International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

Lightweight Design of a 90M³ Side Dump Trailer Web Flange Subframe Using Manufacturing-Constrained Topology Optimization

[1]*Totok Haryanto, [2]Sulardjaka, [3]Agus Suprihanto

[1,2,3]Department of Mechanical Engineering, Diponegoro University, Semarang, Central Java, Indonesia
*Corresponding Author's E-mail: totok.haryanto78@gmail.com

***Abstract* - Reducing the structural weight of heavy-duty mining trailers is an effective strategy for improving payload capacity, fuel efficiency, and overall operational productivity. Although topology optimization has been widely applied to lightweight structural design, its industrial implementation is often constrained by the generation of complex geometries that are difficult to manufacture using conventional fabrication processes. This study proposes a manufacturing-constrained topology optimization approach for the web flange subframe of a 90 m³ side dump trailer, aiming to achieve weight reduction while preserving structural integrity and fabrication feasibility. A 3D finite element model of the subframe was developed in SolidWorks Simulation using the actual operational loading conditions of the trailer. Baseline static analysis was first performed to identify regions with low material utilization, followed by topology optimization to determine the optimal material distribution. The optimized topology was subsequently reconstructed into a manufacturable design by reducing the flange plate thickness from 12 mm to 10 mm in non-critical regions while maintaining continuous load paths around the axle mounting zones. A secondary finite element analysis was conducted to validate the redesigned structure under identical loading conditions. The optimized design reduced the subframe mass from 1,805.57 kg to 1,613.29 kg, corresponding to a weight reduction of 10.6%. Although the maximum Von Mises stress increased from 26.0 MPa to 46.4 MPa, the redesigned structure maintained a satisfactory factor of safety of 7.54, remaining well below the material yield strength. The proposed approach demonstrates that integrating topology optimization with manufacturing-oriented reconstruction provides a practical and production-ready lightweight design solution for heavy-duty mining trailers, offering improved structural efficiency without compromising safety or manufacturability.**

Keywords: Topology optimization; finite element analysis; lightweight design; web flange subframe; side dump trailer; manufacturing-constrained design.

I. INTRODUCTION

The mining industry continuously demands heavy-duty transportation equipment with higher payload capacity, lower operating costs, and improved energy efficiency. Side dump trailers play a crucial role in bulk material transportation, where structural performance directly influences vehicle productivity, fuel consumption, maintenance costs, and overall operational efficiency. As payload regulations and production targets become increasingly stringent, reducing the structural weight of trailer components without compromising safety has become one of the primary challenges in modern heavy-vehicle engineering. Lightweight structural design not only increases payload capacity but also reduces fuel consumption, greenhouse gas emissions, and component fatigue during long-term operation, thereby improving the sustainability of mining transportation systems [1], [3].

One of the most effective approaches for achieving lightweight structures is structural optimization using finite element analysis (FEA) and topology optimization techniques. Finite element analysis enables engineers to accurately evaluate stress distribution, deformation, and structural safety under realistic loading conditions, while topology optimization systematically identifies inefficient material distribution by removing low-stress regions and preserving load-carrying paths [4], [5]. Over the past decade, topology optimization has been widely adopted in aerospace, automotive, civil engineering, and heavy equipment industries because it provides an efficient computational framework for maximizing structural stiffness while minimizing material usage [6], [7].

Several studies have demonstrated the successful application of topology optimization for heavy vehicle structures. Padhiyar[3] investigated weight reduction of truck

International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

chassis frames using finite element analysis, while Kilic and Topaç[2] optimized heavy-duty truck chassis under multiple loading conditions to improve stiffness-to-weight performance. Agarwal and Sitembu[5] further applied structural optimization techniques to commercial vehicle chassis, whereas Wang and Zhang [6] combined topology optimization with experimental verification for lightweight subframe structures. More recent studies have integrated topology optimization with multi-objective design strategies considering fatigue life, crashworthiness, and manufacturing limitations [8], [10]. These studies collectively confirm that topology optimization is an effective methodology for reducing structural mass while maintaining acceptable mechanical performance.

