CAD modelling of front control lower arm using Creo design software

Ravi [1], Ghanshyam Dhanera [2], Purushottam Sahu,[3]Adity Sharma[4]

[1]*Research Scholar,*
[2]*Ghanshyam Dhanera, Professor, Dept. of Mechanical Engineering, BM College, MP, Indore*
[2]*Purushottam Sahu, Professor, Dept. of Mechanical Engineering, BM College, MP, Indore*

---***---

Abstract - *This study explores the application of additive manufacturing optimization techniques to achieve this weight reduction. Specifically, we investigate the use of lattice structures and topology optimization methods. The Finite Element Analysis (FEA) identifies critical areas with significant stresses and deformation. By employing lattice structure technology and topology optimization, we can significantly reduce the weight of the lower control arm.*

Our findings indicate that, among various methods considered, the topology optimization approach is the most effective for minimizing the weight of the front lower control arm while maintaining its structural integrity.

Key Words: Topology optimization, front control lower arm

1.INTRODUCTION

Additive Manufacturing

Industries such as aerospace, automotive, and healthcare are increasingly adopting additive manufacturing due to its versatility and efficiency. This technology is being applied in an expanding range of scenarios. In the next paragraph, we will delve deeper into the specific advantages of additive manufacturing compared to conventional production techniques.

1.1 Topological Optimization

Topological optimization is a computational approach used to optimize the material layout within a given design space, subject to specific constraints and performance requirements. This method aims to achieve the best possible structural performance by redistributing material to areas where it is most needed while removing it from areas that are less critical.

By applying topological optimization, designers can create lightweight, efficient structures that maintain or even enhance their strength and durability. This technique is particularly valuable in industries like aerospace, automotive, and civil engineering, where weight reduction is crucial for performance and efficiency. Topological optimization leverages advanced algorithms and finite element analysis (FEA) to identify optimal material distribution, resulting in innovative designs that are often unattainable through traditional design methods.

1.2 Finite Element Method

The Finite Element Method (FEM) is a computational approach used to address intricate engineering challenges by decomposing a complex system into smaller, manageable elements known as finite elements. These elements are connected at specific points called nodes, forming a mesh that represents the entire system. FEM is instrumental in analyzing various physical phenomena, including structural mechanics, heat transfer, and fluid dynamics.

Using FEM, engineers can simulate and predict how a design will behave under real-world conditions such as forces, vibrations, and thermal effects. This method offers detailed insights into stress distribution, deformation, and potential failure points, which are crucial for optimizing designs to enhance performance and safety. FEM is extensively utilized in sectors such as aerospace, automotive, civil engineering, and biomechanics.

General Procedure of the Finite Element Method

The Finite Element Method follows a structured approach to solving complex problems, typically involving the following steps:

1. Problem Definition and Modeling**

Identify the specific physical problem, such as structural stress analysis, thermal distribution, or fluid flow.

- Develop a geometric model that represents the problem's physical domain.

2. Discretization of the Domain

- Break down the geometric model into smaller, finite elements, creating a mesh. These elements can vary in shape, such as triangles, quadrilaterals, or tetrahedrons.

- Define nodes at the element corners and edges where calculations for field variables will occur.

3. Selection of Element Type

- Choose the appropriate type of elements based on the problem's characteristics and the required precision. Common choices include 1D (bars), 2D (triangles, quadrilaterals), and 3D (tetrahedrons, hexahedrons) elements.

4. Derivation of Element Equations**

Formulate the governing equations for each element, usually based on principles like equilibrium, conservation laws, or material properties.

Construct the element stiffness matrix, load vector, and other necessary attributes.

5. Assembly of the Global System

- Integrate the individual element equations into a global system representing the entire problem domain, combining all the element stiffness matrices and load vectors.

6. Application of Boundary Conditions

Apply the boundary conditions to the global system to simulate the physical constraints of the problem, such as fixed supports, prescribed displacements, or applied loads.

7. Solution of the System Equations

- Solve the global system using numerical methods to obtain the values of field variables at the nodes. Common methods include direct solvers like Gaussian elimination or iterative solvers such as the conjugate gradient method.

8. Post-Processing of Results

- Analyze and visualize the results, often by generating plots of deformed shapes, stress distributions, temperature fields, etc.

Perform further calculations if needed, such as deriving secondary quantities like strain or flux.

9. Verification and Validation

- Verify the results for convergence and accuracy.

- Validate the model by comparing FEM results with experimental data or analytical solutions to ensure its reliability.

