BMSM
Bulletin of Materials Science & Metallurgy
Periodical Scientific Journal of UCTEA Chamber of Metallurgical and Materials Engineers

Volume 2, Issue 1, 2025, Pages 21-27 Received: 17 September 25 Accepted: 30 October 2025
https://doi.org/10.70342/WFFG7935

Research Article

Optimisation of Machining Fixture Design for Milling Operation of Ductile Iron Coach Suspension Arm Component

Mustafa Burak Terzioğlu*, Bülent Şirin, İsmail Şile

DÖKTAŞ Dökümcülük Ticaret ve Sanayi A.Ş.

*Corresponding author's email: mustafaburak.terzioglu@doktas.com

Abstract

Within the scope of this study, the machining fixture design and optimization studies of a bus rear suspension arm made of spheroidal cast iron were carried out using CAD (computer aided design) and CAE (computer aided engineering) programs. The suspension arm is the safety part that turns the bus rear suspension more comfortable, to which the air bellows, shock absorbers and axle are connected. The machining fixture plays an important role in the machinability of the part. The fixture of the suspension arm part was shaped using topology optimization. The fixture design, which was created and subjected to a DFM (Design for Manufacturability) study, was finalized through structural analysis, taking into account the cutting and clamping forces. The cutting and clamping forces used in the structural analyses were calculated empirically, and the resulting values were optimized using DOE (Design of Experiments) methods. As a result, a fixture design based on engineering-oriented topology optimization was successfully developed in this study. The resulting fixture was found to be approximately 42,45% lighter than those designed using conventional methods, yet it exhibited higher rigidity. Structural analyses confirmed that the fixture provided sufficient resistance to the cutting and clamping forces encountered during machining. Dimensional measurements conducted after the machining process demonstrated that the workpiece met the technical drawing tolerances. In this respect, the optimized fixture design offers an effective solution that enhances both manufacturing accuracy and structural efficiency.

Keywords

Machining Fixture Design, Topology Optimization, Spheroidal Cast Iron, Finite Element Analysis (FEA), Vibration and Rigidity

1. Introduction

In contemporary industrial applications, casting methods continue to hold significant importance in the manufacturing of components with complex geometries and high structural strength. Among these, spheroidal graphite cast irons (EN-GJS grade) are widely preferred in the automotive and heavy vehicle sectors due to their superior properties of ductility, strength, and machinability. In this material class, the transformation of graphite morphology from lamellar to spheroidal form markedly enhances the mechanical properties. To achieve this spheroidization, rare earth elements such as cerium (Ce) and magnesium (Mg) are added to the molten iron bath, resulting in a microstructure that simultaneously provides ductility and strength [1]. Owing to these attributes, spheroidal graphite cast irons play a critical role in applications requiring load-bearing capacity, vibration resistance, and long service life. In this study, the machining fixture design and optimization of a rear suspension arm of a bus, manufactured from EN-GJS-500-7 spheroidal graphite cast iron, were carried out to ensure suitability for milling operations. The suspension arm, serving as the component to which air bellows, shock absorbers, and axle elements are attached, directly affects comfort and safety in the rear suspension system of the vehicle. Due to its complex surface geometry, limited clamping regions, and exposure to high cutting loads, the safe and accurate machining of such parts in CNC machining centres is only possible through the implementation of a properly designed fixture [2]. Fixture design not only ensures the positioning and clamping of the workpiece but also guarantees rigidity against cutting and clamping forces during machining, which necessitates optimization according to engineering criteria. In this context, during the initial phase of the study, a fixture system suitable for machining the component was modelled in Siemens NX CAD environment using conventional methods. Special attention was given to the regions subject to surface milling operations, clamping and supporting points were defined accordingly. With indicated x1, x2, x3 points as shown in figure 1, datum plane created for locating the part on fixture. After that, with y1, y2 and z1 points alignment has been ensured.