Despite these advances, several limitations remain. Most previous investigations have focused on complete chassis systems or generalized vehicle frames, whereas localized optimization of critical structural members has received relatively limited attention. Furthermore, many published studies present mathematically optimal topologies characterized by complex organic geometries that are impractical or economically infeasible to manufacture using conventional steel fabrication processes. As a result, a considerable gap still exists between computational topology optimization and its implementation in industrial production environments, particularly for heavy-duty mining trailers where fabrication simplicity, welding accessibility, and maintenance considerations are equally important as structural efficiency.

The web flange subframe represents one of the primary load-carrying components of a side dump trailer. This structural member transfers payload-induced forces from the vessel body to the suspension and axle assemblies while resisting bending and torsional loads generated during loading, transportation, and dumping operations. Because of its significant contribution to the overall structural weight, the web flange subframe offers substantial opportunities for lightweight redesign. However, any reduction in material must ensure that the resulting structure satisfies strength requirements and remains compatible with existing manufacturing processes.The overall structure and initial geometry of the subframe component prior to the optimization process are illustrated in detail in Figure 1.

To address these challenges, this study proposes a manufacturing-constrained topology optimization approach for the web flange subframe of a 90 m³ mining side dump trailer.

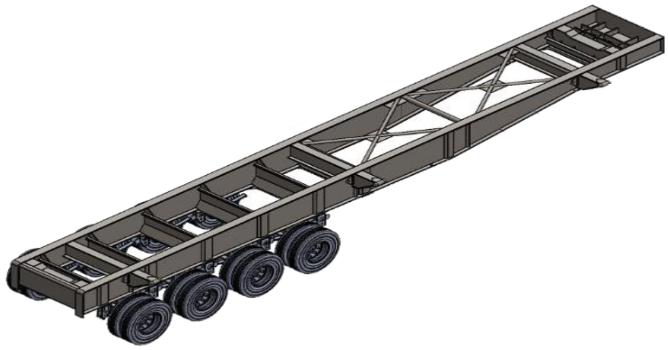

Figure 1: 3D model of the side dump trailer chassis high- lighting the web flange subframe & axle assembly

Unlike conventional topology optimization studies that primarily emphasize mathematical optimality, the proposed methodology integrates finite element analysis, topology optimization, and engineering reconstruction to produce a simplified, production-ready design that can be fabricated using standard industrial manufacturing techniques. Material removal is performed only within non-critical regions while preserving continuous load paths around critical mounting interfaces, thereby ensuring both structural safety and manufacturability.

The main contributions of this study are threefold. First, a finite element-based structural assessment is performed to identify regions with excessive safety margins and low material utilization in the original subframe design. Second, a manufacturing-constrained topology optimization framework is developed to achieve lightweight structural redesign while maintaining critical load-transfer paths. Third, the optimized configuration is reconstructed into a practical engineering design and subsequently validated through secondary finite element analysis under identical loading conditions. The proposed methodology demonstrates that significant structural weight reduction can be achieved without compromising mechanical integrity, providing a practical design strategy for lightweight heavy-duty mining trailers.

II.METHODOLOGY

2.1 Research Framework

This study employed a numerical simulation approach based on finite element analysis (FEA) and topology optimization to redesign the web flange subframe of a 90 m³ side dump trailer. The overall research methodology consisted of four sequential stages: (1) development of the baseline finite element model, (2) static structural analysis to evaluate the initial mechanical performance, (3) topology optimization to identify low-stress regions suitable for material removal, and (4) reconstruction and verification of the optimized design under identical loading conditions.

International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

This workflow enabled the evaluation of structural performance before and after optimization while ensuring that the redesigned component remained suitable for industrial manufacturing.

The investigated component was the web flange subframe located between the trailer vessel and the axle assembly. This component serves as the principal load-transfer structure, carrying vertical and torsional loads generated during hauling and dumping operations. Because of its significant contribution to the total structural weight, the web flange subframe was selected as the optimization target.

2.2 Material Properties

The subframe assembly was manufactured using structural steels JIS G3101 SM490 and SS400, which are commonly employed in heavy-duty trailer fabrication due to their high strength, weldability, and durability under severe operating conditions. The finite element model assumed homogeneous, isotropic, and linearly elastic material behavior throughout the simulation.

The mechanical properties assigned to the model included Young's modulus, Poisson's ratio, density, and yield strength according to the corresponding material specifications. These material parameters were used consistently throughout both the baseline analysis and the verification of the optimized structure.

