International Journal of Industrial Engineering & Production Research December 2025, Vol. 36, No. 4: 1-12
DOI: 10.22068/ijiepr.36.4.2340

RESEARCH PAPER

Ergonomics Assessment of Hose-Rolling Posture and Simulation-Based Validation of an Assistive Tools for Hose Roller

Salwa Mahmood[1,*], Ahmad Zahin Zainal Rasid[2], Nurul Ainina Nadhirah Tajurahim[3], Helmy Mustafa El Bakri[4], Ismail Abdul Rahman[5], Noorul Azreen Aziz[6]

Received 12 March 2025; Revised 01 July 2025; Accepted 16 July 2025;
© Iran University of Science and Technology 2025

ABSTRACT
This study addresses ergonomic risks faced by firefighters during hose rolling activities, a physically demanding task that can lead to musculoskeletal issues. Building on a previous project conducted at the Bukit Gambir Fire and Rescue Station, this research expands the analysis by comparing four different hose rolling postures: roll and coil, dutch roll, flaking, and figure of eight. The Rapid Entire Body Assessment (REBA) method was used to evaluate the ergonomic risk levels of each posture. To further enhance firefighter safety, a preventive strategy tool was proposed and developed. The tool's design was tested using Finite Element Analysis (FEA) in SolidWorks to assess the structural performance of its alloy steel hook and shaft under load. Simulation results showed stress levels below the material's yield strength, and factor of safety (FOS) analysis confirmed the tool's structural reliability. This project takes a holistic approach to understanding and mitigating ergonomic risks in firefighting. This study found that the structural of an assistive tool is safe and confirming the robustness and reliability of both the hook and alloy steel shaft designs. By combining ergonomic assessment with engineering simulation, it not only identifies high-risk postures but also provides a practical solution to reduce strain and prevent injury. Ultimately, the project contributes to improving the safety and well-being of firefighters, supporting a safer work environment for those who risk their lives to protect others.

KEYWORDS: *Ergonomic risk assessment (ERA); Rapid entire body assessment (REBA); Finite element analysis (FEA).*

1. Introduction

Ergonomics, defined as the scientific discipline concerned with understanding human interactions within systems, plays a crucial role in enhancing both human well-being and overall system performance [1][2]. As highlighted by [3], the core objective of ergonomics is to design workspaces, tools, and interfaces that are aligned with human capabilities and limitations, thereby promoting comfort, safety, and efficiency. One of the most common and impactful ergonomic issues in the workplace is the occurrence of musculoskeletal disorders (MSDs), which arise from repetitive motions, excessive physical strain, or sustained awkward postures. These disorders can severely affect worker productivity, health, and quality of life [4]. MSDs are typically characterized by damage or discomfort in muscles, joints, tendons, and related structures, often worsened by unsuitable workplace conditions and poorly designed tasks [5].

To systematically address these issues, Ergonomic Risk Assessment (ERA) is employed as a structured method to identify, assess, and control ergonomic hazards within work environments [3, 6]. Advanced ERA tools, such as the Rapid Entire Body Assessment (REBA), enable a detailed evaluation of whole-body postural risks. These tools focus on detecting and reducing potential ergonomic stressors that could lead to injury or discomfort, making them essential in efforts to improve occupational health and safety [7].

This study explores the ergonomic risks firefighters face during hose rolling, a physically demanding and repetitive task. Using long, heavy hoses can strain the body, and poor posture may

* *Corresponding author: Salwa Mahmood*
msalwa@uthm.edu.my

1. *Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh Campus, 84600 Panchor, Johor, Malaysia*
2. *Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh Campus, 84600 Panchor, Johor, Malaysia*
3. *Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh Campus, 84600 Panchor, Johor, Malaysia*
4. *Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh Campus, 84600 Panchor, Johor, Malaysia*
5. *National Institute of Occupational Safety and Health, Kompleks Mutiara Johor Land, Jalan Bukit Mutiara, Bandar Dato Onn, 81100, Johor Bahru, Johor, Malaysia*
6. *National Institute of Occupational Safety and Health, Kompleks Mutiara Johor Land, Jalan Bukit Mutiara, Bandar Dato Onn, 81100, Johor Bahru, Johor, Malaysia*

lead to musculoskeletal disorders (MSDs). The research focuses on four common hose-rolling techniques, namely roll and coil, Dutch roll, flaking, and figure of eight.

Previous research conducted an initial Ergonomic Risk Assessment (ERA) of hose rolling activities among firefighters [8], revealing the need for a more in-depth analysis through advanced ERA methods. It lacked ergonomic and structural strength analysis of the developed tool, limiting its evaluation, particularly in terms of safety factors. Responding to this gap, the present study evaluates the ergonomic impact of four commonly used hose rolling postures using the Rapid Entire Body Assessment (REBA) method. Additionally, it incorporates the simulation of an assistive tool design proposed by Mustapha (2023) [9], aimed at improving firefighters' posture during these activities. To ensure the tool's practicality and structural integrity, Finite Element Analysis (FEA) was carried out using SolidWorks. However, the design would be incomplete without simulation to evaluate whether critical components such as the hook and alloy steel shaft are structurally reliable and capable of withstanding the applied loads. FEA enables a thorough assessment of the tool's mechanical performance, including stress distribution, deformation, and identification of potential failure points [10, 11]. These analyses provide critical insights into the tool's effectiveness in minimizing ergonomic risks and enhancing firefighter safety.

The main objective of this study is to evaluate four hose rolling postures using the REBA method and to simulate the assistive tool design proposed by Mustapha (2023) [9] to improve firefighter posture during hose rolling activities. The simulation aims to confirm the reliability of key components, such as the hook and shaft. By addressing these ergonomic risks, the study contributes to enhancing firefighter safety and well-being, offering practical insights to reduce musculoskeletal disorders (MSDs) and promote healthier work practices in firefighting. The findings of this study are expected to have broader applicability beyond the local context, particularly in the firefighting profession and other physically demanding occupations.

