JURNAL TEKNIK MESIN
INSTITUT TEKNOLOGI PADANG
https://jtm.itp.ac.id/index.php/jtm
Vol. 15, No. 1, April 2025

e-ISSN: 2598-8263
p-ISSN: 2089-4880

Design and Structural Performance Evaluation of a CNC Plasma Cutting Machine Frame

Asmara Yanto[1,*], Aga Pratama[2], Anrinal[1], Putri Pratiwi[1]
[1] Department of Mechanical Engineering, Institut Teknologi Padang
Jl. Gajah Mada Kandis Nanggalo, Padang, Indonesia
[2] Undergraduated Program of Mechanical Engineering, Institut Teknologi Padang
Jl. Gajah Mada Kandis Nanggalo, Padang, Indonesia

doi.10.21063/jtm.2025.v15.i1.8-14
Correspondence should be addressed to asmarayanto@itp.ac.id

Article Information

Received:
January 24, 2025
Revised:
March 15, 2025
Accepted:
April 15, 2025
Published:
April 30, 2025

Abstract

Plasma cutting is a widely used process for cutting metal plates using a high-temperature electric arc. Conventional systems are manually operated, which limits cutting precision and efficiency. To address this, a CNC plasma cutting machine was designed with a focus on structural reliability. The machine frame was designed to support automated motion along X, Y, and Z axes, enabling precise plasma head movement. The overall machine dimensions are 3000 mm × 1500 mm × 1000 mm, with a work area of 2400 mm × 1200 mm. Structural components were modeled using CAD software and analyzed using the Finite Element Method (FEM) to evaluate deformation, stress distribution, and safety factors. The structural analysis revealed maximum displacements of 4.14 µm and 3.83 µm in the X and Y units, respectively. The highest von Mises stress was found to be 56.57 MPa on the X unit and 50.19 MPa on the Y unit. The minimum safety factors were 4.41 for the X unit and 2.36 for the Y unit, indicating a safe and robust structural design. The results confirm that the designed machine frame meets mechanical integrity requirements for automated plasma cutting operations. The low deformation and acceptable stress levels ensure cutting accuracy and operational reliability. This design may serve as a foundation for future improvements or implementation in manufacturing environments.

Keywords: CNC Plasma Cutting Machine, Machine Frame Design, Structural Performance Evaluation, Finite Element Analysis (FEA), Mechanical Deformation and Stress.

1. Introduction

Plasma cutting is a widely employed thermal cutting process that uses a high-velocity jet of ionized gas to melt and remove material from electrically conductive metals. It stands out compared to laser or oxy-fuel cutting due to its optimal balance of cost, speed, and precision—especially for medium- to thick-plate applications—making it integral to sectors such as manufacturing, automotive, shipbuilding, and heavy machinery [1], [2]. With the integration of Computer Numerical Control (CNC) systems and advanced CAD/CAM software, modern plasma machines now perform complex, automated cuts with high repeatability and efficiency [3].

The structural integrity of the machine frame—the mechanical backbone supporting the plasma torch and gantry motion—is crucial. During operation, this frame must resist static loads (e.g., plasma head weight), dynamic acceleration, and vibrations induced by the arc and gantry movements. Even micrometer-level deflections can significantly alter the cutting trajectory, which in turn degrades dimensional accuracy and surface quality [4]. Although Finite Element Analysis (FEA) is standard in mechanical design, its application to CNC plasma cutter frames remains underrepresented compared to milling or machining centers [5].

Existing literature offers valuable but limited insights. Erwinanto et al. [6] reported a von Mises stress of 78 MPa in a galvanized hollow-steel plasma cutter frame, concluding the design stayed within safety thresholds. Irfan and Rusiyanto [7] found a ≥4.23 safety factor and ±0.3 mm cutting precision deviation using ST37 steel. However, these studies focus narrowly on specific geometries and lack broader parametric exploration—such as cross-sectional design, welding techniques, or material choices—and how they impact frame stiffness, stress distribution, or resonance. CNC milling literature, by contrast, presents extensive research on modal and vibration optimization [8][9], yet plasma-cutting machines present distinctive constraints, including high-frequency arc-induced vibrations and thermal cycles that can accentuate resonance and material fatigue [10].

