International Journal of Innovation Studies

DESIGN OPTIMIZATION AND RIGID BODY DYNAMICS OF A ROBOT ARM USING ADVANCED SIMULATION TOOLS

Mohammed Rehan Khan[1], Dr. Vinay Kumar [2], Prof. Anuj Muley[3]
[1]Dept. of Mechanical Engineering & TGPCET
[2]Dept. of Mechanical Engineering & TGPCET
[3]Dept. of Mechanical Engineering & TGPCET

Abstract - Robotic arm design and optimisation are essential for improving performance, precision, and dependability in a variety of applications, including automation, medical robotics, and manufacturing. The objective of this project is to use sophisticated simulation tools, namely SolidWorks and ANSYS, to optimise the design and analyse the rigid body dynamics of a robotic arm. In order to obtain optimal performance while minimising wear, energy consumption, and mechanical failures, the study focusses on enhancing the mechanical design by optimising important factors such joint configurations, link lengths, and material qualities. While ANSYS is used to do thorough structural and dynamic simulations, evaluate stress, strain, and deformation, and examine the arm's behaviour under various loading circumstances, SolidWorks is used for 3D modelling. Through simulations, the dynamic performance—which includes the arm's motion, stability, and reaction to external forces—is examined, enabling the discovery of crucial design elements. The results of this investigation offer valuable perspectives for enhancing the design of the robotic arm, guaranteeing increased effectiveness, durability, and accuracy. The findings pave the way for the deployment of robotic systems in a variety of applications by advancing more dependable and effective robotic systems.

Key Words: Robot arm, FEA, CAD, Optimization, Solid works, Ansys Workbench, Structural Analysis

1.INTRODUCTION

Robotic arms are at the forefront of automation and precision jobs in a variety of industries, including manufacturing, research, medical applications, and aerospace, in the quickly developing field of robotics. Optimising the design and functionality of robotic arms has become crucial due to the growing need for more accurate, productive, and adaptable robotic systems. These systems are essential for automating procedures like welding, packing, assembly, and surgery because of their unparalleled precision in carrying out repetitive activities. However, creating a robotic arm that satisfies these demanding performance standards requires resolving a number of intricate issues with its dynamics, kinematics, and mechanical structure. The capacity to execute precise movements with high strength, durability, and low energy consumption is the foundation of robotic arm design. This entails optimising a number of design aspects, such as the actuators employed, joint configurations, link geometry, and material selection. Furthermore, dynamic performance is essential since robotic arms need to be able to react to changing loads, outside disruptions, and quickly shifting operational circumstances. Any of these problems could result in catastrophic failures, decreased performance, or mechanical wear. Therefore, to guarantee that a robotic arm operates dependably and effectively over time, design optimisation and rigid body dynamics analysis are essential. Engineers and designers are increasingly using simulation tools to model and assess robotic arm systems in a virtual environment prior to actual installation in order to address these issues. By reducing the expense and duration of physical testing, these simulation tools enable the optimisation of robotic arm designs and the assessment of their dynamic and structural performance under realistic circumstances. The industry makes extensive use of SolidWorks, a potent computer-aided design (CAD) program, to produce 3D models and robotic system prototypes. SolidWorks is a crucial tool for modelling robotic arms because it makes it possible to create intricate mechanical structures and replicates their behaviour within the program. Designers may simply alter geometry and experiment with various design iterations because to the platform's user-friendly interface and parametric model creation features. Following the creation of the geometric model in SolidWorks, comprehensive analysis of the structural and dynamic performance of the robotic arm is carried out using ANSYS, a top simulation program. For doing finite element analysis (FEA), rigid body dynamics simulations, and multibody dynamics (MBD) simulations—all essential for assessing the arm's behaviour under various operating conditions—ANSYS offers a range of robust tools. Engineers may examine forces, stresses, and deformations, simulate the arm's motion, and make sure the design can support actual operating loads by utilising ANSYS. The process of methodically enhancing the robotic arm's geometry and material characteristics to maximise performance while reducing resource consumption—such as energy, materials, and weight—is known as design optimisation.