2. Methodology

This study employed a dual-method quantitative approach incorporating both REBA and FEA techniques. Before conducting the risk assessment, approval was sought and successfully obtained. Subsequently, firefighters were identified and observed to understand their working conditions thoroughly. Detailed observations of their postures were recorded through photos and video footage to support the ergonomic risk assessment. To precisely measure posture angles and load distribution, goniometers in Kinovea and weight scales were employed. The REBA method was applied to assess ergonomic risk levels among the participants. In parallel, Finite Element Analysis (FEA) simulations were conducted using SolidWorks software to evaluate the structural integrity and performance of the assistive rolling tool proposed by Mustapha (2023) [9]. The flowchart of this study is illustrated in Figure 1.

2.1. Ethical approval for conducting this study

This endorsement underscored the commitment to obtain ethical approval for this research titled "Firefighter Hose Roller: Ergonomics Assessment and Fabrication". This research received approval from the Research Ethics Panel Committee, Universiti Tun Hussein Onn Malaysia, during the meeting no. 3/2022 on November 9, 2022.

2.2. Design of study

The research began by identifying ergonomic risks faced by firefighters during hose rolling activities, particularly due to awkward postures that could lead to MSDs. To address this, a posture-assistive tool was proposed to improve body mechanics during the task. The study not only assessed the ergonomic impact of various hose rolling techniques using the REBA method but also simulated the proposed tool's structural performance using FEA in SolidWorks. Through this process, the design was refined and validated based on both ergonomic outcomes and engineering reliability. This aligns with the core of a design study, which focuses on creating and evaluating a solution in response to a practical challenge, aiming to improve safety, usability, and effectiveness in real operational contexts.

2.3. Participants and setting

Before beginning the assessment, the research team prepared cameras to capture appropriate videos and images. The images taken had to be clear and precise to allow accurate measurement

of body posture angles, as required by the REBA technique. Four firefighters participated in the process, each demonstrating four different postures commonly used during hose rolling activities. These visual recordings served as essential references for posture analysis and ergonomic risk evaluation. All participating firefighters were male and took part in the study on a voluntary basis. They are actively engaged in hose rolling tasks as part of their operational responsibilities and each has over five years of professional experience in the firefighting field. This study was conducted at Bukit Gambir Fire and Rescue Station, Johor, Malaysia.

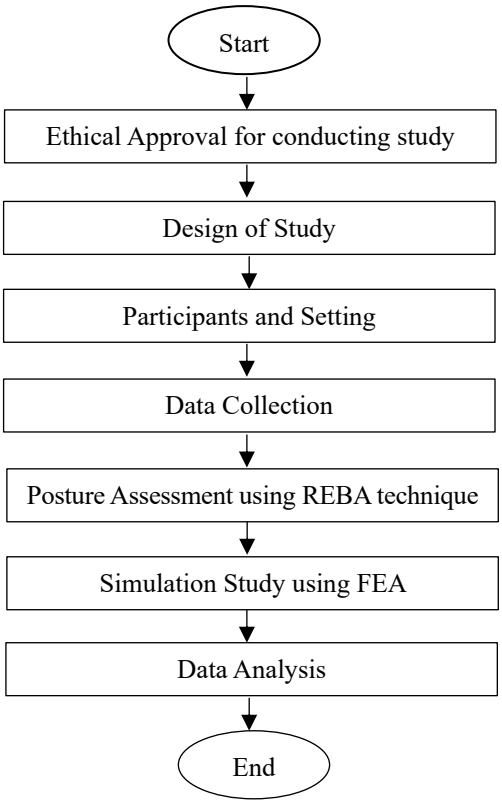


Fig.1. Flowchart of the study

2.4. Data collection procedures

This study employed observational techniques, beginning with a thorough understanding of the hose rolling methods used by firefighters. Each firefighter involved was required to perform a complete demonstration of the hose rolling activity from start to finish. During the demonstration, photos and video recordings were taken to capture body postures at various stages of the task for ergonomic assessment using the REBA method. To ensure accuracy in posture analysis, Kinovea software was applied to measure precise body angles [12, 13].

The physical load involved in the task was assessed by weighing the hose and associated equipment using digital scales. This step is essential in evaluating ergonomic risk factors, as load handling plays a significant role in the development of MSDs.

2.5. Posture assessment using rapid entire body assessment (REBA)

REBA is an established method developed to evaluate working postures that may contribute to work-related musculoskeletal disorders (WMSDs). REBA is acknowledged as an assessment framework to assess entire body posture for WMSDs risks, specifically designed to handle unpredictable working postures common in the service industry [14]. It is particularly effective in assessing both static and dynamic tasks across various industries [15-18]. In this study, the REBA method was applied to analyze the postures adopted by firefighters during hose rolling activities across four commonly practiced techniques. Before the assessment, participants were asked to select the posture they considered most representative or challenging, based on factors such as physical difficulty, duration of posture maintenance, and the load handled. The REBA assessment involves a standardized worksheet, which calculates scores by evaluating the positioning of the neck, trunk, legs, upper arms, lower arms, and wrists. The worksheet is structured into two parts Part A and Part B, allowing for a comprehensive assessment that reflects the cumulative ergonomic risk associated with each posture.

Snapshots were taken at critical points during the activity, including lifting, coiling, and placing the hose, to capture a full range of postures for evaluation. Each posture was scored based on REBA criteria, which evaluate joint angles, force and load, coupling, and repetition activity. The final score determined the level of MSDs risk, guiding the need for intervention [19].

2.6. Simulation study using finite element analysis (FEA)

Finite Element Analysis (FEA) is a highly effective engineering tool that employs advanced calculations, detailed modeling, and simulations to accurately predict and analyze how a component or structure will behave under various physical conditions, such as stress, heat, and vibration

[10, 19]. It enables researcher to systematically evaluate and detect potential weaknesses or failure points in their design prototypes, thereby offering crucial insights into the structural integrity, durability, and performance of engineered products [20].