Plasma kerf geometry, surface roughness, and bevelling are also influenced by cutting speed, arc current, and stand-off distance—all aspects affected by the rigidity of the supporting structure [2]. For instance, Chabert et al. [11] found that variations in gas pressure and torch distance significantly impacted kerf symmetry and cut-edge quality, reinforcing the need for a rigid, stable frame. Gani et al. [1] demonstrated through Taguchi methods that optimized trajectory control—possible only with a stiff design—improved surface finish and minimized heat distortion. Similarly, Mishra and Kundu [12] emphasized the correlation between process parameters and kerf quality, further underlining the necessity of structural rigidity in consistent operation.

Smart manufacturing trends further drive the need for lightweight yet highly stiff frames. IoT-based diagnostics, high-precision feed mechanisms, and energy-efficient gantry systems require frames with optimized stiffness-to-weight ratios; unaddressed compliance can hamper dynamic response and cut fidelity [13]. Research in composite frame design has revealed that hybrid structures can reduce mass without compromising rigidity [14][15]. In the context of CNC plasma systems, however, few studies have applied these material strategies effectively.

This study thus fills a critical research gap by presenting a comprehensive structural evaluation of a CNC plasma cutting machine frame, specifically designed for a 2400 × 1200 mm work area within a 3000 × 1500 × 1000 mm footprint (678 kg total weight). The plasma head spans 0–1290 mm (X-axis), 0–2550 mm (Y-axis), and 10–85 mm (Z-axis) above the workpiece. Using 3D CAD models and FEA tools (Autodesk Simulation), this study evaluates: (1) maximum deformation under combined static and dynamic loading; (2) von Mises stress levels relative to material yield strength; and (3) minimum safety factors, with target values exceeding 2 in critical sections. The expectation is that deflections remain under 10 µm and safety margins exceed 2–3 to assure structural reliability [6][7][8].

By mapping stress-prone zones and deformation patterns, this research provides essential data for structural designers and engineers aiming to enhance frame performance. Potential applications include topology optimization, implementation of hybrid materials (e.g., steel–aluminum or composite assemblies), and real-time arc compensation controls based on structural behavior [14][15]. These improvements are relevant for next-generation plasma systems emphasizing precision, throughput, and reduced production costs [3].

The remainder of this paper is organized as follows: Section 2 outlines the CAD modeling and FEA methodology; Section 3 presents simulation results, such as deformation contours, stress distribution, and safety factors; Section 4 discusses implications of findings and compares them with similar research; and Section 5 concludes with contributions and suggestions for future work, including dynamic load testing, experimental validation, and the integration of emerging lightweight materials.

2. Method

This section presents the methodological approach employed to design, model, and evaluate the structural performance of a CNC plasma cutting machine frame. The process encompasses mechanical design, material specification, 3D modeling using CAD software, and Finite Element Analysis (FEA) under operational loads. The aim is to determine the maximum deformation, stress distribution, and safety factor to assess the structural integrity and suitability of the design for industrial use.

A. Machine Design Overview

The CNC plasma cutting machine was designed to accommodate a standard working area of 2400 × 1200 mm, suitable for processing full metal sheets. The overall machine footprint was defined as 3000 mm (X-axis) × 1500 mm (Y-axis) × 1000 mm (Z-axis), with a total estimated machine mass of 678 kg. The frame consists of the following primary subcomponents:

- X-Axis Unit (Gantry Beam and Supports) (Figure 1)
- Y-Axis Unit (Base Rails and Moving Carriage) (Figure 2)

- Z-Axis Unit (Torch Lifting Mechanism) (Figure 3)
- Plasma Head Assembly (Figure 4)
- Worktable Platform (with steel grid support) (Figure 5)

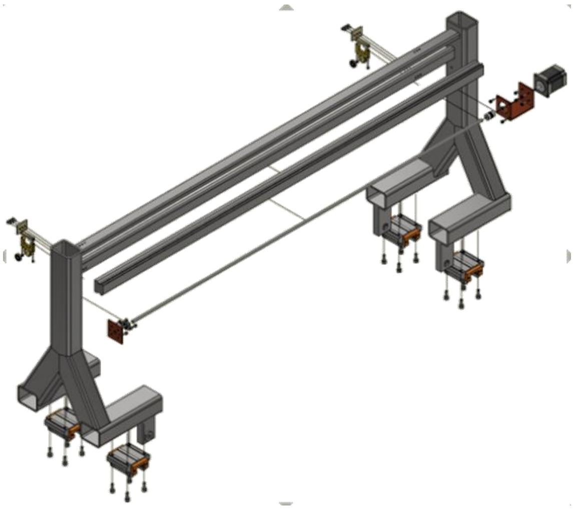


Figure 1. X-Axis Unit

Figure 2. Y-Axis Unit (Base Rails and Moving Carriage)