2.3 Finite Element Model (FEA)

A 3D CAD model of the web flange subframe was developed using SolidWorks and subsequently imported into SolidWorks Simulation for structural analysis. The baseline geometry employed a uniform flange plate thickness of 12 mm, representing the existing production design.

The computational domain was discretized using high-quality second-order tetrahedral solid elements. Local mesh refinement was applied around stress concentration regions, particularly near the axle mounting interfaces and welded joints, to improve numerical accuracy while maintaining reasonable computational efficiency.

2.4 Boundary Conditions and Loading

The boundary conditions are detailed in Table 1,were established to represent the actual operational condition of a fully loaded 90 m³ side dump trailer. The suspension mounting locations were constrained to simulate the support provided by the axle assembly, while the payload was represented by an equivalent vertical static load distributed through the vessel support structure.

Table 1: Material and Boundary Conditions

Parameter	Information
Material	JIS G3101 SM490 & SS400
Payload	$90\ m^3$
Material Density (ρ)	$900\ kg/m^3$
Applied Vertical Force	$\frac{N_r}{2} = 391.029$ N

The maximum applied load was 391,029 N, corresponding to the fully loaded operational condition. This loading configuration represents the most critical static loading scenario experienced during material transportation before the dumping operation.

The applied constraints and loading conditions ensured that the numerical model realistically represented the primary load path from the trailer vessel to the suspension system.

2.5 Baseline Structural Analysis

Before performing topology optimization, a baseline FEA was conducted to evaluate the structural performance of the original design. The static analysis calculated: Von Mises stress distribution; total displacement; equivalent strain; and Factor od Safety (FOS).

The stress distribution was evaluated according to the Von Mises yield criterion, while the factor of safety was calculated by comparing the material yield strength with the maximum equivalent stress.

The baseline analysis provided the reference structural performance and identified regions with excessive safety margins, which subsequently became candidates for material reduction during the optimization process.

2.6 Topology Optimization

Topology optimization was performed using the Topology Study module available in SolidWorks Simulation. The optimization objective was to maximize the stiffness-to-weight ratio while satisfying the prescribed structural constraints.

International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

Table 2: Topology Optimization Parameters

Parameter	Information
Optimization Goal	Best Stiffness to Weight Ratio
Mass Constraint	Reduce Mass by 20%
Manufacturing Constraint	Continuous load path for axle mounting

Based on these parameters, the optimization process iteratively evaluated the strain energy density of each element. Material in low-stress zones was progressively penalized and removed from the subframe assembly. Concurrently, a manufacturing constraint was enforced to maintain a continuous load path for the subsequent manufacturing phase, ensuring structural integrity during high-stress operations likes the axle mounting [2], [13]. The optimization loop terminated automatically upon solution convergence [14], achieving the targeted 20% mass reduction. The optimization process was terminated automatically after satisfying the convergence criteria specified by the software.

2.7 Manufacturing-Oriented Design Reconstruction

The optimized topology generated by the numerical solver produced irregular geometries that were unsuitable for direct manufacturing. Therefore, the optimization results were interpreted and reconstructed into a simplified engineering design suitable for industrial fabrication.

Instead of introducing complex internal cut-outs, the redesign strategy focused on reducing the flange plate thickness from 12 mm to 10 mm within the identified low-stress regions while maintaining the original structural configuration in critical load-transfer areas. This reconstruction approach preserved continuous load paths and ensured compatibility with existing workshop fabrication procedures, including cutting, welding, and assembly. The reconstructed geometry represents a balance between computational optimization and practical manufacturability.

2.8 Structural Verification

A secondary finite element analysis was performed to validate the structural performance of the reconstructed design. The optimized model was remeshed and analyzed using the same material properties, loading conditions, mesh strategy, and boundary conditions applied during the baseline simulation. The verification focused on comparing: maximum Von Mises stress; maximum displacement; factor of safety; and total structural mass.

The effectiveness of the proposed methodology was evaluated by comparing the mechanical performance of the redesigned structure with that of the original configuration. The optimized design was considered acceptable if the resulting stresses remained below the material yield strength while achieving a meaningful reduction in structural mass.