2. Objective

The primary objective of this project is to design, optimize, and analyse a robotic arm using advanced simulation tools, specifically SolidWorks and ANSYS, to improve its performance, structural integrity, and dynamic behaviour. The specific objectives of the project are as follows:

- Design a Robotic Arm Model Using SolidWorks
- Perform Structural and Dynamic Analysis Using ANSYS
- Optimize the Design Parameters for Performance Improvement
- Evaluate the Robotic Arm’s Efficiency and Durability
- Integrate Optimization and Simulation Results
- Reduce Prototyping Costs and Development Time
- Provide Recommendations for Real-World Application

3. Methodology

- **CAD Modelling of Robotic Arm**

The geometric design of the robotic arm was developed using CATIA, a comprehensive computer-aided design (CAD) software widely used in high-precision engineering applications. Each link and joint of the arm was modeled as a rigid body using accurate dimensional and kinematic parameters, such as link lengths, joint types, and connection constraints. The use of CATIA allowed for detailed modeling of complex geometries and facilitated the computation of physical properties such as mass, center of gravity, and moment of inertia. The completed 3D model was then exported in a compatible format for integration into dynamic simulation platforms, ensuring seamless transition from design to performance evaluation.

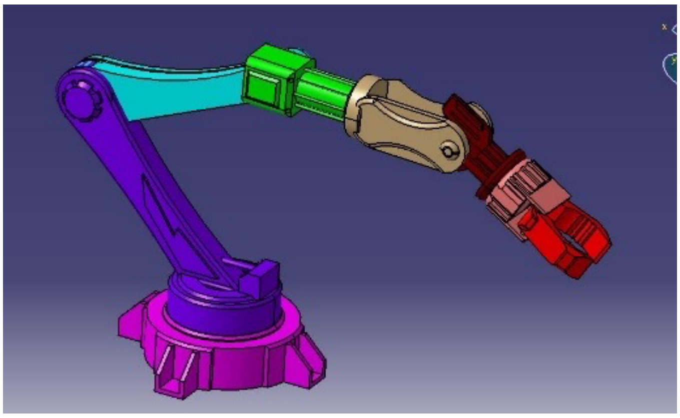

Fig -1: ROBOTIC ARM ASSEMBLY

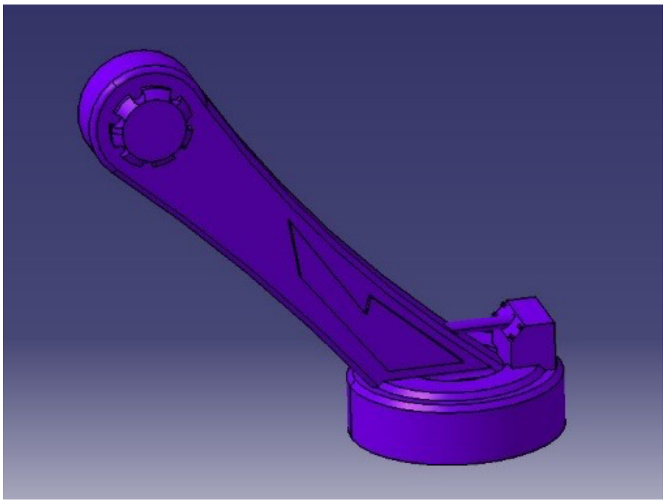

Fig -2: Link 1

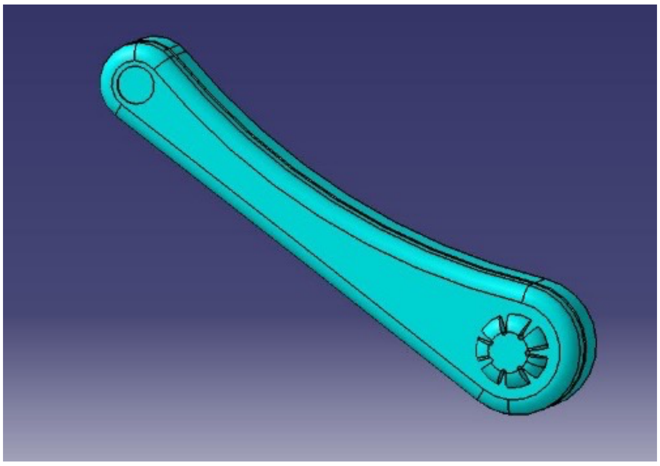

Fig -3: Link 2

- **Analysis of Robotic Arm**

Static structural analysis of the robotic arm was conducted using ANSYS Workbench to evaluate its mechanical integrity under various loading conditions. The 3D CAD model developed in CATIA was imported into ANSYS, where appropriate material properties were assigned to each component, such as aluminum alloys or lightweight composites commonly used in robotic structures. Boundary conditions, including fixed supports at the base and applied loads at the end-effector or joints, were defined to replicate realistic operational scenarios. Mesh refinement was applied to critical regions to ensure accurate stress and deformation predictions. The analysis provided detailed insights into stress distribution, factor of safety, and maximum deflection, enabling the identification of potential failure points and over-designed regions. These results guided structural improvements to optimize strength-to-weight ratio while ensuring that the arm operates safely within its material limits.