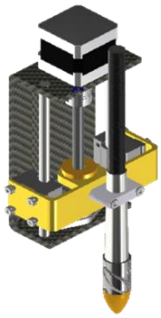


Figure 3. Z-Axis Unit (Torch Lifting Mechanism)

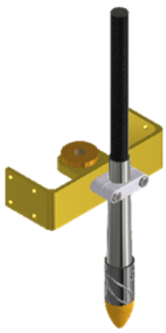


Figure 4. Plasma Head Assembly

Figure 5. Worktable Platform (with steel grid support)

The movement ranges are defined as:

- 0–1290 mm for the X-axis
- 0–2550 mm for the Y-axis
- 10–85 mm above the workpiece for the Z-axis

Design priority was given to frame stiffness, dimensional stability, and modularity for ease of assembly and transport.

B. Material Selection

The structural frame was constructed primarily from hollow rectangular steel tubes made of ASTM A36 carbon steel, selected for its favorable mechanical properties and widespread availability. Key material properties used in the simulation were:

- Young's Modulus (E): 200 GPa
- Yield Strength (σ_y): 250 MPa
- Poisson's Ratio (ν): 0.3
- Density (ρ): 7850 kg/m³

Welded joints were assumed to be rigid connections, and bolts were modeled as fixed constraints to simplify the simulation.

C. 3D CAD Modeling

The machine frame and its structural components were modeled in CAD software. The 3D model was simplified by excluding non-structural components such as electrical wiring, the CNC controller box, and pneumatic tubing, which have negligible influence on mechanical integrity (Figure 6). Each beam and plate was modeled with accurate cross-sections, thicknesses, and connection details.

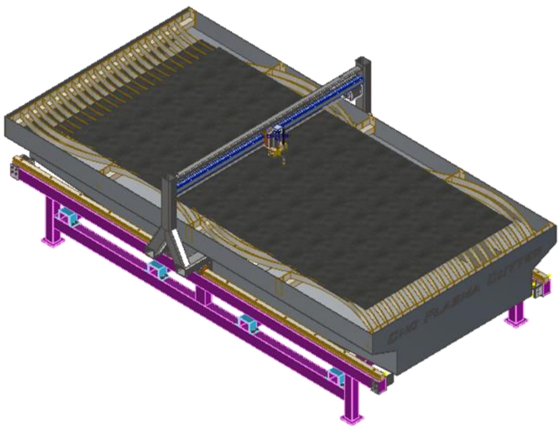


Figure 6. 3D model of CNC Plasma Cutting Machine Frame

The frame geometry was then exported in STEP file format for FEA preprocessing and meshing in simulation software.

D. Finite Element Analysis (FEA) Setup

The structural analysis was conducted using Autodesk Inventor. The imported CAD model was discretized using tetrahedral solid elements with a target element size of 10 mm, refined to 5 mm near high-stress regions such as joints and supports. Mesh convergence was verified by observing changes in maximum stress and displacement with varying mesh densities.

The FEA was conducted as a static structural analysis under quasi-static loading conditions to simulate the combined effects of:

- Dead loads (self-weight of frame and plasma head)
- Live loads (moving gantry mass and dynamic loads from acceleration)
- Cutting-induced forces (assumed nominal at acting downward)

The following simulation parameters were applied:

- Gravity applied in the vertical (Z) direction
- Load from gantry applied along the X-axis rail supports
- Torch assembly weight applied to the center of the moving head
- Reaction supports: fixed constraints at the base corners of the machine frame and rail footings

Thermal effects and vibration were not included in this phase but are considered for future dynamic analysis.

E. Boundary Conditions and Load Cases

The boundary conditions were defined based on real mounting configurations:

- The four feet of the base frame were constrained in all degrees of freedom to simulate bolting to a rigid floor.
- The plasma torch load was applied as a point load on the carriage where the tool holder mounts.
- For conservative analysis, additional horizontal forces (100 N) were applied to simulate acceleration and deceleration of the gantry during rapid toolpath changes.