III. RESULTS AND DISCUSSIONS

3.1 Baseline Structural Evaluation

The initial finite element analysis was conducted to evaluate the structural performance of the original web flange subframe with a uniform plate thickness of 12 mm. Under the maximum operational load of 391,029 N, the maximum von Mises stress reached 26.0 MPa, which is significantly lower than the yield strength of the structural steel. The corresponding factor of safety (FOS) was 13.4, indicating that the original design possessed a considerable structural reserve.

These empirical results mathematically pin-pointed extensive under-stressed and over-designed regions across the subframe. The dominance of low material utilization zones provided a solid engineering justification for systematic mass reduction without risking structural failure[1], [15].The preliminary phase of this study involved a baseline static evaluation using FEA to determine the mechanical behavior of the original web flange subframe in Figure 2 and Figure 3.

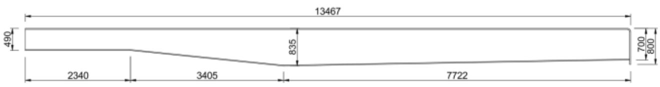

Figure 2: Engineering drawing of the original web flange subframe with key dimensions

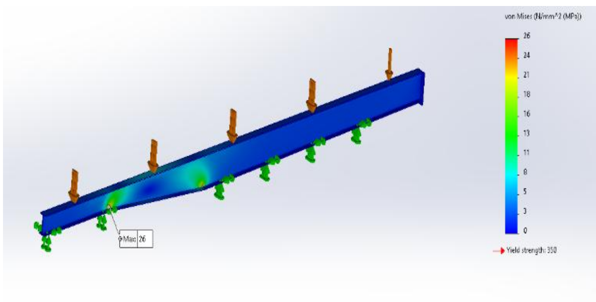

Figure 3: Initial static FEM analysis showing low material utilization

The stress distribution revealed that only a limited portion of the subframe experienced relatively high stress, while large areas remained underutilized. This finding indicates that the original structure was overdesigned and

International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

provided sufficient opportunity for material reduction without compromising structural safety.

3.2 Topology Optimization and Design Modification

Based on the baseline analysis, topology optimization was performed to identify regions with low structural utilization. The optimization results suggested that material could be safely removed from several non-critical areas while maintaining the primary load-transfer path between the trailer vessel and the axle assembly.

Because the raw topology generated irregular geometries that were unsuitable for fabrication, the optimized model was reconstructed into a practical engineering design. Instead of introducing complex openings, the flange plate thickness was reduced from 12 mm to 10 mm in low-stress regions while preserving the original geometry around the critical mounting locations. This approach simplified manufacturing using conventional cutting and welding processes.

The topology study generated a material mass plot highlighting under-utilized zones [16].This design iteration accommodated standard workshop cutting and fabrication processes are presented in Figure 4, Figure 5, and Figure 6.

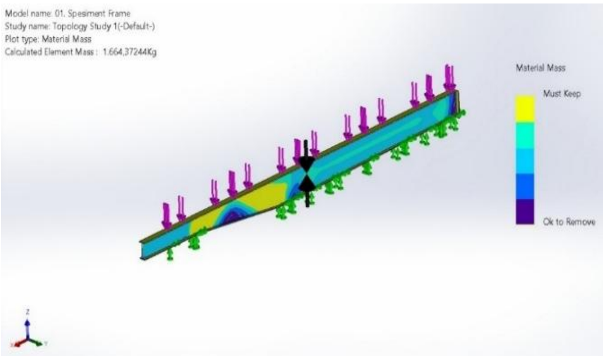

Figure 4: Topology plot identifying target areas for mass elimination

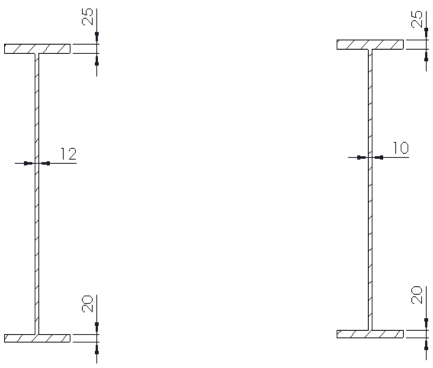

Figure 5: Cross-sectional comparison of the original (a) and redesigned (b) web flange subframe

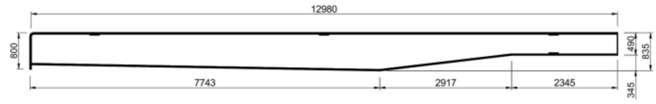

Figure 6: Detailed engineering drawing of the redesigned web flange subframe

3.3 Structural Verification

The reconstructed model was verified using the same loading and boundary conditions applied to the baseline model. The redesigned subframe exhibited a maximum von Mises stress of 46.4 MPa (Figure 7), representing an increase compared with the original design due to the reduction in plate thickness. Nevertheless, the stress remained well below the material yield strength, indicating that the redesigned structure retained adequate structural integrity.