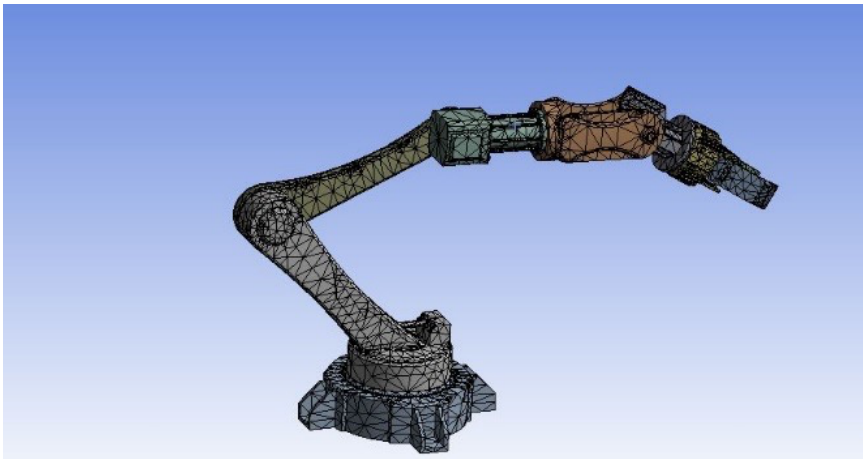

Fig -4: Mesh View

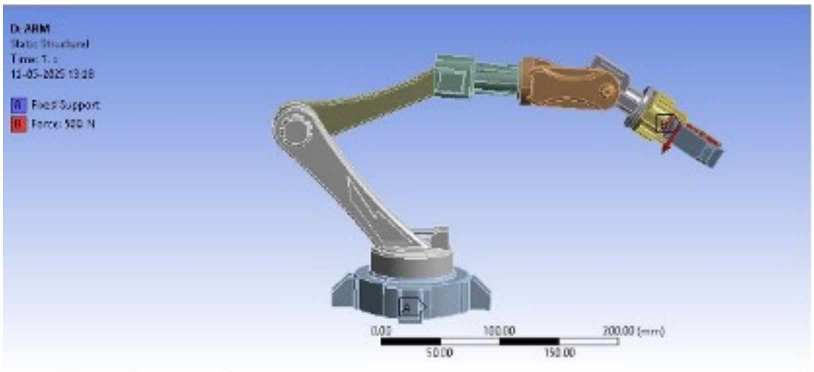

Fig -5: Boundary Condition

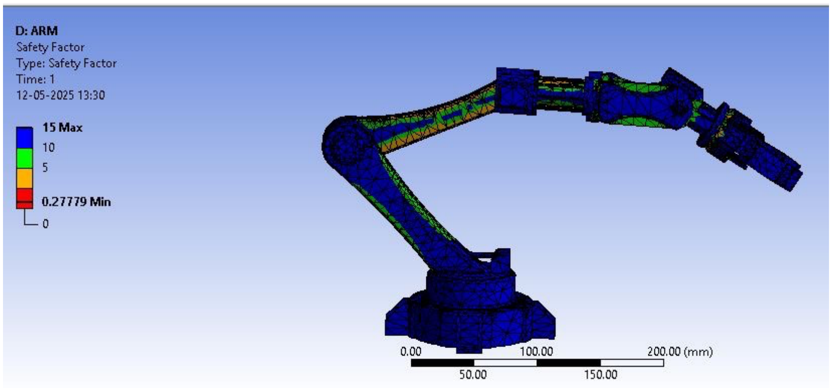

Fig -6: Factor of Safety

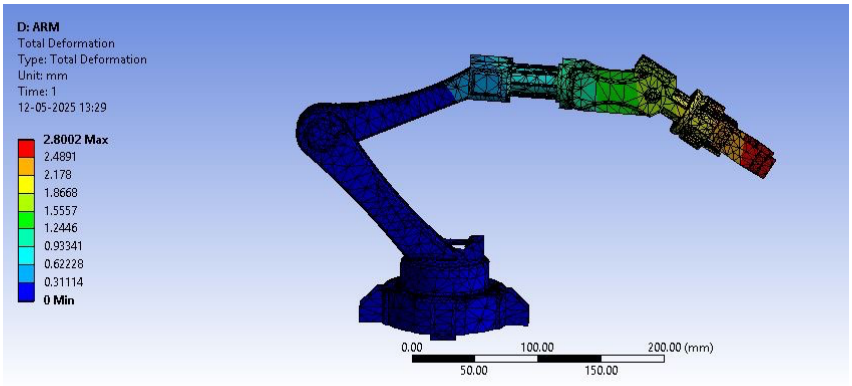

Fig -7: Total Deformation

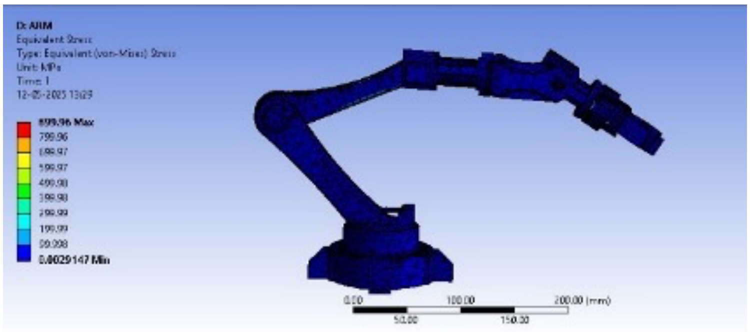

Fig -8: Equivalent Stress

- **Optimization of CAD model According to analysis**

After Doing the analysis on the CAD model of robotic arm, we have found the areas where zero stress occurred that can be optimized by using manual modelling technique using CATIA software. Using optimization technique, link 1 and link 2 is optimized as shown in the below figure without compromising the strength of the arm.