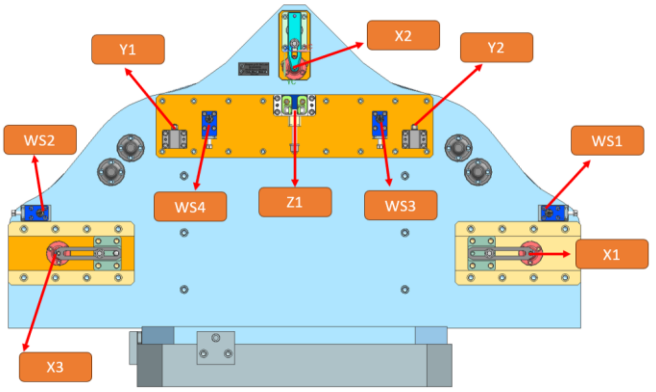

Figure 1. Fixture design in a 3D CAD environment

Subsequently, cutting forces generated during machining were calculated empirically, and their effects on the fixture supports were evaluated through structural analysis using Altair OptiStruct software. The analyses identified deformation and vibration-prone regions of the workpiece, which were reinforced by integrating support elements (work supports), thereby establishing a more rigid clamping system. Following the validation of the fixture design, topology optimization was applied to the same model with the aim of reducing mass while maintaining structural integrity. As a result of optimization studies conducted in the OptiStruct platform, the initial fixture mass of 1,477 kg was reduced by approximately 42,45%, resulting in a final weight of 850 kg. This improvement not only provided significant weight reduction but also enhanced performance. According to structural analysis results, the maximum displacement, which was 88 microns in the initial model, decreased to 68 microns in the topology-optimized fixture, indicating higher rigidity despite the lower weight. In conclusion, a comprehensive fixture design process was conducted within this thesis by integrating CAD and CAE tools with engineering-based parameters. The developed methodology produced a fixture design that is both manufacturable and structurally optimized, enabling reliable clamping that minimizes deformation and vibration-induced quality issues during milling operations. This study contributes to the literature by presenting a systematic methodology applicable to low-rigidity and geometrically complex workpieces, combining manufacturability principles with advanced structural optimization to achieve an efficient and reliable fixture design.

2. Materials and Methods

2.1. Materials

2.1.1. Material of Machining Part (Coach Suspension Arm)

Spheroidal graphite cast iron is a type of cast iron enriched in carbon, in which the graphite structures are present in a nodular form. Owing to its spheroidal (nodular) graphite particles in the microstructure, this material provides superior properties such as high ductility, impact resistance, and strength [1]. This material is defined by a minimum tensile strength of 500 MPa and a minimum elongation of 7%. The Microstructure of EN-GJS-500-7 captured by optical microscope has shown in figure 2.

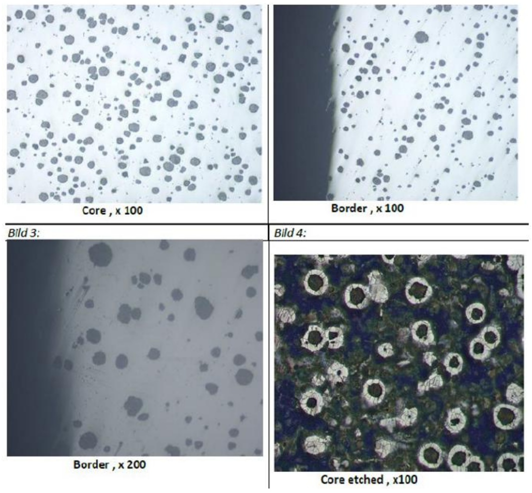

Figure 2. Microstructure of EN-GJS-500-7

2.1.2. Material of Base Plate and Rear Plate

The body elements used in fixture design (the base plate and the rear plate) are the fundamental structures that provide the overall rigidity of the fixture and directly interface with the machine table. Since these components both support the self-weight of the fixture and absorb the loads generated by cutting tools and clamping forces, the selected material must be evaluated in terms of its mechanical properties, weldability, machinability, and cost. For cases where the plates must be welded prior to machining, ST 52-3 provides significant advantages. It can be easily joined using MIG/MAG welding methods without preheating and exhibits low susceptibility to embrittlement in the heat-affected zone [3]. In this study, the fixture bodies were manufactured by plasma cutting, surface finishing, and welding of ST 52-3 plates, with the rear plate being 65 mm thick and the base plate 50 mm thick. This configuration provided durability against cutting forces during milling operations while also offering a rigid mounting surface to the machine table.