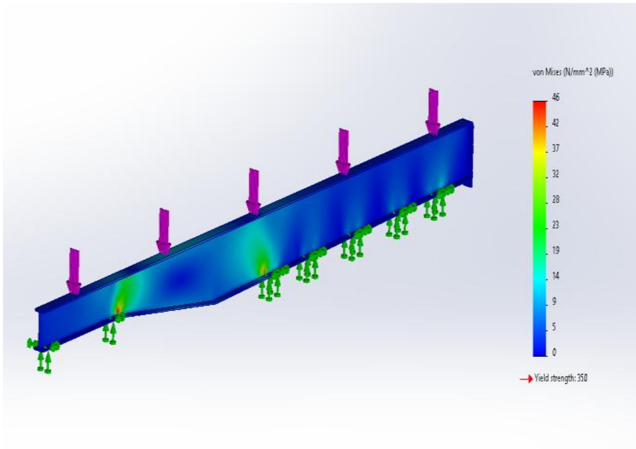

Figure 7: Stress verification of the redesigned web flange

The factor of safety decreased from 13.4 to 7.54, which is still considered sufficient for heavy-duty structural applications. At the same time, the total subframe mass decreased from 1,805.57 kg to 1,613.29 kg, corresponding to a weight reduction of 10.6%.The comparison between the original and optimized designs is summarized in Table 3.

Table 3: Comparison of Original and Redesigned Metrics

Parameter	Original Design	Re-design	Change
Flange Thickness	12 mm	10 mm	16.7%
Total Mass	1,805.57 kg	1,613.29 kg	10.6%
Dimension (Length Total)	13,467 mm	12,980 mm	3.6%
Factor of Safety	13.4	7.54	43.7%

**Note: Baseline and redesign mass figures correspond to evaluated subframe assemblies.*

Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

Overall, the results demonstrate that the proposed optimization strategy effectively improved material utilization by reducing unnecessary structural mass while maintaining a satisfactory safety margin. The optimized design also offers practical advantages for manufacturing because the final geometry can be produced using conventional fabrication techniques without introducing complex structural features.

The reduction in structural weight is expected to improve payload capacity and reduce operational costs, making the proposed design suitable for heavy-duty mining trailer applications.

IV. CONCLUSION

This study investigated the application of manufacturing-constrained topology optimization to reduce the structural weight of the web flange subframe in a 90 m³ side dump trailer. Finite element analysis of the original design revealed a maximum von Mises stress of 26.0 MPa and a factor of safety of 13.4, indicating significant potential for material reduction due to the excessive structural reserve.

Topology optimization was subsequently performed to identify low-stress regions suitable for weight reduction. The optimized topology was reconstructed into a practical and manufacturable design by reducing the flange plate thickness from 12 mm to 10 mm in non-critical areas while preserving the primary load-transfer paths and critical mounting interfaces. The redesigned structure was then verified through a secondary finite element analysis under identical loading conditions.

The optimized subframe achieved a mass reduction of 10.6%, decreasing the total weight from 1,805.57 kg to 1,613.29 kg. Although the maximum von Mises stress increased to 46.4 MPa, the stress level remained well below the material yield strength, resulting in a satisfactory factor of safety of 7.54. These results confirm that the proposed optimization approach successfully improves material utilization while maintaining structural integrity and operational safety.

From an engineering perspective, the proposed methodology provides a practical framework for integrating topology optimization with conventional fabrication requirements in heavy-duty trailer design. The resulting lightweight structure has the potential to enhance payload capacity, improve transportation efficiency, and reduce operational costs without requiring significant modifications to existing manufacturing processes.

Future research should investigate the structural performance of the optimized design under dynamic loading, fatigue conditions, and impact scenarios that more closely represent actual mining operations. Experimental validation and full-scale field testing are also recommended to further verify the long-term reliability of the proposed design.