Applications of the Finite Element Method

FEM is a versatile computational tool employed in various engineering and applied science fields. Key applications include:

1. Structural Analysis

Civil Engineering: Design and analysis of infrastructure such as buildings, bridges, and dams. FEM helps assess stress, strain, and displacement under different loads.

-Mechanical Engineering*: Analysis of components like beams, frames, and shells, crucial in designing machinery, vehicles, and aerospace structures.

2.Thermal Analysis

Heat Transfer: FEM models conduction, convection, and radiation heat transfer in solids and fluids, applied in electronics cooling, heat exchangers, and insulation design.

- Thermal Stress*: Analyzing stresses and deformations due to temperature changes, essential in designing turbines, engines, and reactors.

3. Fluid Dynamics

Computational Fluid Dynamics (CFD)*: FEM simulates fluid flow and heat transfer in fields like aerodynamics, hydrodynamics, and process engineering.

- Multiphase Flows*: Studying interactions between different fluid phases, such as in oil and gas extraction, chemical processing, and environmental engineering.

4.Electromagnetic Analysis

Electromagnetic Fields: FEM assists in designing and analyzing electric and magnetic fields in devices like motors, transformers, and sensors.

- Wave Propagation*: Used in telecommunications, radar, and microwave engineering to study electromagnetic wave behavior.

5.Acoustics

- Noise Control*: Applied in designing systems for reducing noise in automotive, aerospace, and architectural applications.

- *Vibration Analysis*: Evaluating mechanical system vibrations to prevent resonance and ensure structural integrity.

6. Biomechanics

Medical Implants: FEM is used to design and analyze implants such as hip joints and dental prosthetics, ensuring their strength and biocompatibility.

- Biomechanical Analysis*: Studying the mechanical properties of biological tissues like bones, muscles, and organs.

7. Manufacturing Processes

Metal Forming*: Simulating processes like forging, stamping, and rolling to optimize material flow and minimize defects.

Additive Manufacturing: Analyzing layer-by-layer material addition to predict stresses, deformations, and thermal behavior.

8. Geotechnical Engineering

Soil-Structure Interaction*: Analyzing foundations, tunnels, and retaining structures to ensure stability and safety.

Seepage Analysis*: Studying groundwater flow and its effects on structures and environmental systems.

9. Aerospace Engineering

Aircraft Design*: Analyzing airframe structures, wings, and fuselage to ensure safety and performance under aerodynamic loads.

Spacecraft Components Studying thermal and mechanical stresses in satellite components and space structures.

10. Energy Sector

Wind Turbines*: Structural and fluid dynamics analysis of wind turbine blades and towers.

Nuclear Reactors*: Thermal and structural analysis of reactor components to ensure safe operation under extreme conditions.

11. Automotive Industry

Crash Simulation: Evaluating vehicle crashworthiness and occupant safety during collisions.

Engine Components: Thermal and structural analysis of engine parts to improve performance and longevity.

FEM's accuracy and adaptability make it a crucial tool for the design, analysis, and optimization of complex systems in these diverse fields.

LITERATURE REVIEW

Dr. J.Mahishi [1] this article is a part of a comprehensive study that investigates various factors leading to material failure, with a specific focus on mechanical failures. All instances of failure can be attributed to human error. The three most prevalent types of human error are errors of comprehension, errors of action, and deliberate errors. Finite element analysis is a highly efficient technique for analysing mechanical structures, as it scrutinises the entire structure as a multitude of segments. The status of each component is determined by applying mathematical solutions, which are then combined to determine the overall condition of the structure.

Kang, B., Sin, H.-C., & Kim, J. (2007) [2] Fracture mechanics aims to establish a mathematical framework for analysing failures caused by cracks and fractures. The relationship between the size of a defect, the onset of a crack, the level of stress applied, and the material's brittleness is established. The aforementioned formula can be utilised to calculate the maximum permissible fault size, the necessary breaking force, the load exerted on the component during failure, the materials utilised, and the design's overall quality. This tool can be utilised to assess the magnitude of a defect and the duration it takes for the substance to degrade.

Nawar A. Al-Asady [3] The objective of Finite Element Analysis (FEA) is to determine a distinct solution for every constituent of a system through the application of mathematical techniques to elucidate intricate engineering problems. Finite Element Analysis (FEA) involves the mathematical estimation of the mechanical and physical behaviour of individual components. The present study involves an analysis of the fundamental Finite Element

Analysis (FEA) of a truss, which is a type of rod.