2.1.3. Material of Machining Fixture Components

In the machining fixture developed within the scope of this thesis, the blocks containing fixed locators and clamping mechanisms were manufactured from AISI 4140 alloy steel in order to meet both high mechanical strength and machinability requirements. This steel belongs to the group of quenched and tempered steels and is widely used in industry due to its suitability for heat treatment as well as its versatile engineering properties. AISI 4140 is a chromium-molybdenum alloy steel, equivalent to 42CrMo4 in the DIN standard. Its chemical composition typically includes 0.38–0.43% carbon, 0.75–1.0% chromium, 0.15–0.25% molybdenum, and 0.8–1.1% manganese [4]. The combination of these alloying elements provides the material with high tensile strength, hardenability, toughness, and wear resistance. These attributes are particularly critical for fixture elements directly subjected to mechanical loads, such as clamping and guiding during machining operations. Furthermore, tempering reduces brittleness and enhances impact resistance, ensuring reliable performance against sudden loads, impact forces, or vibrations occurring during workpiece placement [5]. In fixtures, fixed locators are employed to accurately determine the position of the workpiece and to prevent positional shifts during machining.

2.2. Method

2.2.1. Fixture Design in CAD Environment

Initially, the fixture design was modelled using Siemens NX CAD software based on the 3-2-1 locating principle. Considering the complex geometry of the workpiece and the force distribution during machining, the fixture was meticulously designed with clearly defined fixed locators, clamping points, and support regions. Fixture locating and alignment points have to be defined for every machined parts. This phenomenon calls as 3-2-1 principle which shown in figure 3.

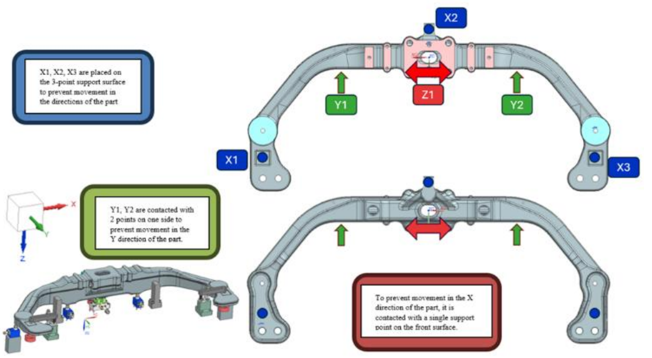

Figure 3. Representation of 3-2-1 principle.

2.2.2. Displacement and Modal Analysis

The finite element method (FEM) is a computational technique used in engineering to obtain approximate solutions of boundary value problems. A boundary value problem is defined as a mathematical problem in which one or more dependent variables must satisfy a differential equation throughout a known domain of independent variables while also fulfilling specified conditions at the boundaries of the domain The fundamental characteristics of finite elements are embedded in the stiffness matrix. For a structural finite element, the stiffness matrix contains the information on geometric and material behaviour that demonstrates the resistance of an element subjected to loads against deformation Figure 4 illustrates a spring element.

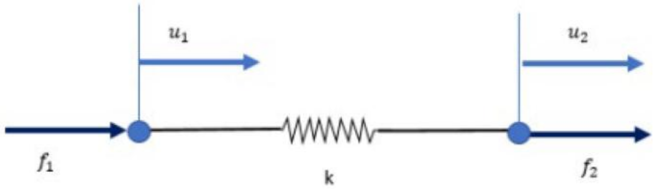

Figure 4. Spring system schematic [6].