In the context of this study, FEA was utilized to assess the structural performance of an assistive tool designed to support firefighters during hose rolling activities. This tool, proposed by Mustapha (2023) [9], aims to reduce ergonomic strain by improving posture and load management. The FEA simulations were conducted to evaluate key components of the tool specifically the hook and alloy steel shaft under expected operational loads. By applying the Finite Element Method (FEM), which divides the structure into smaller elements connected at nodes, the tool's stress distribution and deformation patterns were analyzed [10][21]. This localized analysis provided essential data to verify the tool's reliability, ensuring it can withstand the physical demands of hose rolling tasks without compromising firefighter safety. Thus, FEA played a pivotal role in confirming the ergonomic and structural feasibility of the proposed design within real-world firefighting scenarios.

2.7. Data analysis

Descriptive statistics were employed to analyze the REBA scores obtained from each of the four hose rolling techniques stated. The REBA scores were summarized using comparative analysis was then performed to identify which posture posed the highest ergonomic risk, based on the REBA score interpretations. This comparison provides insight into which technique may require intervention or ergonomic redesign. Although inferential statistical tests were not applied due to the small sample size, the descriptive approach offers valuable preliminary data for identifying high-risk postures.

3. Results and Discussion

3.1. REBA for flaking

In the REBA assessment, limbs and body postures are most frequently used, and they focused close attention on extreme joint angles by using Kinovea software. Initial observations were conducted for Part A, which includes the neck, trunk, and legs. Subsequently, Part B, comprising the upper arm, lower arm, and wrist postures, was analyzed to obtain the respective scores. The flaking posture is illustrated in Figure 2.

Fig.2. Flaking posture

The evaluation begins with Part A, which focuses on determining the neck position using the REBA method. A score of 3 is assigned based on the neck angle measurement of 111.4°, as shown in Figure 3 which noting the neck twist. Besides, the evaluation shifts to the trunk position, with a score of 4 given for a 24.9° inclination from the front, which noting trunk twist. The leg position assessed which warranting a score of 4 due to a 114.8°. Hence, the posture score of 9 is obtained by combining these individual scores. Meanwhile, the posture score is 9 referred to Table A in REBA worksheet, the force or load score is determined based on a load exceeding 22 pounds, resulting in a score of 3 and adding these yields a total score A is 12.

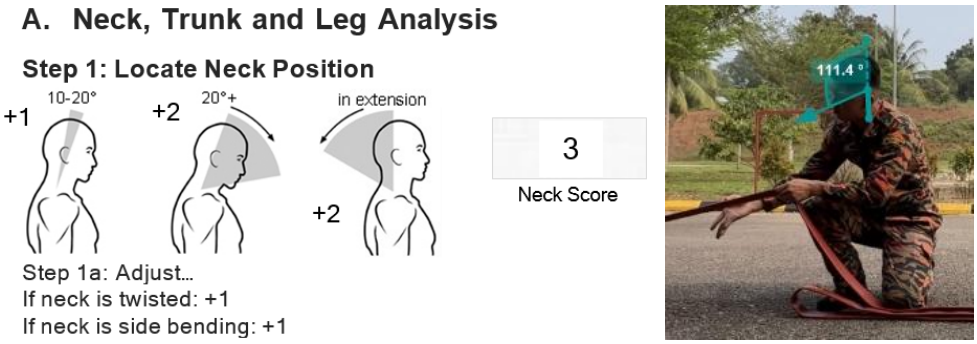


Fig.3. Locate neck position

The evaluation of flaking posture for Part B using REBA, which assesses the upper arm position. A score of 5 is assigned based on the observation of 153.2°, with the upper arm positioned 90° above and abducted during the task of flaking a fire hose. Besides, the evaluation of lower arm position assigns a score of 2 for a 179.1° angle observed during the rolling of the fire hose. The wrist position is assessed, resulting in a score of 2 due to a 14.9° and wrist bending with twisting. A posture score of 8 is determined. Besides, the coupling score is evaluated, assigning a score of 2 for an acceptable but suboptimal handhold or coupling. These results are combined to derive a score B of 10. To summarize, the combined score from Sections A and B yields a preliminary posture score, which is then referenced against Table C in REBA worksheets to determine the final REBA score. Additionally, an activity score of +1 is applied to reflect rapid or repetitive posture changes or body instability. This adjustment results in a final REBA score of 13. This score, documented in the REBA worksheet, is crucial for evaluating ergonomic risk. Specifically, the flaking posture scored 13, indicating a very high-risk level that necessitates immediate ergonomic intervention. The same evaluation process was consistently applied to the other hose rolling techniques to ensure a comprehensive comparison across all postures.

3.2. Comparison of all postures

The posture assessments were conducted using the REBA method, obtaining scores of 11, 12, 13, and 12 for all the hose rolling techniques as detailed in Table 1. Analysis of these scores clearly indicated that the flaking posture had the highest ergonomic risk, suggesting it causes greater discomfort and potential musculoskeletal strain for firefighters during hose rolling activities. By integrating the REBA method with FEA simulations in SolidWorks, designers can perform a comprehensive evaluation of the ergonomic impact of various tools used in hose rolling. While REBA focuses on assessing postures, forces, and repetitive movements, FEA provides detailed insights into the structural behavior of the tools under load. This combined approach enables a thorough understanding of how tool designs influence user comfort and safety by simulating forces, stress distributions, and strain responses. Ultimately, the synergy between REBA and FEA establishes a systematic framework to optimize tool designs, ensuring they not only fulfill functional requirements but also prioritize the ergonomic health and well-being of firefighters during demanding hose rolling tasks.

