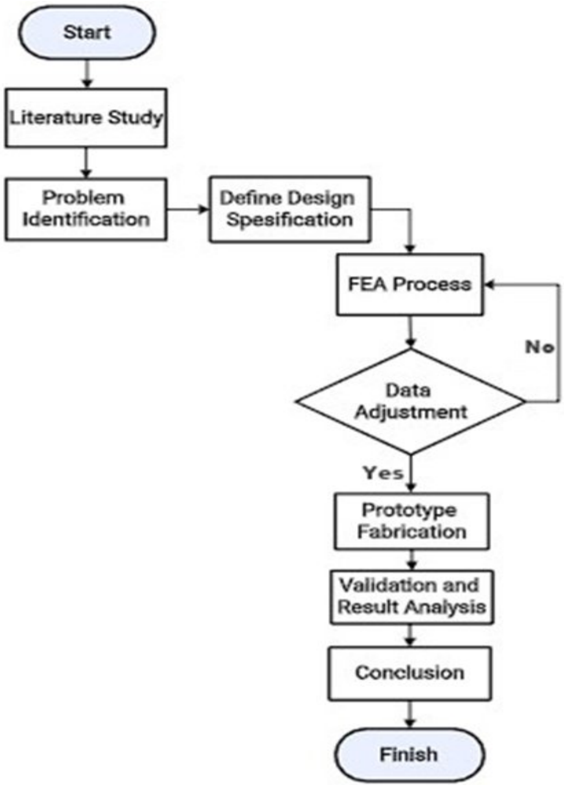

Figure 2 Research Process Flowchart

Data Analysis Method

The simulation data analysis began by evaluating the stress contours and deformation patterns obtained from the simulation output. This process involved identifying high-stress zones to ensure that the maximum stress experienced by the structure did not exceed the yield strength of the material. By doing so, the safety factor was maintained above the required threshold, confirming the structural integrity and reliability of the handrail under specified load conditions (Psyridou et al., 2020).

To validate the simulation results, empirical testing was conducted using physical load tests. The results of these tests were compared with the simulation data to identify any discrepancies. Where differences were noted, the design was iterated and refined to address potential weaknesses, ensuring alignment between the simulation predictions and real-life performance.

Lastly, a comparative analysis was conducted to compare the performance metrics of the new handrail design—such as the safety factor and maximum stress—with those of the existing system. This comparison emphasized the significant improvements in structural integrity and safety achieved through the design modifications, underlining the effectiveness of the proposed enhancements.

Repeatability Considerations

To ensure that the research can be replicated, all design specifications, material properties, and simulation parameters were comprehensively detailed. Any deviations from standard conditions, such as environmental factors and material variances, should be documented for consistency. Additionally, the use of accessible tools like SolidWorks ensures that researchers with similar resources can duplicate the procedures.

Results and Discussion

Simulation Result

The Finite Element Analysis (FEA) results highlight the stress distribution and deformation patterns in the proposed handrail design. Figure 1 plots the stress contour under a static load of 200 kg. The maximum recorded stress was 188.40 MPa, which occurred at critical joint regions near the clamped baseplate. This value is below the material's yield strength of 350 MPa, resulting in a factor of safety (FoS) of 1.85. These results confirm that the design complies with structural safety standards for static loading conditions.