In the analysis of systems using FEM, bodies that are in reality composed of infinitely many points are discretized into a finite number of elements. These elements contain different numbers of nodes depending on their type. Calculations are carried out through these nodal points. Essentially, there are three types of elements: one-dimensional, two-dimensional, and three-dimensional [7]. Within these categories, the number of edges, surfaces, and nodal points varies depending on the element type. The choice of element type for a component is determined by considering the geometrical characteristics of the component. A significant number of studies in fixture design have focused on improving machining performance by reducing the adverse effects of deformation, vibration, and stress on the workpiece. The existing literature provides valuable insights into fixture design principles, optimization techniques, and analytical methods for evaluating fixture performance. In this study, finite element analysis (FEA) was conducted to evaluate the deformation and stresses induced on the workpiece under the effect of cutting forces during machining. The workpiece and fixture components were modelled in a CAD environment and integrated into the analysis software. Material properties were defined under the assumption of linear elasticity, while boundary conditions were determined by considering the forces arising during machining. Taking into account the cutting parameters, the load acting on the workpiece was calculated as 5261 N. Under these conditions, the finite element modelling studies were carried out using Altair HyperMesh. Tetrahedral elements were employed in the meshing stage. During the face milling operation, the tool load was applied in the +Z direction at four points on the workpiece surface, corresponding to the regions where the cutting insert was expected to exert the maximum effect. The vector of applied cutting forces for milling operation has been realized as indicated in figure 5.

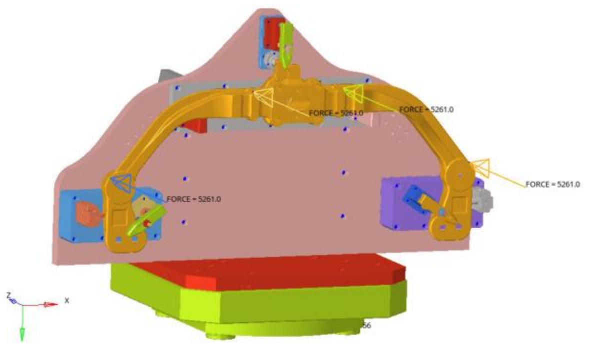

Figure 5. Representation of forces applied during the face milling process.

Modal analysis, also referred to as frequency analysis, is employed to determine the natural frequencies of a structure and the corresponding mode shapes. Vibration modes define how a structure vibrates in the absence of damping and external excitation forces. Although every real structure inherently possesses infinitely many degrees of freedom, it nevertheless exhibits a finite number of discrete vibration modes. Each mode is associated with a specific frequency and a corresponding mode shape, representing the state in which the stiffness forces of the structure are balanced by the inertial forces. In this study, the modal analysis of the fixture and workpiece assembly was performed in Altair OptiStruct using the EIGRL method. The structure was modelled as mounted from the machine-tool interface region, and its operational condition was examined.

2.2.3. Topology Optimization

Optimization is the act, process, or method of making something (such as a design, system, or decision) as flawless, functional, or effective as possible [6]. Topology optimization, on the other hand, involves determining the ideal material distribution within a design. A key factor contributing to the increasing preference for topology optimization methods among designers in recent years is that the optimal structure can be identified at the early stages of the design process. Two commonly used approaches in topology optimization studies are the homogenization method and the material distribution method [8,9]. The definitions related to topology optimization were implemented in Altair HyperMesh. The solver employed was Altair OptiStruct, the first commercial structural optimization code. The optimization process was carried out iteratively, and the final result was obtained after 36 iterations. The optimization outcome is presented as follows. In the legend, points representing values close to 1 indicate regions where material should be retained in the design, whereas elements with values close to 0 were automatically removed using Altair HyperView post-processing software, thereby revealing the ideal material distribution. The outputs of topology optimization analyses at Altair HyperView represented in figure 6.