ACKNOWLEDGEMENT

In this optional section, the authors may acknowledge any individuals or institutions that contributed to the research.

REFERENCES

[1] Wood, J., & Cebon, D. (2017). Material selection and structural optimisation for lightweight truck trailers. *International Journal of Vehicle Design*, 74(1), 23-45.
[2] Kilic, M., & Topaç, M. M. (2019). Topology Optimization of a Heavy-Duty Truck Chassis Under Multiple Loading Conditions. *Journal of Optimization in Mechanical Engineering*, 7(2), 89-102.
[3] Padhiyar, B. (2014). Optimization of truck chassis frame for weight reduction. *International Journal of Future Technology and Engineering Research*, 1(1), 12-21.
[4] Upadhayaya, S. (2025). Topology optimization of a ladder-frame chassis model for heavy commercial vehicles. *Theseus Applied Science Reports*, 12, 45-58.
[5] Agarwal, G., & Sitembu, S. (2022). Structural analysis of heavy motor vehicle chassis frame and design optimization. *SAGE Journal of Mechanical Engineering*, 42(3), 204-215.
[6] Wang, X., & Zhang, Y. (2025). Research and experimental verification of an efficient subframe lightweighting method integrating SIMP topology. *MDPI Applied Sciences*, 15(15), 8192.
[7] Kumar, R., & Singh, M. (2021). Weight optimization of heavy-duty truck chassis using response surface method. *ScienceDirect Materials Today: Proceedings*, 47, 1045-1052.
[8] SolidWorks Corporation. (2024). SolidWorks Simulation User Guide – Finite Element Analysis and Topology Optimization. *Dassault Systèmes*.
[9] Cavazzuti, M., 4 Baldini, A., Bertocchi, E., & Dumont, N. (2011).High-performance automotive chassis design: a topology optimization approach. *Structural and Multidisciplinary Optimization*, 44(1), 45-57.
[10] Charles, T. R., & Dev, K. J. (2017). Finite element analysis of automotive chassis with various cross-sections under static loading. *Journal of Automobile Engineering*, 231(8), 1112-1124.

International Current Journal of Engineering and Science (ICJES)
Volume 5, Issue 8, pp 13-19, August 2026
ISSN (Online): 2583-7257
Article DOI: https://doi.org/10.47001/ICJES/2026.508002

[11] Zhou, H., & Liu, L. (2024). Topology optimization and model fusion reconstruction of concrete pump truck boom structure. *SAGE Journal of Process Mechanical Engineering*, 238(4), 1300-1311.
[12] Harrison, P., &Cebon, D. (2013). Effects of web and flange thickness on bending stiffness of commercial vehicle chassis frames. *International Journal of Heavy Vehicle Systems*, 20(2), 154-168.
[13] Fadhil, M. H. (2024). Optimizing structural safety and thickness reduction in heavy transport trailer frames. *Journal of Mechanical Structural Analysis*, 19(2), 88-97.
[14] Kim, J., & Lee, S. (2023). Multi-objective topology optimization of commercial vehicle subframes considering fatigue life. *International Journal of Automotive Technology*, 24(3), 411-422.
[15] Chen, Y., & Wang, T. (2021). Lightweight design of heavy trailer components using evolutionary structural optimization. *Thin-Walled Structures*, 162, 107532.
[16] S. Slavov and M. Konsulova-Bakalova, "Optimizing weight of housing elements of two-stage reducer by using the topology management optimization capabilities integrated in SOLIDWORKS: A case study," *Machines*, vol. 7, no. 1, 2019, doi: 10.3390/machines7010009.
[17] Takahashi, R., & Sato, H. (2022). Stiffness evaluation and mass reduction of heavy-duty trailer mainframes via FEA. *Journal of Structural Engineering and Mechanics*, 80(5), 589-601.

Citation of this Article:

Totok Haryanto, Sulardjaka, & Agus Suprihanto. (2026). Lightweight Design of a 90M³ Side Dump Trailer Web Flange Subframe Using Manufacturing-Constrained Topology Optimization. *International Current Journal of Engineering and Science (ICJES)*, 5(8), 13-19. Article DOI: https://doi.org/10.47001/ICJES/2026.508002

****