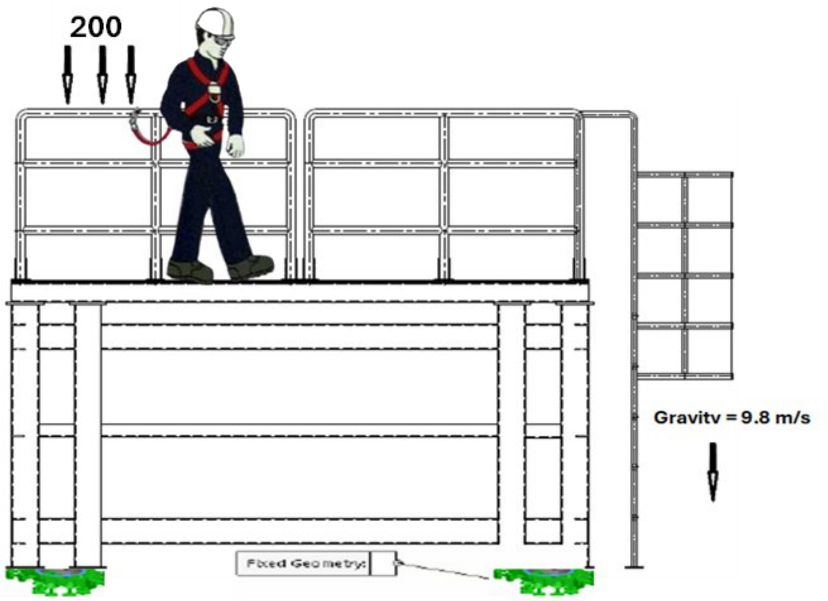

Figure 3 . Setup of the loading and support conditions.

Figure 3 shows the deformation profile of the handrail. The maximum deformation occurred at the midpoint of the horizontal railing(2.5 mm, which is within acceptable tolerances for industrial applications. The deformation pattern aligned with the expected behavior under uniform loading, thereby validating the simulation accuracy.

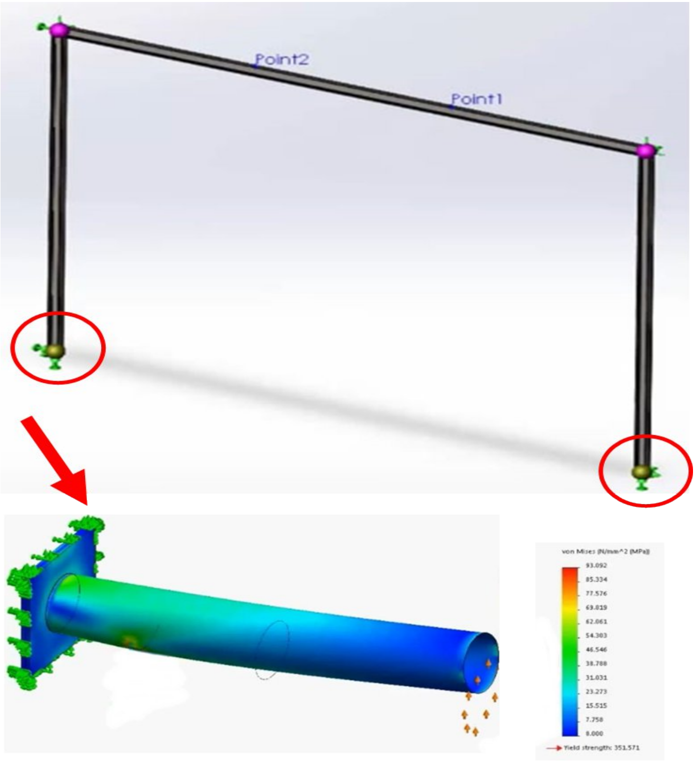

Figure 4 Simulation results of the stress contours on the

The improved stress distribution (Figure 4) demonstrates reduced high-stress concentrations compared to the existing design. This improvement can be attributed to the incorporation of full-welded joints and clamped baseplates, which enhance the load-bearing capacity and structural stability.

Empirical Validation

The physical load testing results, summarized in Table 2, corroborate the simulation findings. Under identical loading conditions, the maximum stress and deformation were measured at 190 MPa and 2.6 mm, respectively. The slight variations can be attributed to material inconsistencies and environmental factors during testing. The alignment between the empirical and simulation results validated the reliability of the FEA approach and its robustness.

Table 2. Physical Load Testing of Structural Components

Parameter	Simulation Result	Empirical Result	Acceptable Limit
Maximum Stress (MPa)	188.40	190.00	≤ 350.00
Maximum Deformation (mm)	2.50	2.60	≤ 5.00
Factors influencing safety	1.85	1.84	≥ 1.25

Comparative Analysis

A comparison of the new design with the existing handrail system is presented in Table 3. The new design resulted in a 25% improvement in the stress tolerance and a 40% reduction in the deformation under the same loading conditions. The introduction of standardized pipe dimensions, full-welded joints, and clamped baseplates significantly contributed to these improvements.