3.3. Simulation results

In this study, Finite Element Analysis (FEA) is utilized to evaluate the structural response of two critical components, the hook and the shaft of the hose rolling tool as illustrated in Figures 4 and Figure 5, respectively. The primary aim is to assess how these parts withstand and perform under prolonged static loading conditions. By applying FEA to these specific elements, engineers gain detailed insights into their stress distribution, deformation patterns, and potential failure points.

Tab.1. REBA score of posture assessment for all techniques

REBA	Roll and coil	Dutch roll	Flaking	Figure of eight
Step 1: Locate neck position	2	2	3	1
Step 2: Locate trunk position	3	4	4	3
Step 3: Legs	4	3	4	3
Step 4: Look-up posture score in Table A	7	7	9	5
Step 5: Add force/load score	2	3	3	3
Step 6: Score A, find row in Table C	9	10	12	8
Step 7: Locate upper arm position	3	3	5	6
Step 8: Locate lower arm position	1	1	2	2
Step 9: Locate wrist position	1	1	2	2
Step 10: Look-up Posture score in Table B	3	3	8	9
Step 11: Add coupling score	1	2	2	2
Step 12: Score B	4	5	10	11
Step 13: Activity score	1	1	1	1
Table C score	10	11	12	11
REBA Score	**11**	**12**	**13**	**12**

This deeper understanding helps identify structural weaknesses early, ensuring that the components are both durable and safe for long-term use. Such an approach is vital for improving the overall quality, reliability, and resilience of the engineering design, ultimately resulting in more robust products capable of withstanding real-world operational demands.

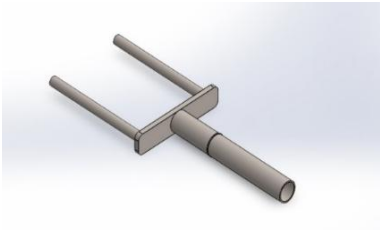


Fig.4. Hook

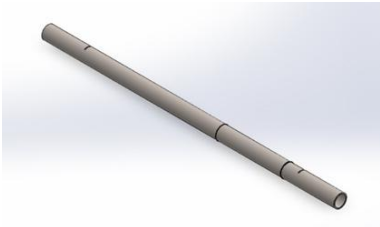


Fig.5. Shaft

3.3.1. Hook

The FEA simulation conducted in SolidWorks focuses on a model referred to as the hook, which is treated as a solid body throughout the analysis. A thorough understanding of the hook's volumetric properties, detailed in Table 2, is essential for an accurate evaluation. These properties provide crucial insights into the physical composition and mechanical behavior of the hook under various conditions.

Tab.2. Hook volumetric properties

Treated As	Volumetric Properties	
Solid Body	Mass	0.271854 kg
	Volume	3.53057e-05 m^3
	Density	7,700 kg/m^3
	Weight	2.66417 N

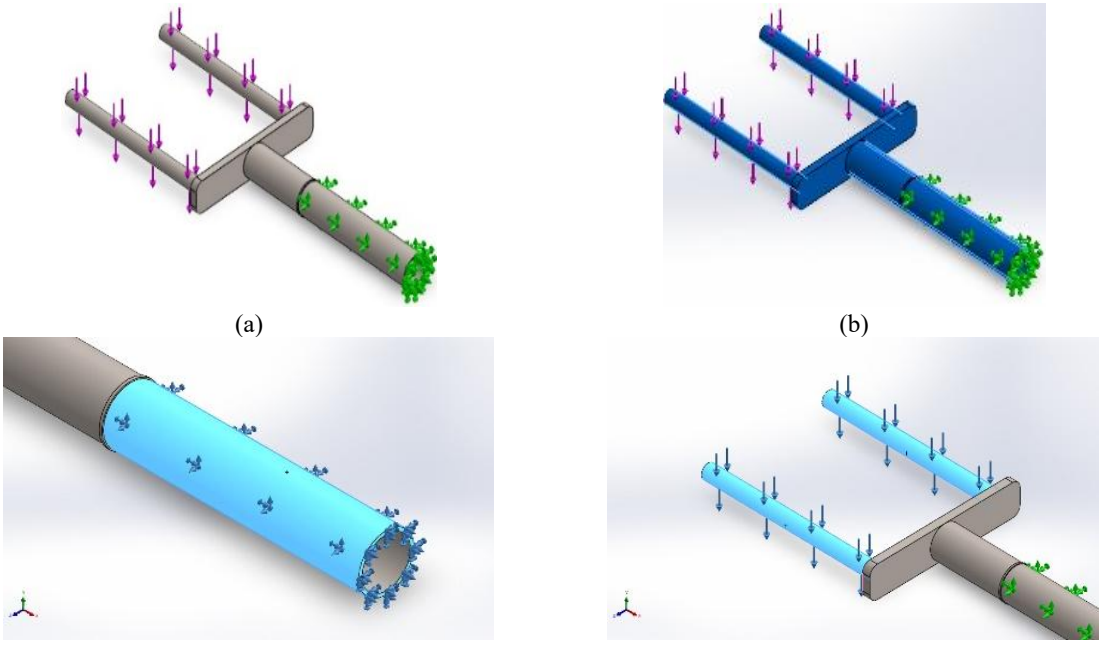
For this study, the simulation is configured as a static analysis, utilizing a solid mesh to ensure precise modeling of the component. Thermal effects are also incorporated, with temperature loads applied and the zero-strain temperature set at 298 Kelvin. It is important to highlight that the simulation parameters, including the hook's geometry and dimensions, are based on the specifications shown in Figure 6(a), ensuring consistency and accuracy in the structural assessment.

The material properties assigned to the hook model in the SolidWorks FEA simulation are carefully defined to ensure accurate modeling and analysis of its mechanical behavior. These properties, detailed in Table 3, correspond specifically to the hook's geometric characteristics, with the selected material illustrated in Figure 6 (b).