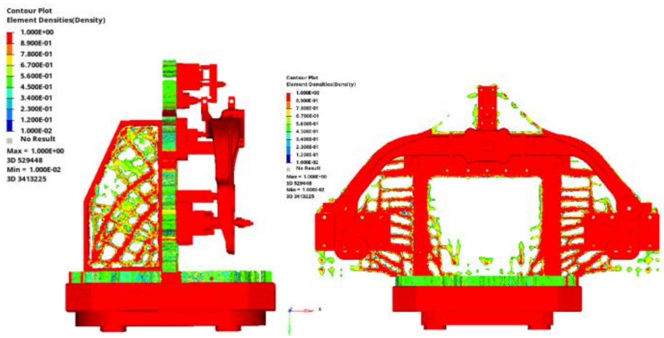

Figure 6. Results of topology optimization

2.2.4. Design for manufacturing

In this study, the fixture geometry outputs obtained through topology optimization were primarily analysed in terms of sheet metal processing, plasma cutting, and 3-axis CNC machining methods. The curvilinear and organic structures resulting from the optimization are, in many cases, too complex to be directly manufactured using conventional subtractive manufacturing techniques [10]. Therefore, planar surfaces and 2.5D features were preserved, particularly in load-bearing plates and support elements, to facilitate cutting, machinability, and ease of clamping. From a DFM (Design for Manufacturability) perspective, these modifications simplified each component to allow manufacturing with a minimum number of machining operations. For instance, instead of curvilinear internal cavities, triangular or rectangular openings were used, which both facilitated material removal and simplified tool path planning. Additionally, regions requiring welding during assembly were replaced with bolted connections, enhancing both disassemblability and maintainability. This approach not only reduced manufacturing time but also minimized operator errors.

2.2.5. Determination of cutting and clamping forces

The cutting forces applied to the workpiece were calculated using empirical formulas, based on the cutting parameters of a Ø160 mm face milling tool and the EN-GJS-500-7 material. The clamping forces were determined based on the maximum forces applicable by the mechanical clamping systems used to secure the workpiece.

3. Results and Discussion

In this study, a machining fixture for a bus rear suspension arm component, manufactured from spheroidal graphite cast iron (EN-GJS-500-7), was designed using Siemens NX CAD (Computer-Aided Design) in accordance with conventional 3-2-1 clamping principles. Fixed support and clamping points were established, and subsequently, milling operations with a Ø160 face mill—including both roughing and finishing passes—were simulated along with the clamping forces on the fixture using Altair OptiStruct CAE (Computer-Aided Engineering) software. Cutting forces generated during material removal were applied over regions corresponding to the cutter edge size, and both modal and structural analyses were performed. The initial design exhibited a displacement of 262 μm, indicating that structural interventions were required in vibration-prone regions of the system. Accordingly, the thickness of the rear plate was increased and mechanical support elements were integrated at the ends of the workpiece. These improvements led to Version 2 of the design, where the displacement was reduced to 82 μm, which was established as the system acceptance criterion which shown in figure 7.

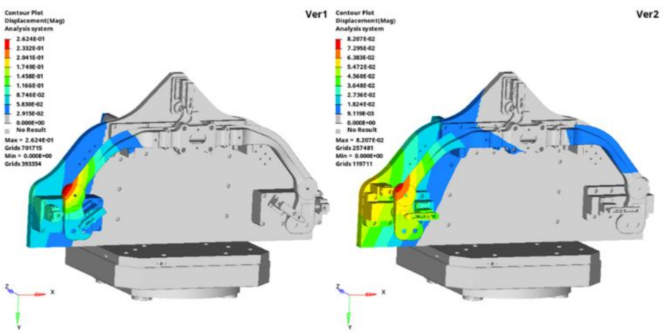

Figure 7. Displacement result comparison of version1 and version2 designs.