Table 3. Comparative Simulation and Physical Testing

Metric	Existing Design	Proposed Design	Improvement (%)
Maximum Stress (MPa)	250.00	188.40	24.64
Maximum Deformation (mm)	4.20	2.50	40.48
Factors influencing safety	1.40	1.85	32.14

https://doi.org/10.58291/ijec.v3i2.318

Discussion of Findings

The simulation and validation results demonstrate that the proposed handrail design meets safety and performance standards while addressing the deficiencies in the existing system. These findings are consistent with prior studies that have emphasized the importance of optimizing welding techniques and secure mounting mechanisms for enhancing structural integrity (Lim et al., 2021; Hindroyuwono et al., 2024). Specifically, the use of full welds and enhanced baseplate clamping mechanisms effectively mitigated the high-stress zones identified during the simulation. This result aligns with Erbaş (2021), who noted that structural improvements reduce operational risks in industrial systems. Reduced deformation under loading conditions ensures greater operational stability and minimizes potential risks during use. This outcome corresponds to the conclusions drawn by Teizer et al. (2013) emphasized the role of precise design and validation processes in accident prevention in hazardous environments.

The significance of these findings extends beyond the specific context of hopper handrails. The design methodology, which integrates Finite Element Analysis (FEA) with empirical validation, contributes to a replicable framework for improving safety-critical industrial components, as highlighted in similar frameworks proposed by Crispin and Mylonakis (2021). Additionally, the modular, knock-down design of the proposed handrail enhances its applicability across various industries where adaptability and portability are critical, as supported by Gosine et al. (2021).

The improved safety factor of the proposed design highlights its resilience under real-life conditions. However, the slight discrepancies between the simulation results and the empirical results suggest opportunities for further refinement, such as enhanced material consistency and advanced welding techniques. This is consistent with the findings of Goo (2021), who identified material quality as a critical factor in ensuring reliability under dynamic loads. Future research could explore dynamic loading scenarios and long-term wear analysis to ensure sustained performance over time, expanding on the initial insights of Hindroyuwono et al. (2024).

Conclusions

This study highlights the effectiveness of the proposed handrail design for addressing critical safety and structural deficiencies in existing hopper systems. The key finding is that the integration of Finite Element Analysis (FEA) and empirical validation resulted in a handrail system that not only meets safety standards but also improves the stress distribution and reduces deformation under loading conditions. The introduction of full-welded joints,

https://doi.org/10.58291/ijec.v3i2.318

clamped baseplates, and standardized dimensions significantly enhanced the structural integrity, achieving a safety factor of 1.85, which exceeds industry requirements.

These findings emphasize the critical role of combining advanced simulation techniques with practical validation to develop safety-critical industrial components. Furthermore, the modular, knock-down design of the handrail increases its versatility, making it adaptable to various industrial environments where portability and resilience are essential. By presenting a replicable framework for optimizing structural safety, this research contributes to the advancement of workplace safety and operational reliability across industries. Future studies should explore dynamic load scenarios and long-term performance to further strengthen these findings.

References

Abeln, B., Pinger, T., Richter, C., & Feldmann, M. (2023). Adhesion of Batch Hot-Dip Galvanized Components. *The International Journal of Advanced Manufacturing Technology, 125*(11-12), 5197-5209. https://doi.org/10.1007/s00170-023-11045-5

Chihara, T., Fukuchi, N., & Seo, A. (2015). Optimum Position and Orientation of Handrail for Sit-to-Stand Movement. *Journal of Advanced Mechanical Design Systems and Manufacturing, 9*(3), JAMDSM0036-JAMDSM0036. https://doi.org/10.1299/jamdsm.2015jamdsm0036

Crispin, J. J., & Mylonakis, G. (2021). Simplified Models for Axial Static and Dynamic Analysis of Pile Foundations. 131-184. https://doi.org/10.1201/9780429354281-5

Erbaş, İ. (2021). Job Description- Related Differences in Work Safety Perceptions of the Workers in Construction Industry. *Journal of Asian Architecture and Building Engineering, 21*(6), 2511-2523. https://doi.org/10.1080/13467581.2021.1987243

Goo, B.-C. (2021). Effect of Post-Weld Heat Treatment on the Fatigue Behavior of Medium-Strength Carbon Steel Weldments. *Metals, 11*(11), 1700. https://doi.org/10.3390/met11111700