Tab.3. Hook material properties

Properties	
Name	**Alloy Steel**
Model type	Linear Elastic Isotropic
Default failure criterion	Max von Mises Stress
Yield strength	6.20422e+08 N/m^2
Tensile strength	7.23826e+08 N/m^2
Elastic modulus	2.1e+11 N/m^2
Poisson's ratio	0.28
Mass density	7,700 kg/m^3
Shear modulus	7.9e +10 N/m^2
Thermal expansion coefficient	1.3e-05/Kelvin

In setting up the FEA simulation, particular attention was given to fixture and load conditions. Two faces of the hook model are designated as fixed, simulating stable, restrained conditions based on the geometry. The load is applied to two faces along edge <1>, where a force of 147.15 N is imposed. This force corresponds to the gravitational load derived from a 15 kg mass multiplied by the acceleration due to gravity (9.81 m/s^2). These selections align precisely with the geometric specifications provided. This detailed configuration ensures a realistic simulation of the hook's structural behavior under specific boundary conditions and external forces, as illustrated in Figures 6(c) and 6(d), enabling a thorough evaluation of its performance in real-world scenarios. The analysis of resultant forces on the entire hook model reveals minimal displacement in the X and Z directions, while a significant force of 294.301 N is concentrated along the Y axis. Examination of reaction moments, free body forces, and moments confirms a state of equilibrium, underscoring the structural stability of the hook under the specified loading conditions. These refined meshing and analysis outcomes provide a comprehensive and detailed understanding of the hook's mechanical response, with a total resultant force of 294.3 N.



(a) (b)

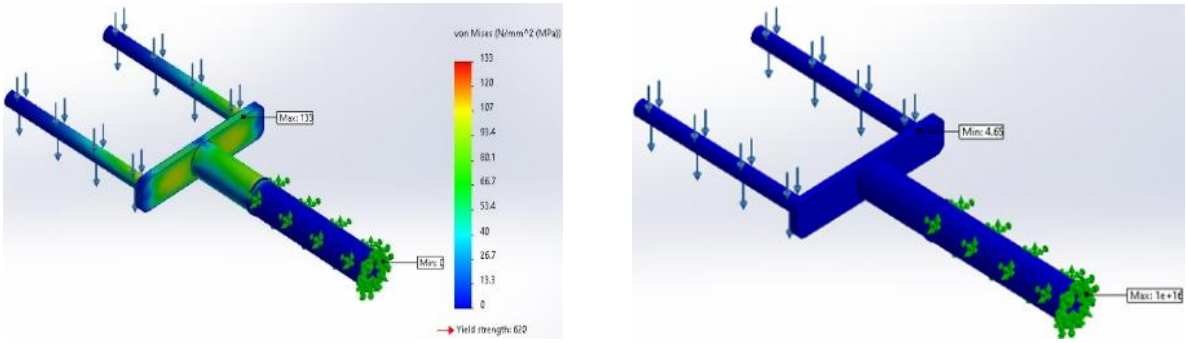
(c) (d)

Fig.6. Hook (a) Geometry, (b) Material selected, (c) Fixture, (d) Load

The FEA results for Stress 1 (von Mises stress) offer critical insights into the hook's material behavior under applied loads. The minimum stress recorded is 0 N/mm² at Node 36, while the maximum stress peaks at 133 N/mm² at Node 16387. These values are crucial for assessing the hook's structural integrity, with deformation scaled at 1 for accuracy. Given that the maximum von Mises stress of 133 N/mm² is well below the material's yield strength of 620 MPa, the hook is confirmed to operate safely within its material limits. These detailed stress distributions, illustrated in Figure 7(a), deepen the understanding of how the hook responds structurally under the specified load.

Furthermore, the Factor of Safety (FOS) analysis in SolidWorks provides vital information on the hook's safety margins. The minimum FOS of 4.65 at Node 16387 indicates a strong safety buffer, while the maximum FOS at Node 36 reaches an exceptionally high value of 1e16, highlighting extremely robust structural integrity. The color-coded FOS map—where red indicates values less than 2 (potential concern) and blue represents values of 2 or greater (safe)—offers a clear visual of critical and safe zones. This thorough FOS distribution, detailed in Figure 7(b), enables precise identification of potential weak points and confirms the hook's overall reliability. This in-depth evaluation ensures the design exceeds safety requirements, guaranteeing resilience and durability under anticipated operational loads without risking structural failure.

The FEA simulation conducted in Solidworks focuses on a robust hook model treated as a solid body with defined alloy steel material properties.



(a) (b)

Fig.7. Hook (a) Von Mises Stress, (b) Factor of Safety

The analysis employing a linear elastic isotropic model involves precise fixture and load specifications. Two faces are fixed to simulate restrained conditions, while a gravitational load of 147.15 N is applied to designated faces. Hence, the results indicate von Mises stress values ranging from 0 to 133 N/mm^2, well within the material's yield strength of 620 MPa. The FOS assessments reveal a minimum of 4.65 at Node 16387, indicating a substantial safety margin, and a maximum FOS of 1e^{16} at Node 36, showcasing exceptional robustness. Besides, visual representation through color coding enhances identification of critical zones, ensuring the hook model surpasses safety standards and maintains structural integrity against anticipated loads. This comprehensive analysis guarantees the structural design's resilience and reliability under specified conditions.

3.3.2. Shaft

The FEA simulation conducted in SolidWorks focuses on a shaft model, which is treated as a solid body throughout the analysis. The detailed specifications of the shaft are provided in Table 8. This study employs a static analysis framework with a solid mesh configuration, incorporating thermal effects to enhance simulation accuracy. Thermal considerations include the application of temperature loads and the specification of a zero-strain temperature set at 298 Kelvin. All aspects of the simulation adhere strictly to the geometric and boundary details outlined in Figure 8(a).