Sritharan. G, [4] The objective of the study was to assess the accuracy of a vehicle suspension component model by comparing it with experimental data. This component is an undercarriage arm designed for a vehicle equipped with a 2000cc engine. The component's critical point, loading conditions, and projected lifespan were established through a series of stress, strain, and wear tests. The strain distribution resulting from the testing process was determined to be in agreement with the one computed through Finite Element Analysis (FEA), despite the complexity of the component's shape. The information obtained from the road test was notably distinct from the findings of the FEA. To properly cite a source, you need to include the author's name, the title of the work, the date of The article titled

"Crashworthiness, Simulation of Lower Control Arm" by Miguel was published in the SAE Journal in April 2005. The objective of the Finite Element Analysis (FEA) was to pinpoint the crucial areas of the component. The model was created using CATIA software, and subsequent strain and fatigue analyses were conducted using MSC Nastran and MSC Fatigue.

IiM. M. Noor and M. M. Rahman [5] In order to fully leverage the potential of FEA analysis, it is imperative to employ factual data to authenticate its precision and soundness in the times ahead. The researchers identified optimal locations for strain sensors by analysing strain analysis data that was verified to be accurate by the FE model. It is recommended to collect experimental data for every individual component. The frequency of injuries caused by fatigue was moderate per cycle, however, the overall number of cycles more than made up for it.

4.2 FEA Analysis Steps

The FEA structural analysis steps are described below:

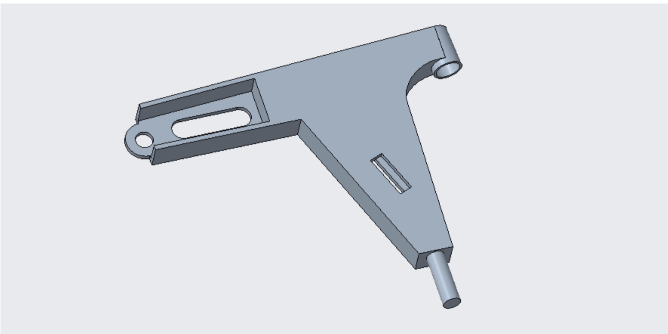
4.2.1 CAD Modelling of front control lower arm

The CAD model of the front control lower arm was generated utilizing the sketch, extrude, and pattern functionalities of the Creo design software. The design of the robotic limb was based on existing literature. [30].

Figure 4.1: CAD model of front control lower arm (generic design)

The CAD model of lattice design of front control lower is developed in Creo as shown in figure 4.2 below. The model is developed with lattice design

1



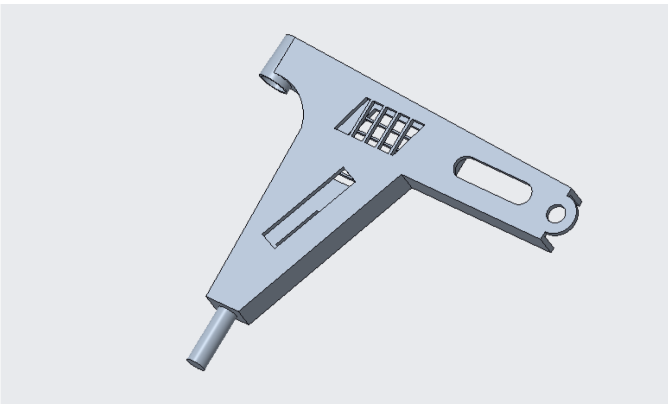


figure 4.2 below. The model is developed with lattice design

Loads and Boundary Conditions

Figure 4.10 depicts the loads that are applied to the front control arm, as well as the boundary criteria that must be satisfied. A cylindrical region is outfitted with a fixed support, and a nodes chosen to take a point load with a magnitude of three thousand newtons.

Figure 4.3: Loads and boundary condition on front control lower arm

4.2.5 Solver settings

The FEA modelling was completed using the sparse matrix approach. The global stiffness matrix is the sum of the matrices for each individual component. Results at nodes are often obtained using matrix operations like inversions and multiplications, whereas results along the length of an element's edge are typically obtained via interpolation.

Generic Design Results of Front Control Lower Arm

The utmost stress that can be measured at the point where the burden is applied is 229.89 MPa. The yellow zone represents the location with the greatest level of tension. The presence of a dark blue hue on the opposing sides of the lower front control arm indicates that the tension levels there are comparable.