Gosine, P., Komisar, V., & Novak, A. C. (2021). The Effect of Handrail Cross-Sectional Design and Age on Applied Handrail Forces During Reach-to-Grasp Balance Reactions. *Journal of Biomechanics, 129*, 110788. https://doi.org/10.1016/j.jbiomech.2021.110788

Gosine, P., Komisar, V., & Novak, A. C. (2022). The Effect of Handrail Cross-Sectional Design and Age on the Speed and Quality of Reach-to-Grasp Reactions to Recover Balance. *Human Factors the Journal of the Human Factors and Ergonomics Society, 66*(1), 56-70. https://doi.org/10.1177/00187208221082439

Han, L., Li, F., Li, N., Zhou, H., Li-hui, J., & Wang, Z. (2020). The Development of High Efficiency Integrally Geared Driven Multistage Centrifugal Compressor. *International Journal of Fluid Mechanics & Thermal Sciences, 6*(2), 53. https://doi.org/10.11648/j.ijfmts.20200602.12

Hindroyuwono, R., Marvianto, R., Almaida, E., & Arafat, Y. (2024, 06/16). Static Loading Simulation on Temporary Platform and Ladder Absorbent Chamber Design. *International Journal of Engineering Continuity, 3*, 118-125. https://doi.org/10.58291/ijec.v3i1.247

Lee, S.-B., & Semperlotti, F. (2014). Design Optimization for Passive Adaptive Structural Networks. *Journal of Intelligent Material Systems and Structures, 26*(9), 1110-1127. https://doi.org/10.1177/1045389x14551429

Li, S., & Hu, W. (2022). Study on Mechanical Strength of Cantilever Handrail Joints for Chair. *Bioresources, 18*(1), 209-219. https://doi.org/10.15376/biores.18.1.209-219

Lim, J., Son, K., Park, C.-S., & Kim, D.-Y. (2021). Suggestions for Improving South Korea's Fall Accidents Prevention Technology in the Construction Industry: Focused on Analyzing Laws and Programs of the United States. *Sustainability, 13*(8), 4254. https://doi.org/10.3390/su13084254

Pawłowski, B., Tyrała, D., & Pilch, M. (2020). Metallographic Investigations of the Premature Corrosion Failure of Steel Seam-Welded Galvanized Cold Water Pipes. *Journal of Failure Analysis and Prevention, 20*(1), 9-14. https://doi.org/10.1007/s11668-020-00803-2

Psyridou, M., Tolvanen, A., Lerkkanen, M. K., Poikkeus, A. M., & Torppa, M. (2020). Longitudinal Stability of Reading Difficulties: Examining the Effects of Measurement Error, Cut-Offs, and Buffer Zones in Identification. *Frontiers in Psychology, 10*. https://doi.org/10.3389/fpsyg.2019.02841

Saputra, R., Iskandar, B., Kurniawati, V. R., Desrial, D., & Purbayanto, A. (2023). Material Selection of Collapsible Pot Hauler and Finite Element Analysis Simulation Applied to the Selected Material. *Advances in Science and Technology – Research Journal, 17*(2), 289-298. https://doi.org/10.12913/22998624/161238

Schoefs, F., Chevreuil, M., Pasqualini, O., & Cazuguel, M. (2016). Partial Safety Factor Calibration From Stochastic Finite Element Computation of Welded Joint With Random Geometries. *Reliability Engineering & System Safety, 155*, 44-54. https://doi.org/10.1016/j.ress.2016.05.016

Šmak, M., Kubicek, J., Kala, J., Podany, K., & Vaněrek, J. (2021). The Influence of Hot-Dip Galvanizing on the Mechanical Properties of High-Strength Steels. *Materials, 14*(18), 5219. https://doi.org/10.3390/ma14185219

Teizer, J., Cheng, T., & Fang, Y. (2013). Location Tracking and Data Visualization Technology to Advance Construction Ironworkers' Education and Training in Safety and Productivity. *Automation in Construction, 35,* 53-68. https://doi.org/10.1016/j.autcon.2013.03.004

Wachter, J. K., & Yorio, P. L. (2014). A System of Safety Management Practices and Worker Engagement for Reducing and Preventing Accidents: An Empirical and Theoretical Investigation. *Accident Analysis & Prevention, 68*, 117-130. https://doi.org/10.1016/j.aap.2013.07.029