Tab.4. Shaft volumetric properties

Treated As	Volumetric Properties	
Solid Body	Mass	0.303959 kg
	Volume	3.94751e-05 m^3
	Density	7,700 kg/m^3
	Weight	2.97879 N
	Mass	0.000216131 kg
	Volume	2.80689e-08 m^3
	Density	7,700 kg/m^3
	Weight	0.00211808 N

The shaft material selected for the FEA simulation is alloy steel, modeled as a linear elastic isotropic material. The maximum von Mises stress criterion is used as the default failure indicator. The mechanical properties listed in Table 9 are essential for developing a detailed and precise simulation, offering in-depth insights into the alloy steel's mechanical and thermal behavior under various conditions, as illustrated in Figure 8(b).

Tab.5. Shaft material properties

Properties	
Name	**Alloy Steel**
Model type	Linear Elastic Isotropic
Default failure criterion	Max von Mises Stress
Yield strength	6.20422e+08 N/m^2
Tensile strength	7.23826e+08 N/m^2
Elastic modulus	2.1e+11 N/m^2
Poisson's ratio	0.28
Mass density	7,700 kg/m^3
Shear modulus	7.9e+10 N/m^2
Thermal expansion coefficient	1.3e-05 /Kelvin

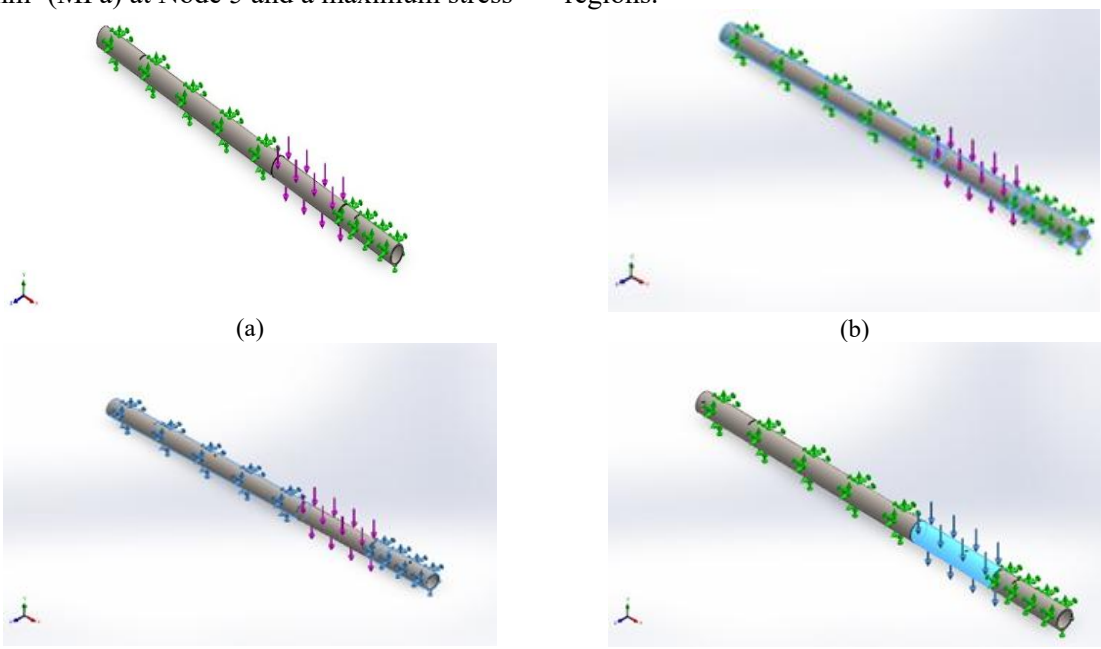
Fixture conditions involve constraining two faces using a fixed geometry type, indicated by blue arrows on the round shaft in Figure 8(c). This setup ensures these faces remain immobile during the analysis, establishing a stable reference frame for accurate simulation. Meanwhile, the applied load is defined as a force of 147.15 N on one face, corresponding to edge <1>. This force is calculated based on a 15 kg load subjected to gravitational acceleration of 9.81 m/s² and is visually represented by a blue arrow on the relevant shaft section in Figure 8(d). The careful selection and precise placement of fixtures and loads are critical to realistically replicating real-world conditions, enabling a comprehensive assessment of the shaft's structural response under specified external forces and boundary constraints in the FEA simulation.

The interaction within the model is governed by a global bonded interaction, which is applied to one component and visually indicated by a light blue color on the shaft's surface. This bonded interaction ensures that the connected surfaces remain fully adhered throughout the simulation, with no separation occurring, thereby guaranteeing a unified structural response. Additionally, a blue arrow marks the precise location of the applied load on the shaft's surface, perfectly aligned with the defined interaction parameters. This meticulous modeling of interactions, combined with clear visual indicators and strict adherence to specifications, contributes to a sophisticated simulation that accurately reflects the shaft's structural behavior under the applied loading conditions.

The analysis of resultant forces reveals a total load of 147.15 N acting along the Y-axis, while

forces in the X and Z directions are negligible. Reaction moments and free body moments throughout the entire model are uniformly zero, indicating a state of equilibrium. Calculated free body forces in the X, Y, and Z directions are minimal, with an overall resultant force of just 2.18×10^{-5} N. These precise results, detailed in Table 10, provide essential insights into the shaft's structural response under the applied loads, delivering a comprehensive understanding of the model's mechanical behavior.

The results reveal a minimum von Mises stress of 0 N/mm² (MPa) at Node 5 and a maximum stress of 6.81 N/mm² (MPa) at Node 3013. The deformation scale used in the analysis is set to 1, ensuring an accurate representation of displacement. These stress values serve as vital indicators of the shaft's structural performance: the minimum stress highlights areas with negligible stress concentration, while the maximum stress identifies potential points of concern. These findings are evaluated against the material's yield strength of 620 N/mm² (MPa), as shown in Figure 9 (a), providing crucial insights into stress distribution and helping to pinpoint critical regions.