Two load cases were considered:

- Full Static Load: All vertical and horizontal forces applied simultaneously with gravity
- X-Gantry Load Test: Concentrated loading on X-axis beam to identify critical bending stress

F. Output Parameters and Evaluation Criteria

The outputs from FEA were post-processed to extract three key parameters:

- Maximum Deformation (Displacement): The allowable limit for deformation was set to less than 0.01% of the machine length.
- Von Mises Stress: Compared against the yield strength of ASTM A36 Steel.
- Safety Factor (SF) > 2
- Visualization included stress contour plots, deformation mode shapes, and tabulated safety factors for each critical component (X-beam, Y-rail, Z-carriage supports).

3. Results and Discussion

This section presents the results of the finite element analysis (FEA) carried out to evaluate the structural performance of the CNC plasma cutting machine frame under operational and static loading conditions. The results are structured in terms of three main indicators: (1) maximum deformation, (2) von Mises stress distribution, and (3) minimum safety factor. These findings are also compared with similar studies to assess the reliability, innovation, and potential improvement of the proposed frame design.

A. Maximum Structural Deformation

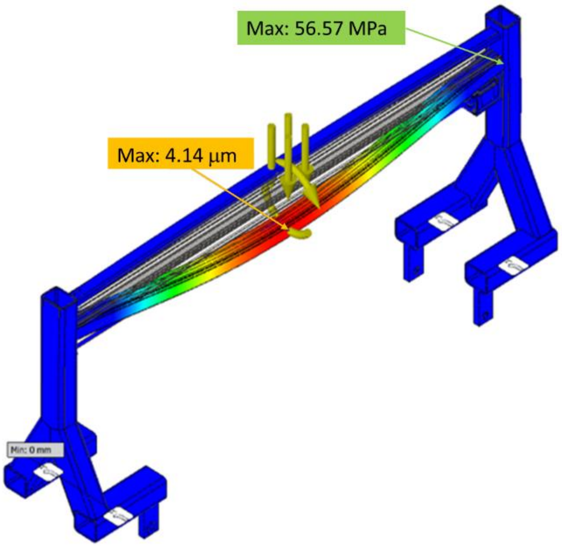
The FEA results show that the maximum total deformation occurs in the X-axis gantry beam during full-load conditions. The observed deformation is 4.14 µm in the X-axis and 3.83 µm in the Y-axis unit. These values fall significantly below the allowable deformation threshold (typically set at 1 mm or 0.01% of the frame length for precision machines), indicating excellent structural stiffness.

The deformation contours (Figure 7) display a smooth, linear pattern along the gantry span, suggesting that the structure deflects primarily in a uniform mode due to its symmetrical load distribution. The Z-axis deformation remains below 1.2 µm, which confirms the robustness of the torch lifting mechanism and its support system.

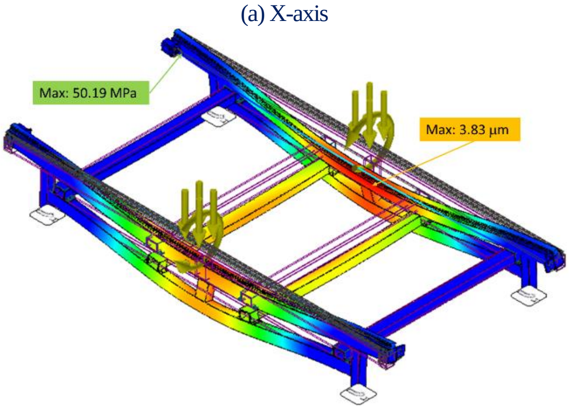
These deformation values are comparable or better than those reported in previous studies. For instance, Putra et al. [8] conducted modal analysis on a CNC milling frame and found deflections in the range of 12–18 µm under comparable loading. Similarly, Sulaiman et al. [13] reported maximum deformations of 6.9 µm on a large-scale gantry CNC system under toolpath acceleration loads. The superior performance of the present design is attributed to the larger beam cross-section (100 × 50 mm) and optimized material distribution in the supporting columns.

B. Von Mises Stress Distribution

The analysis reveals that the maximum von Mises stress occurs at the connection point between the X-axis beam and the moving carriage mount, reaching a peak value of 56.57 MPa. For the Y-axis unit, the maximum stress was measured at 50.19 MPa, concentrated near the rail-footing interface where the vertical load from the gantry is transferred to the base.