Based on this criterion, topology optimization was applied to Version 2, with the objective of reducing the remaining volume of the fixture plates by 50% which it's called as Version 3 in the experiment. The topology optimization outputs were examined, incorporated into the 3D model, and the updated design was subjected to further modal and structural analyses. Also, there were two additional studies have been carried out till the final result at version 6. In the version 4, in order to increase the system rigidity and decrease the displacement values, triangular shaped support plates have been designed and located to the appropriate locations is shown in figure 8. After the finite element analyses for version 4, the displacement values have been decreased to 168 μm and modal frequency has been decreased to 111 Hz.

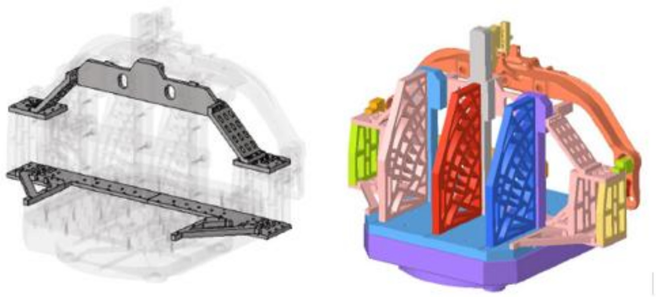

Figure 8. Additional supports on Version 4

At version 5, support plates that shown in figure 9 which have been designed at version 4, have been linked each other with supports to reduce the vibration at construction. With this method, displacement values have been decreased to 92 μm.

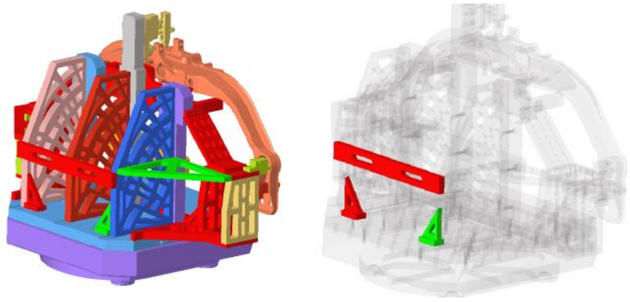

Figure 9. Additional supports on Version 5

Design revisions have been finalised when the displacement was reduced under 82 μm which is the acceptance criteria for the version 2. The final design (version 6) achieved a displacement of 67 μm, and the first natural frequency remained well above the spindle frequency range of the machining center, confirming that the system posed no resonance risk. Consequently, Version 2 was determined to provide excellent rigidity, ensuring surface quality and tool life. In addition to improvements in structural strength and first mode frequency, the topology-optimized design, referred to as Version 6, reduced the fixture weight from 1477 kg to 850 kg. This weight reduction not only decreases manufacturing costs but also enhances the machining accuracy and dynamic stability of dual-pallet CNC machining centres, thereby providing industrial benefits and contributing to the national economy. The version 2 fixture model, obtained from structural analyses, was subjected to topology optimization with a target volume reduction of 50%. During this process, a minimum wall thickness of 20 mm and a maximum of 50 mm were set as constraints, considering manufacturability via plasma cutting. The topology optimization results, generated using Altair Inspire software, were interpreted from an engineering perspective to develop versions 3, 4, 5, and finally version 6 of the design. Each new version underwent modal and strength analyses again, maintaining an iterative improvement cycle. The fixture has been simulated for 4 different machining areas and the results has been presented in Figure 10.

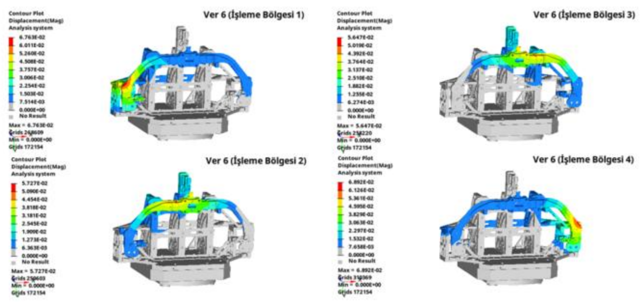

Figure 10. Version6 results of four machining areas

3.1. Weight and Material Consumption

As a result of topology optimization, the fixture weight was measured as 1477 kg in Version 2 and 850 kg in the Version 6 design. This corresponds to an approximate 42,45% reduction in weight. This weight reduction not only provided ergonomic advantages but also had a direct positive impact on the dynamic performance of CNC machines.