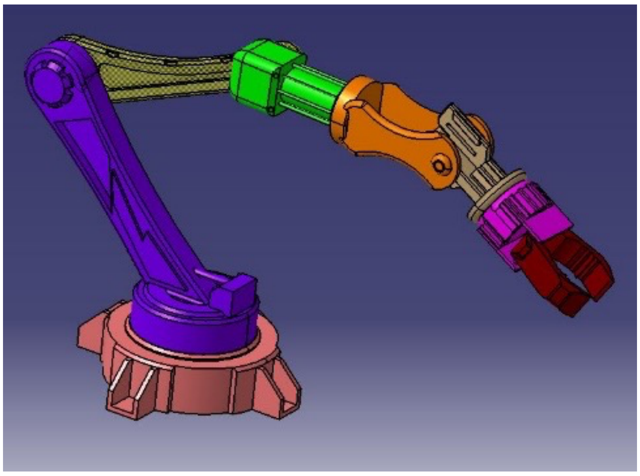

Fig -9: Robotic ARM Assembly_Optimized

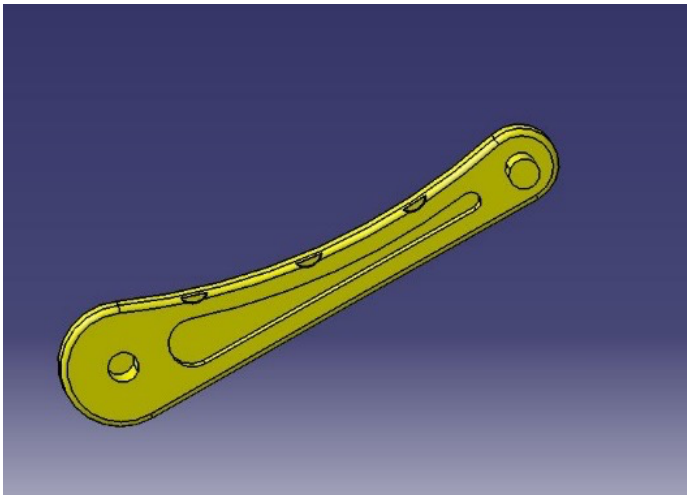

Fig -10: Link_1_Optimized

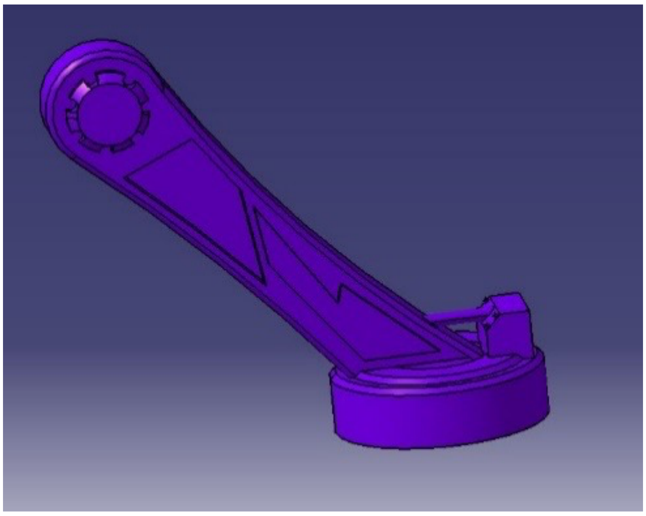

Fig -11: Link_2_ Optimized

- **Analysis of Optimized CAD model**

Static structural analysis is carried out to find out the stresses and total deformation which is occurred in the robotic arm during the loading condition. These analysis gives us the complete insight of the optimize robotic arm.

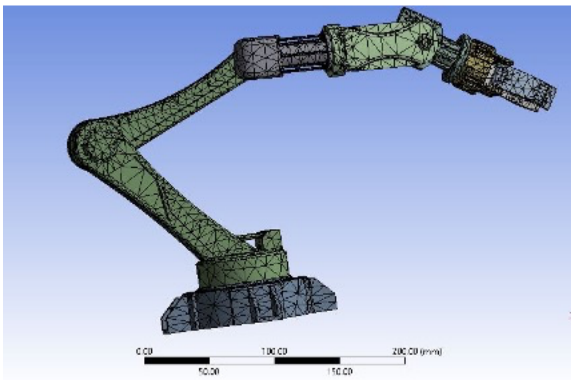

Fig -12: Mesh View Optimized Geometry

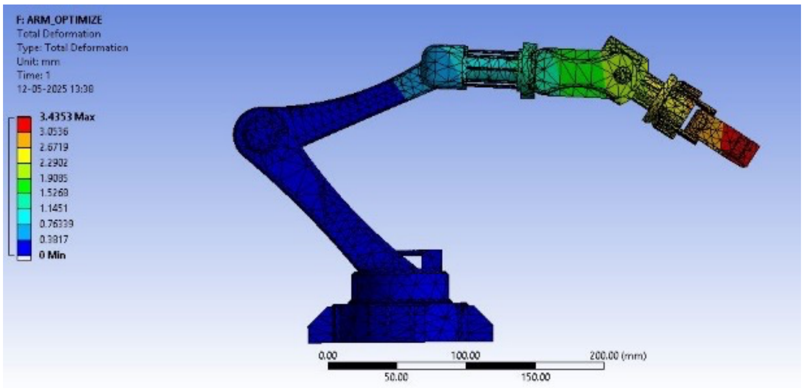

Fig -13: Total Deformation for Optimized Geometry

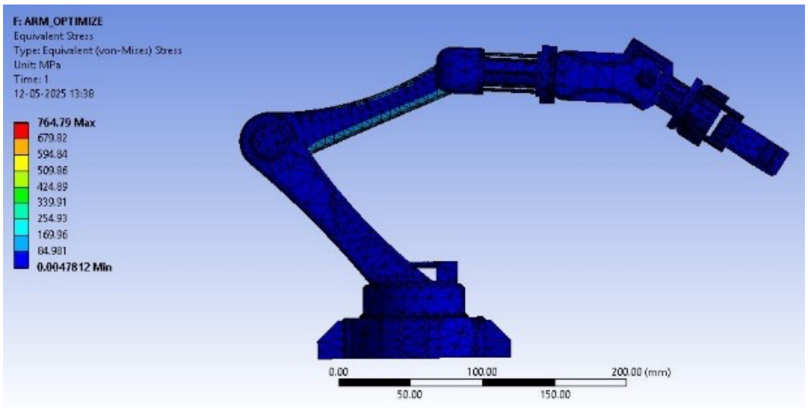

Fig -14: Equivalent Stress for Optimized Model

4. CONCLUSIONS

This study demonstrated a comprehensive approach to the design optimization and static structural analysis of a robotic arm using advanced CAD and simulation tools. The initial design, analyzed in ANSYS Workbench, exhibited a maximum total deformation of 2.8 mm and a peak stress value of 899 MPa under static loading conditions. Following geometric optimization aimed at reducing stress concentrations and improving load distribution, the revised design showed a slightly increased total deformation of 3.43 mm but a significantly reduced maximum stress of 764 MPa. This trade-off is acceptable and beneficial, as the optimized structure not only operates within the allowable stress limits of the material but also enhances safety and durability by minimizing high-stress zones. The results validate the effectiveness of simulation-driven design optimization in improving structural performance while maintaining mechanical efficiency. Future work may involve dynamic and fatigue analysis, as well as real-time control integration, to further refine and validate the robotic arm design..