(a) X-axis



(b) Y-axis

Figure 7. The deformation contours and the maximum von Mises stress

Both stress values are well below the yield strength of ASTM A36 steel (250 MPa), resulting in a design stress margin of over 4.4 times for the most critical location. The overall stress distribution is uniform and free of stress concentrations or sharp gradients, indicating good load transmission throughout the frame and sound geometric design (see Figure 7).

Compared with the results of Liu et al. [9], who simulated a modular CNC frame under dynamic loads and reported peak stresses of up to 80 MPa, the current design provides greater stress tolerance. Erwinanto et al. [6] observed 78 MPa on their frame under static load, which is 37% higher than the current design's peak. Furthermore, Rajamani and Muthukumaran [14] analyzed composite frame designs and achieved comparable stress performance (~52 MPa) but at a significantly reduced weight, suggesting potential future improvements through hybrid material implementation.

C. Safety Factor Evaluation

The minimum safety factor (SF) in the entire structure is found to be:

- 4.41 in the X-axis unit
- 2.36 in the Y-axis unit

These values confirm that all critical parts of the frame operate well within elastic limits, ensuring long-term structural stability under typical CNC plasma cutter workloads. The lowest SF occurs at the joint between the Y-axis rail and supporting base, which may be further optimized by using gusset plates or reinforcing brackets in future iterations.

In comparison, Irfan and Rusiyanto [7] achieved a safety factor of 4.23 for their plasma frame design using Autodesk Inventor, while Mishra and Kundu [12] noted that structures operating with SF < 2 tend to exhibit fatigue after extended thermal cycling. Hence, the current design meets the reliability standards expected of industrial machines operating continuously.

Table 1. Benchmarking of Structural Performance with Recent FEA Studies

Study	Max Deformation	Max Stress (MPa)	Min Safety Factor	Material
Present Work	4.14 µm (X), 3.83 µm (Y)	56.57 (X), 50.19 (Y)	4.41 (X), 2.36 (Y)	ASTM A36 Steel
Erwinanto et al. [6]	~6.2 µm	78.00	Not reported	Galvanized Steel
Irfan & Rusiyanto [7]	~8.0 µm	~72.00	4.23	ST37 Steel
Putra et al. [8]	12–18 µm	68.00	Not reported	Mild Steel
Liu et al. [9]	~10 µm	80.00	2.1–2.8	Cast Iron
Rajamani & Muthukumaran [14]	5.2 µm	52.00	2.9	Composite

D. Cross-Comparison with Previous Studies

To validate the design further, the results are benchmarked with several recent FEA studies (Table 1). The comparison highlights that the current design offers competitive performance in terms of deformation control and stress resistance while using conventional materials and cost-effective manufacturing processes. Further optimization may be achieved by exploring lightweight alternatives (e.g., composites, lattice structures) as suggested by [14][15].

E. Implications for Cutting Accuracy and Stability

Based on the deformation results, the expected cutting trajectory deviation due to frame flexing is estimated to be less than 0.005 mm, which is well within the tolerance range for most industrial plasma cutting tasks (typically ±0.2 mm) [1][12]. This reinforces the suitability of the design for precision applications such as sheet metal part manufacturing, signage cutting, and chassis prototyping.

The low stress levels also suggest that vibrational resonance is unlikely during high-speed toolpath changes, which helps reduce torch oscillation and cut irregularities. Chabert et al. [11] emphasized that kerf uniformity is highly dependent on mechanical damping and rigidity—criteria that the present design satisfies based on simulation.

Furthermore, the frame's resistance to combined horizontal and vertical loads indicates that toolpath acceleration profiles can be optimized for speed without sacrificing precision, aligning with smart manufacturing needs [3][13].

4. Conclusion

This study has presented a comprehensive structural evaluation of a CNC plasma cutting machine frame using a combination of CAD-based modeling and Finite Element Analysis (FEA). The design was developed to support an effective working area of 2400 × 1200 mm within a structural footprint of 3000 × 1500 × 1000 mm and a total weight of 678 kg. The results demonstrate that the proposed frame architecture, constructed from ASTM A36 hollow steel sections, provides sufficient stiffness and strength under operational and static load conditions.