Using a lighter fixture, especially in dual-pallet CNC machines, eliminated load imbalance, resulting in increased machining precision. Additionally, the reduced weight helped prevent wear on spindle bearings, aiming to extend the machine's service life.

3.2. Modal Analysis Characterization

When comparing modal analysis results for different versions, the first mode frequency was measured as 82 Hz for Version 2 and 118 Hz for Version 6. The higher natural frequency indicates that the system is more resistant to external vibration sources and poses no resonance risk. This condition positively affects cutting tool life and surface quality, particularly during long-duration machining operations.

3.3. Modal Analysis Characterization

Rigidity, one of the most critical criteria in fixture design, was enhanced with each version. According to the finite element analyses conducted:

- Version 2 fixture: 82 Hz first mode frequency, 82 µm maximum displacement
- Version 6 fixture: 118 Hz first mode frequency, 67 µm maximum displacement

This difference revealed a significant increase in dynamic rigidity between the versions. Additionally, this increase was confirmed to have a direct positive effect on the workpiece surface quality through surface roughness measurements.

3.4. Mass Reduction and Manufacturing Ease

As a result of topology optimization, a mass reduction of up to 42,45% in the fixture weight was achieved, positively impacting the CNC machine performance:

- Version 2 weight: 1477 kg
- Version 6 weight: 850 kg

This weight reduction not only saved material and lowered production costs but also improved the machining balance of the CNC machine. Especially, the pallet imbalance problem occurring in dual-pallet systems was largely eliminated.

3.5. Modular Structure

The modular approach adopted in the design has made this fixture usable for different parts as well. Components that can be assembled and disassembled like Lego pieces have been made reusable for parts of different sizes with minimal modifications. Thus, there was no need for a separate fixture investment for each part, providing flexibility in the production process. This flexibility demonstrates that the fixture design offers not only a technical but also an economical solution. In order to present as image, render has been created at NX CAD which indicated as figure 11.

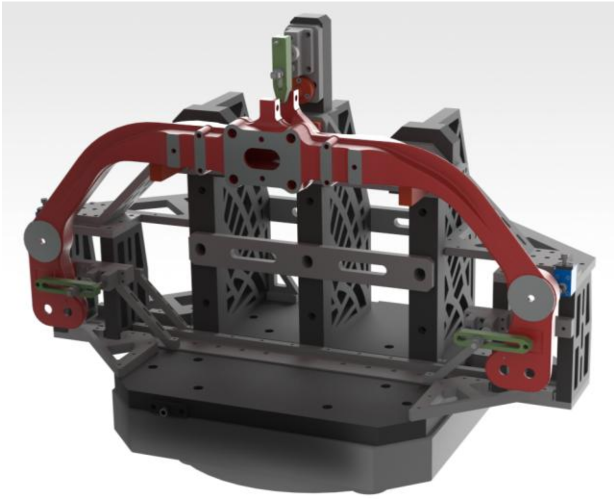

Figure 11. Render images of the final fixture design

3.6. Manufacturing Accuracy and Quality

Measurements taken with the CMM showed that all dimensions of the suspension arm machined with the Version 6 fixture remained within the technical drawing tolerance ranges. This indicates that the fixture was able to prevent surface deformations and enabled first-time-right manufacturing. Additionally, vibration marks on the part after production were significantly reduced compared to the non-optimized fixture, and the cutting tool life increased. This also reveals the fixture's potential to reduce maintenance and tooling costs in the long term.

