Finite Element Analysis and Design Optimisation of Welding robot base using ANSYS

Dr. Fahad Alhajri[1]; Hadyan Ali Alajmi[2]
[1]Specialist Trainer (A), the Public Authority for Applied Education and Training, Ain Shams University, Egypt
[2]Specialist Trainer (B) in Public Authority for Applied Education and Training, the University of Arab Academy for Science, Technology and Transport

Abstract:- The study on Finite Element Analysis and Design Optimization of a welding robot base using ANSYS explores the crucial role of welding robots in enhancing industrial productivity and quality. It addresses the challenges of structural integrity, operational efficiency, and durability under harsh conditions. The research focuses on optimizing the robot base design to withstand high production demands, using ANSYS for detailed modelling and simulation. It reviews previous studies highlighting the importance of vibrational characteristics and structural dynamics in robot design. The methodology involves creating a geometric model, defining material properties, meshing, and applying boundary conditions and loading scenarios. The results indicate that the optimized design significantly improves stress distribution, reduces deformation, and enhances the dynamic response, making the robot base more robust and reliable. The study concludes with recommendations for broader applications in robotic systems to improve safety and productivity in industrial settings.

Keywords:- *Finite Element Analysis, Welding Robot, ANSYS Software, Productivity and Quality, Structural Integrity, Operational Efficiency, Durability.*

I. INTRODUCTION

A robot is a mechanical machine created by humans to perform programmed tasks to make life easier and speed up work completion. The importance of robots has emerged in industry, as they have been utilized to increase production and quality. The robot is characterized by reliability because it is a machine, meaning it is accurate and fast, so it is utilized in welding [1]. A welding robot is one of the best solutions in harsh environments where temperatures and fumes are very high, as manual welding is ineffective in these conditions. [2]

The welding stage is carried out using the robot in two ways: the first is the welding stage, and the second is the analysis stage. In the first, the robot must track and maintain the direction of the tangent and the second requires an operator to evaluate the piece [2]. Factories today are moving to increase the number of welding robots in production lines [2], and this requires continuous improvement and development of this robot, to reduce the problems and risks incurred due to its malfunction, the automobile and shipping industries are examples of applications of welding robot.

A welding robot may require minimal human intervention, which means inaccuracies and variations in the dimensions of the workpiece, thermal distortions and Imperfect edge problems might happen [2]. The base of the welding robot is one of its most important parts, so it must be designed to have a large capacity to withstand high production [1].

The welding robot consisted -in general – of three sliders to position it in any direction, when the welding process begins, these sliders remain constant. [4], all moveable pieces accelerate very slowly because of the extremely smooth alignment before welding. This means that the mechanism's design can employ static approaches to account for the smallest loading increment caused by dynamic forces. The robot's common operating state—that is, the slider positioning—does not result in increased dynamic loading [4].

Therefore, the significance of this paper is comprehensive and has multiple aspects in the field of industrial automation. This study highlights the fundamental challenge of improving the robustness of welding robots in particular, and improving the effectiveness and reliability of robots.

Since welding robots have a vital and widespread role in the operations of various industries [4]. This improvement is crucial because these robots majorly contribute to the efficiency of production lines across different sectors. This research aims to make welding robots more resilient to faults and better overall performance. This is to ensure that the robots are able to carry out all their tasks with sufficient accuracy and reduce the possibility of breakdown, to make these robots more powerful and reliable. This research contributes significantly to the advancement of technology in industrial automation, enhancing overall productivity and production quality in various industries.

Addressing structural integrity and extending the life of the robot: Continuous operation of welding robots leads to corrosion, which may cause failure or damage to one of the robot parts. This research aims to analyse some of these failures of the robot rule. This includes determining the type and loading conditions under which these failures occur. This is crucial, with the aim of preventing downtime and extending the service life of these robots.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24SEP877

Improve the design of the robot base: The welding robot base plays an important role in its work. The robot's base is the essential load-bearing component. It therefore has a vital role in the overall stability and function of the robot. This study involves the design and analysis of new robotic parts using ANSYS, with a focus on improving their efficiency and durability compared to older designs. This leads to an increase in the lifespan of robots in general, and thus contributes to improving industrial profitability.