REFERENCES

1. Kevin Morales, Carlos Hoyos, Jesús M. García “Mechanical Structure Design and Optimization of a Humanoid Robot Arm for Education”, Journal of Autonomous Intelligence (2022) Volume 5 Issue 1.

2. CHERIPALLY NAVEEN, G. BALAJI, Dr. N. JEEVAN KUMAR “DESIGN AND ANALYSIS OF ROBOTIC ARM”, Material Science and Technology, Vol.23 No.02 Feb, 2024.

3. JEEVAN, DR. AMAR NAGESWARA RAO "Modeling and Analysis of Robot ARM using ANSYS", International Journal of Scientific Engineering and Technology Research Volume.04, IssueNo.33, August-2015.
4. Samir Chakravarti, Souvik Sarkar, Suvro Mukherjee, Moinak Biswas, "STATIC AND DYNAMIC ANALYSIS OF ROBOTIC ARM WITH A GRIPPER", International Research Journal of Modernization in Engineering Technology and Science Volume:03/Issue:06/June-2021.
5. Anurag singh, Rashmi Arora, Yashpal Singh Chouhan, "Design and Static Analysis of Robotic Arm using Ansys", International Journal of Recent Technology and Engineering (IJRTE), Volume-9 Issue-1, May 2020.
6. Dodla Mandeep, Mettu Ranjith, Paka Sumanth, Pola Ravi Kumar, Chetla Venu Gopal, Dr K. Siva Prasad, "Design, Analysis and Fabrication of Pick and Place Robotic Arm with Multipurpose", International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 12 Issue III Mar 2024.
7. P Rajesh, Naveen Kumar, K Ganesh, "DESIGN AND ANALYSIS OF ROBOTIC ARM FOR EFFICIENT PICK AND PLACE OPERATIONS", IJCRT | Volume 12, Issue 5 May 2024.
8. Deepak Katal, Amritpal Singh, "Weight reduction of Robotic Arm using topology optimization and Lattice Structure Technique in additive manufacturing", IJCRT | Volume 10, Issue 3 March 2022.
9. Harihara. S, Dr. Vivekananthan.R, Dr. Jamuna.R, Dr. Lavanya.D , Meenakshi.S, "Static and Dynamic Analysis of Articulated Robotic Arm for Domestic Purposes", International Journal of Research Publication and Reviews, Vol 4, no 1, pp 417-421, January 2023.
10. Renish P. Vadhadiya, Prof. B.K. Patel, Chiragsinh Zala, Chintan C. Patel, "Static Analysis of Five Degrees of Freedom Articulated Robot Arm", IJSRD - International Journal for Scientific Research & Development| Vol. 3, Issue 03, 2015.
11. Abhishek Gupta, Purushottam Sahu, Ghanshyam Dhanera , "The lattice structure design enables topology optimization of a robotic arm", International Research Journal of Engineering and Technology (IRJET), Volume: 10 Issue: 06 | Jun 2023..
12. Tauseef Uddin Siddiqui, "CAD MODELING AND FINITE ELEMENT ANALYSIS OF AN INDUSTRIAL ROBOTIC ARM", JETIR January 2019, Volume 6, Issue 1.
13. Abdulhameed Muneam HAMEED and Serdar AY, "A COMPARATIVE STUDY ON THE MATERIAL SELECTION OF A SPOT WELDING ROBOTIC ARM", ACADEMIC JOURNAL OF MANUFACTURING ENGINEERING, VOL.21, ISSUE 1/2023.
14. Vijay Palled and Dr.M.S.Hebbal, "Design, Analysis and Implementation of Robot Arm", International Journal of Advanced Research in Science & Technology (IJARST) Volume 8, Issue 1, August 2020.
15. Mukund Narayan Pandey, Amit Kr Srivastava, Paveen Kumar Maurya, "Design & Analysis of Robotic Arm's Part for Carbon Composite Material", IJARIIT, ISSN: 2454-132X.