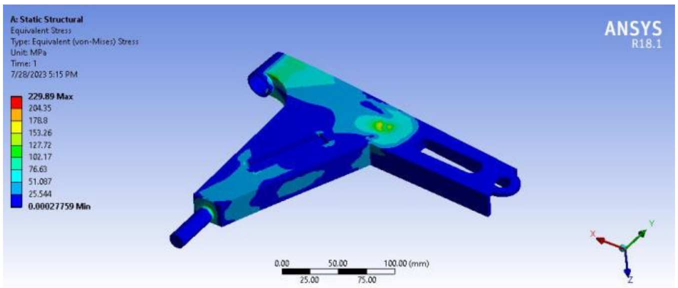


Figure 5.2 depicts the analogous tension observed in the general front control mechanism 'slower control arm

Table 5.1: Comparison table for front control lower arm

Design Type	Equivalent stress(Mpa)	Deformation(mm)	Mass (Kg)
Generic	229.89	.5419	1.7586
Lattice structure 1	263.32	.589	1.6614
Lattice structure 2	260.68	.673	1.6143
Topological optimization	157.44	.2447	.95211

The deformation comparison plot is shown for different designs of front control lower arm. The maximum deformation plot is observed for lattice structure 2 and minimum deformation is observed for topological optimized design of front control lower arm. The minimum deformation obtained for topologically optimized is .2447mm.

Table 5.1: Comparison table for front control lower arm

Design Type	Equivalent stress(Mpa)	Deformation(mm)	Mass (Kg)
Generic	229.89	.5419	1.7586
Lattice structure 1	263.32	.589	1.6614
Lattice structure 2	260.68	.673	1.6143
Topological optimization	157.44	.2447	.95211

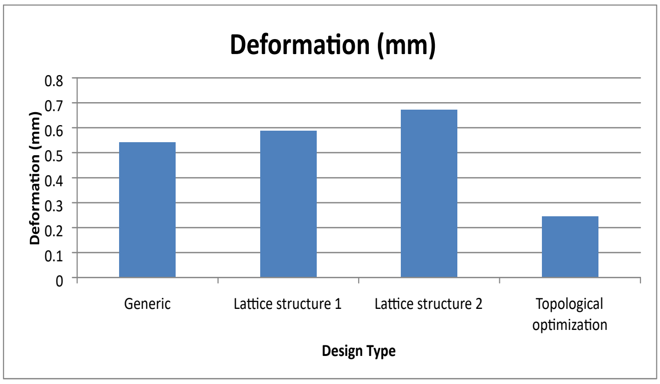


Figure 5.3: Deformation comparison plot

Conclusion

The structural analysis is conducted on the front control lower arm using FEA. The critical regions of high stresses and deformation are identified from FEA results. Significant mass reduction of front control lower arm is achieved using lattice structure technique and topology optimization technique. The detailed findings are:

Using lattice structure 1, the mass reduction of front control lower arm obtained is 5.52% as compared to generic design.

Using lattice structure 2, the mass reduction of front control lower arm obtained is 8.2% as compared to generic design.

1.The topology optimization technique has been able to reduce the mass of front control lower arm by 45.85%.

2.Using lattice structure 1, the deformation of front control lower arm obtained is 8.14% higher as compared to generic design.

REFERENCES

1. D. Kornack and P. Rakic, "Cell Proliferation without Neurogenesis in Adult Primate Neocortex," Science, vol. 294, Dec. 2001, pp. 2127-2130, doi:10.1126/science.1065467.
2. M. Young, The Technical Writer's Handbook. Mill Valley, CA: University Science, 1989.
3. R. Nicole, "Title of paper with only first word capitalized," J. Name Stand. Abbrev., in press.
4. K. Elissa, "Title of paper if known," unpublished.
5. IiM. M. Noor and M. M. Rahman, Fatigue Life Prediction of Lower Suspension Arm Using strain-life approach, © Euro Journals Publishing, Inc. 2009.
6. Roy, R. K. (2001). Design of Experiments Using the Taguchi Approach. New York, New York, United States of America: John Wiley & Sons, Inc.
7. Jong-kyu Kim, Seung Kyu Kim, Hwan-Jung Son, Kwon-Hee Lee, Young-Chul Park,
8. "Structural design method of a control arm with consideration of strength", ACACOS'10 Proceedings of the 9th WSEAS international conference on applied computer and applied computational science, 2010, pp 149-152.
9. Kwon Hee Lee, Xue Guan Song, Han Seok Park, Young Chul Park, "Lower control arm optimization application of response surface model and kriging model to ADI", CSO '09 proceedings of the 2009 international joint conference on computational sciences and optimization ,Vol.2, 2009, pp 1013-1016.