4. Conclusion

In this study, the fixture structure required for machining a bus rear suspension arm made of ductile cast iron was designed, analysed, and improved using CAD/CAE-based numerical methods and topology optimization. Moving beyond traditional design approaches, loads based on empirical calculations were modelled with finite element methods, the behaviour under these loads was observed, and rigidity was increased.

As a result of the design improvements and optimization efforts:

- Fixture rigidity was increased by 18%,
- Mass was reduced by 42,45%,
- Vibrations and deformations occurring during machining were minimized.

Additionally, thanks to its modular structure, the fixture was made adaptable to workpieces of different sizes, providing manufacturing flexibility. This enabled both a reduction in per-part costs and increased efficiency in CNC machines.

The developed fixture fully complied with technical drawing tolerances and contributed to the principle of "first-time-right" manufacturing. Improvements observed in surface quality, cutting tool life, and machining time demonstrate the industrial applicability and sustainability of the proposed method.

Authors' Contributions

Conceptualization: Mustafa Burak Terzioğlu, Methodology: Mustafa Burak Terzioğlu, Bülent Şirin, İsmail Şile, Investigation: Mustafa Burak Terzioğlu, Bülent Şirin, İsmail Şile, Validation: Mustafa Burak Terzioğlu, Bülent Şirin, İsmail Şile, Formal Analysis: Mustafa Burak Terzioğlu, Resources: Bülent Şirin, İsmail Şile, Mustafa Burak Terzioğlu, Data Curation: Mustafa Burak Terzioğlu, Writing—Original Draft Preparation: Mustafa Burak Terzioğlu, Writing—Review and Editing: Mustafa Burak Terzioğlu, Bülent Şirin, İsmail Şile, Supervision: Bülent Şirin, Project Administration: Mustafa Burak Terzioğlu

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.

Acknowledgement

We would like to thank Döktaş Dökümcülük R&D department for their support.

References

[1] Çetin, A. (2016), Küresel Grafitli Dökme Demir, Ankara, El Kitabı Dökümhane Eğitim Projesi Özel Sayı Serisi.

[2] Hodgson, A., Chen, X., (2008), A Methodology for The Development of Machining Fixtures for Components with Complicated Geometry,International Journal of Computer Integrated Manufacturing.

[3] ŞIK, Aydın. (2007). A Research into the Effect of Additional Electrode Types and Protective Mixture Gases on the Bending Fatigue Life of Steel Material That is Welded with MIG/MAG Welding Method, Gazi University Journal of Science.

[4] Tecer, M.M. (2020) Effects of Various Heat Treatment Procedures on the Toughness of AISI 4140 Low Alloy Steel. The Internal Journal of Materials and Engineering Technology 003,131-149.

[5] Çoban, O. (2025) Heat Treatment of AISI 1040 and AISI 4140 Steels: Microstructure-Mechanical Property Relationships for Normalization, Spheroidization and Quenching-Tempering. Journal of Innovative Engineering and Natural Science.

[6] Hutton, D.V. (2004) Fundamental of Finite Element Analysis. McGraw Hill Companies, New York.

[7] Martins, J.A., Kövesdy, I., (2012), Overview in the Application of FEM in Mining and the Study of Case: Stress Analysis in Pulleys of Stacker-Reclaimers: FEM vs. Analytical, Applications in Mechanical Engineering. InTech.

[8]- Bendsøe, O.S., (1999), Material interpolation schemes in topology optimization, Archive of Applied Mechanics 69, 635-654 .

[9]- Bendsøe, M.P., Kikuchi N., (1988), Genarating Optimal Topologies in Structural Design Using a Homogenization Method, Computer Methods in Applied Mechanics and Engineering 71, 197-224, North Holland.

[10]- Kerbrat, O., Mognol, P., Hascoet, J-Y., (2011), A New DFM Approach to Combine Machining and Additive Manufacturing, Institut de Recherche en Communications et Cybernetique de Nantes.