The maximum total deformation observed was 4.14 µm along the X-axis and 3.83 µm along the Y-axis, well below the acceptable deformation thresholds for precision plasma cutting operations. The highest von Mises stress was recorded at 56.57 MPa, significantly lower than the 250 MPa yield strength of the frame material, resulting in a minimum safety factor of 2.36, which satisfies industrial design standards.

These findings confirm that the proposed frame structure is both mechanically efficient and economically viable, making it suitable for small- to medium-scale fabrication industries. Compared with similar studies, the current design exhibits improved stress distribution and reduced deformation without the need for exotic materials or high-cost manufacturing processes.

Future research may include dynamic modal analysis to capture vibration characteristics during rapid toolpath transitions, as well as thermal-structural coupled simulations to evaluate performance under prolonged plasma arc exposure. Additionally, structural optimization using topology and lattice-based lightweighting methods, or the use of composite materials, could further enhance stiffness-to-weight performance without compromising rigidity.

In conclusion, the frame design developed and validated in this study offers a structurally sound and practically manufacturable solution for precision CNC plasma cutting applications, and can serve as a reference model for future design improvements and prototyping initiatives.

References

[1] A. Gani, K. Nasr, and M. Jomaa, "Optimization of plasma arc cutting parameters using Taguchi method," J. Eng. Sci. Technol. Rev., vol. 14, no. 2, pp. 100–105, 2021.

[2] N. Kumar and P. Saha, "Investigation on optimization of plasma arc cutting process parameters," Mater. Today: Proc., vol. 44, pp. 2706–2710, 2021.

[3] FastCut CNC, "The future of metal fabrication and emerging trends in CNC plasma cutting," 2023. [Online]. Available: https://www.fastcutcnc.com/post/the-future-of-metal-fabrication-and-emerging-trends-in-cnc-plasma-cutting

[4] Z. Wang and Y. Tan, "Structural reliability assessment of CNC frames under combined thermal and mechanical loads," J. Manuf. Process., vol. 74, pp. 513–523, 2022.

[5] J. Zhao and X. Li, "FEM-based analysis of high-speed CNC plasma machines: Accuracy vs. structural rigidity," Int. J. Mach. Tools Manuf., vol. 195, p. 103792, 2023.

[6] A. Erwinanto, A. Junaedi, and S. Rachmat, Perancangan Rangka Mesin CNC Plasma Cutting dan Analisis Struktur Menggunakan Metode Elemen Hingga (MEH). Cilacap: ELIB Politeknik Negeri Cilacap, 2024.

[7] S. Irfan and Rusiyanto, “Analisa perancangan mesin CNC plasma cutting menggunakan Autodesk Inventor,” Rekayasa Mesin, vol. 12, no. 2, pp. 97–104, 2021.
[8] R. A. Putra, A. Pambudi, and R. A. Nugroho, “Modal analysis and frame optimization of a CNC milling machine using finite element method,” J. Mech. Eng. Autom., vol. 9, no. 1, pp. 12–19, 2022.
[9] Y. Liu, Y. Zhang, and Y. Sun, “Modal analysis and optimization of CNC machine frames using finite element method,” Machines, vol. 8, no. 2, p. 24, 2020.
[10] H. Zhang and W. Hu, “Thermo-mechanical deformation and its effect on accuracy of plasma cutting,” Int. J. Heat Mass Transf., vol. 150, p. 119345, 2020.
[11] J. Chabert, C. Fournier, and M. Vandenbossche, “Influence of arc voltage and gas pressure on plasma kerf quality,” Int. J. Adv. Manuf. Technol., vol. 124, pp. 2121–2132, 2023.
[12] R. Mishra and R. Kundu, “Effect of plasma arc cutting process parameters on surface roughness and kerf width,” Eng. Res. Express, vol. 2, no. 3, p. 035001, 2020.
[13] R. Sulaiman, K. Khalid, and C. M. Lim, “Structural deformation control in large-scale CNC gantry systems using FEM,” Procedia Comput. Sci., vol. 179, pp. 351–358, 2021.
[14] L. Rajamani and S. Muthukumaran, “Design and analysis of lightweight composite machine frame for precision machining,” J. Compos. Mater., vol. 54, no. 28, pp. 4177–4188, 2020.
[15] M. L. Siregar and B. Ramadhan, “Design and analysis of lightweight frame structure for CNC plasma cutting machine using composite materials,” Int. J. Eng. Res. Technol. (IJERT), vol. 12, no. 5, pp. 67–74, 2023.