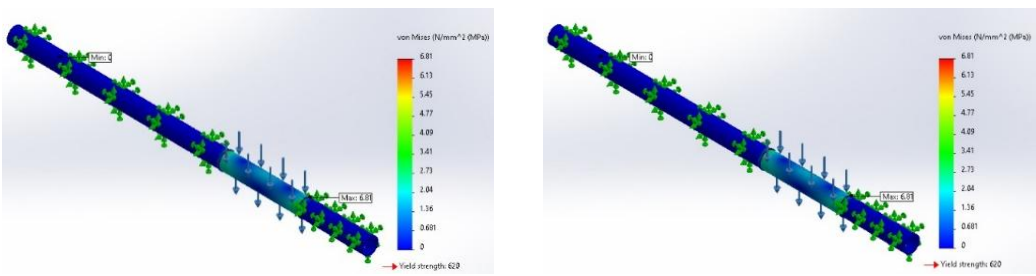
(a) (b)

(c) (d)

Fig.8. Shaft (a) Geometry (b) Material (c) Fixture (d) Load

Tab.6. Shaft resultant forces

Reaction Forces					
Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	-6.19303e-06	147.15	-1.20162e-06	147.15
Free Body Forces					
Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	-3.52199e-06	2.15233e-05	-8.68677e-07	2.18268e-05



(a) (b)

Fig.9. Shaft (a)Von mises stress (b) Factor of safety

This thorough assessment contributes to a comprehensive understanding of the shaft's structural integrity under the specified loading conditions.

The Factor of Safety (FOS) analysis conducted in the FEA offers essential insights into the shaft's structural robustness. The automatic computation of safety margins reveals a minimum FOS of 91.1 at Node 3013, indicating areas of substantial structural strength. Conversely, an exceptionally high maximum FOS of 1e16 at Node 5 reflects regions with an extremely large safety margin. As illustrated in Figure 9 (b), the color-coded visualization uses red to mark regions where the FOS is below 2, signaling potential vulnerability, and blue to indicate areas with an FOS above 2, representing ample safety. This detailed evaluation allows engineers to accurately identify weak spots and overly conservative zones, guiding targeted design improvements to optimize structural reliability.

The FEA simulation conducted in SolidWorks models the shaft as a solid body using alloy steel as the selected material. A load of 147.15 N, accurately derived from a 15 kg mass under gravitational acceleration, is applied to a single surface of the shaft to simulate operational stress conditions. The von Mises stress analysis (Stress1) identifies a minimum stress of 0 N/mm² at Node 5 and a maximum of 6.81 N/mm² at Node 3013, effectively pinpointing stress concentration areas that may present structural concerns. These values are assessed against the material's yield strength of 620 N/mm² to ensure structural safety. Additionally, the Factor of Safety (FOS1) analysis reveals a minimum FOS of 91.1 at Node 3013 and an extremely high maximum of 1e16 at Node 5, indicating both areas of potential concern and regions of over-engineering. A color-coded stress map is used to visually distinguish zones of varying safety levels, aiding in the identification of regions requiring optimization. This detailed evaluation confirms that the shaft design operates well within safe stress limits, ensuring structural integrity, operational durability, and reliability under real-world conditions.

4. Conclusion

This study provides critical insights into the ergonomic challenges firefighters face during hose rolling activities and evaluates a novel assistive tool designed to mitigate these risks. The Rapid Entire Body Assessment (REBA) method identified the flaking technique as particularly risk, with a score of 13 indicating a very high risk of MSDs. Similar assessments of other techniques, including roll and coil, Dutch roll, and figure of eight, also revealed elevated risk levels, underscoring the need for ergonomic interventions.

To address these risks, a Finite Element Analysis (FEA) was conducted on the proposed assistive tool, focusing on critical components such as the hook and alloy steel shaft. The simulation applied a load of 147.15 N, derived from a 15 kg mass under gravitational acceleration. The von Mises stress analysis revealed a maximum stress of 6.81 N/mm², significantly below the material's yield strength of 620 N/mm², indicating a robust design. The Factor of Safety (FOS) analysis further confirmed the tool's reliability, with a minimum FOS of 91.1, suggesting a high margin of safety.

While the findings are promising, the study has limitations. It was conducted in a controlled observational setting with a limited sample of male firefighters from a single location. No real-world implementation or user testing was performed, and the study focused solely on hose rolling tasks. Future research should include on-site trials to test the tool's usability and effectiveness, as well as broader ergonomic assessments across other firefighting tasks or physically demanding roles. User acceptance and comfort studies would also strengthen its applicability. Additionally, integrating digital human modeling and virtual reality simulations could further enhance ergonomic evaluations and tool design.

This study contributes to ergonomic tool development by demonstrating how structured risk assessment combined with engineering validation can inform safer task design in high-risk occupations. It reinforces the value of proactive ergonomic solutions to reduce injury risks and improve workforce well-being. The methodology and findings can serve as a model for developing ergonomic interventions in other physically demanding professions.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to

influence the work reported in this paper.

Acknowledgments

The authors thanks to Universiti Tun Hussein Onn Malaysia, National Institute of Occupational Safety and Health (NIOSH) with Grant No. K447, Bukit Gambir Fire station and Pagoh Fire Station for all contributions.

References

[1]. Kamat, S. R., Md Zula, N. E. N., Rayme, N. S., Shamsuddin, S., & Husain, K. (2017). The ergonomics body posture on repetitive and heavy lifting activities of workers in the aerospace manufacturing warehouse. IOP Conference Series: Materials Science and Engineering, 210(1). Retrieved from https://doi.org/10.1088/1757-899X/210/1/012079.

[2]. Mahmod, W. N. F. W., Abdullah, U. N. N., & Othman, N. (2020). Ergonomics Study among Operators in Water-Jet Production Area in the Aircraft Industry. International Journal of Automotive and Mechanical Engineering, 17(3), 8197–8205. Retrieved from https://doi.org/10.15282/ijame.17.3.2020.13.0617.

[3]. DOSH. (2018). Guidelines for Manual Handling at the Workplace 2018.

[4]. Vieira, E. R., & Kumar, S. (2007). Occupational risks factors identified and interventions suggested by welders and computer numeric control workers to control low back disorders in two steel companies. International Journal of Industrial Ergonomics, 37(6), 553–561. Retrieved from https://doi.org/https://doi.org/10.1016/j.ergon.2007.03.001.

[5]. Nunes, I. L. (2009). FAST ERGO_X–a tool for ergonomic auditing and work-related musculoskeletal disorders prevention. Work, 34(2), 133-148.

[6]. Boriboonsuksri, P., Taptagaporn, S., & Kaewdok, T. (2022). Ergonomic task analysis for prioritization of work-related musculoskeletal disorders among mango-harvesting farmers. Safety, 8(1), 6.

[7]. Hadi, M. H. I. B. A., Hadi, A., & Rahman, M. N. B. A. (2022). Ergonomic Risk Assessments (ERA) on cycling posture using rapid upper limb assessment (RULA) and rapid entire body assessment (REBA) method among cyclists. Research Progress in Mechanical and Manufacturing Engineering, 3(1), 801-811.

[8]. Afzal, M., Kiar, M., Nazri, M., Sabar, A., Fitri, M., Halim, A., & Daruis, D. D. I. (2019). Initial Ergonomic Risk Assessment on Unrolling and Rolling Fire Hose Activity among Firefighters at Putrajaya Fire and Rescue Station. In Human Factors and Ergonomics Journal (Vol. 4, Issue 2).

[9]. Mustapha, M. L. H., Mahmood, S., Bakri, H. M. E., Rahman, I. A., Azis, N. A., & Buang, M. R. (2023, February). Design of hose roller for firefighter: a fatigue study. In International Human Engineering Symposium (pp. 43-56). Singapore: Springer Nature Singapore.

[10]. Srirekha, A., & Bashetty, K. (2010). Infinite to finite: an overview of finite element analysis. Indian Journal of Dental Research, 21(3), 425-432.

[11]. Mahantesh, M. M., Rao, K. R., Chandra, A. P., Vijayakumar, M. N., Nandini, B., Prasad, C. D., & Vasudev, H. (2024). Design and modeling using finite element analysis for the sitting posture of computer users based on an ergonomic perspective. International Journal on Interactive Design and Manufacturing (IJIDeM), 18(8), 5875-5891.

[12]. Kinovea. (2021). Measuring angles—Kinovea 0.9.5 documentation. Retrieved from https://www.kinovea.org/help/en/measurement/angle.html.

[13]. Puig-Diví, A., Escalona-Marfil, C., Padullés-Riu, J. M., Busquets, A., Padullés-Chando, X., & Marcos-Ruiz, D. (2019). Validity and reliability of the Kinovea program in obtaining angles and distances using coordinates in 4 perspectives. PLoS ONE, 14(6). Retrieved from https://doi.org/10.1371/JOURNAL.PONE.0216448.

[14]. Hignett, S., & McAtamney, L. (2000). Rapid Entire Body Assessment (REBA). Applied Ergonomics, 31 (2): 201-5. http://doi: 10.1016/s0003-6870(99)00039-3.

[15]. Mahboobi, M., Taghipour, D., & Azadeh, M. (2020). Assessing Ergonomic Risk Factors Using Combined Data Envelopment Analysis

and Conventional Methods for an Auto Parts Manufacturer. Work (Reading, Mass.), 67. Retrieved from https://doi.org/10.3233/WOR-203257.

[16]. Boriboonsuksri, P., Taptagaporn, S., & Kaewdok, T. (2022). Ergonomic Task Analysis for Prioritization of Work-Related Musculoskeletal Disorders among Mango-Harvesting Farmers. Safety, 8(1). Retrieved from https://doi.org/10.3390/safety8010006.

[17]. Rahma, R. A. A., & Faiz, I. (2019). Work posture analysis of gamelan craft center workers using quick methods of ergonomic risk assessment. Journal of Physics: Conference Series, 1381(1), 012027. Retrieved from https://doi.org/10.1088/1742-6596/1381/1/012027.

[18]. Hita-Gutiérrez, M., Gómez-Galán, M., Díaz-Pérez, M., & Callejón-Ferre, Á. J. (2020). An Overview of REBA Method Applications in the World. International Journal of Environmental Research and Public Health, 17(8). Retrieved from https://doi.org/10.3390/IJERPH17082635.

[19]. Naeini, H. S., Dalal, K., Mosaddad, S. H. & Karuppiah, K. (2018). Economic Effectiveness of Ergonomics Interventions. International Journal of Industrial Engineering & Production Research. 29 (3), 261 – 276. Retrieved from DOI: 10.22068/ijiepr 29. 3.261.

[20]. Kohar, C. P., Greve, L., Eller, T. K., Connolly, D. S., & Inal, K. (2021). A machine learning framework for accelerating the design process using CAE simulations: An application to finite element analysis in structural crashworthiness. Computer Methods in Applied Mechanics and Engineering, 385, 114008.

[21]. Jagota, V., Sethi, A. P. S., & Kumar, K. (2013). Finite element method: an overview. Walailak Journal of Science and Technology (WJST), 10(1), 1-8.

Follow this article at the following site:

Salwa Mahmood, Ahmad Zahin Zainal Rasid, Nurul Ainina Nadhirah Tajurahim, Helmy Mustafa El Bakri, Ismail Abdul Rahman, Noorul Azreen Aziz. "Ergonomics Assessment of Hose-Rolling Posture and Simulation-Based Validation of an Assistive Tools for Hose Roller" IJIEPR 2025; 36 (4): 1-12
URL: http://ijiepr.iust.ac.ir/article-1-2340-en.html