Use of advanced simulation and analysis tools: The research takes advantage of the capabilities of the ANSYS finite element analysis (FEA) program. This allows detailed and accurate modelling and simulation of the robot base, providing insights into stress distribution, deformation, natural frequencies and more complex and necessary details. This analysis is necessary to validate the design and ensure it meets the required performance standards.

Effective contribution to the field of robotic design: The results of this research have broad implications for the design of robotic systems in many industrial applications. By optimizing the size and enhancing the dynamic and static performance of the welding robot base, the research provides valuable insights and methodologies that can be applied to other.

The key goal of this research is to enhance structural integrity, improve operational efficiency and extend the operational life of welding robot bases. This will be done by applying FEA and using design optimization techniques using ANSYS software. This study seeks to address the challenges resulting from continuous operation of welding robots, in addition to dynamic loading conditions. Therefore, developing a robot base will make it more robust, and perfectly designed to withstand the various pressures and loads that robots face during their operation.

II. LITERATURE REVIEW

In this literature review section on the topic of FEA and optimization of welding robot base design using ANSYS. Current research will be critically studied and examined, and the latest developments in this field will be analysed. This for establishing a scientific basis for the current study. This section delves into various studies that have contributed significantly to our understanding of structural dynamics and optimization of welding robot rules.

In 2010, Liao [5] provide an exploration in welding robots optimisation field. Liao focused on investigation the natural frequencies and mode shapes of the welding robot base. He utilised ANSYS benchmark version 10.0. In his paper Liao investigated the dynamic analysis of the welding robot and its base in particular. This paper’s findings revealed that the upper and tail edges of the base are prone to larger vibrations, which is leading to potential fatigue and damage, with a notable maximum amplitude observed at the 7th natural frequency (39.249 Hz). This study underscores the importance of considering vibrational characteristics in the design of robot bases to prevent structural fatigue.

Complementing Liao's work, Chung et al. [6] also applied ANSYS-10.0 software, in designing and analysing procedures of the heavy-duty industrial robots. The studied robots’ weight reaches several hundreds of kilograms. Their research involved static and dynamic analysis for heavy-duty applications. They also included several comparisons of experimental and Finite Element Method (FEM) results. Researchers highlighted the significance of elastic rigid body dynamics to estimate the robustness of robot parts during robots’ operational motion. Which is a critical aspect in the design of industrial robots. The results of their research determined the frequency of the interaction forces, which reached 6.67 Hz, and the natural frequency was equal to 18 Hz.

In 2015, Varma et al. [7] made a significant contribution to this field by designing a welding robot gun and conducting an analysis using ANSYS benchmark version 15.0. Their research was centred on creating a support structure for the welding robot gun, specifically intended for use in light vehicle door frames. The researchers paid special attention to the functionality of the welding subsystem, given its extensive range of applications. They engineered the welding subsystem to include a support structure that could facilitate rapid and robust movements, a crucial attribute for operating effectively in the dynamic welding environments. The study discussed earlier, which involved modelling the robot's base and assessing its vibration behaviour, further underscores the importance of reliable design and analysis tools such as ANSYS. This modelling approach presented in the previous study can be potentially adapted for the design of support structures like the one considered by Varma et al. for their welding robot gun, ensuring robust and efficient performance in various applications.

In their 2021 study, Niresh et al. [8] aimed to ensure the effective mounting of robots under numerous loading conditions. Authers highlight the importance of avoiding overdesign and under-design of the robot's base frame. The proper design avoid several issues like robot’s vibrations, failures, and decreased robot manufacturing costs. Niresh et al. employed ANSYS software to create a three-dimensional model of the robot platform. They applied relevant boundary conditions. They also analysed von-Mises stress and deformation of robot’s bridge riser. They compared these results with the initial design, to determine new design safety aspect. The study optimized the base design in structural aspect. They created a CAD model that was validated using ANSYS. Ultimately researchers showed the superiority of their new optimized design. This research offers valuable insights for optimizing robot mounting base designs using FEA and design optimization.

Yadav et al. [1] conducted a related study focused on analysing robot bases. Robot bases are basic support structures. They provide stability to industrial robots. In this research, Yadav et al. employed ANSYS 15.0 software, in aim to perform a FEA of the robot's base. As a primary step, researchers created a pre-defined 3D model of the base. Then, they employed pre-defined boundary conditions (as robot's weight and the applied forces). The analysis included

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24SEP877

evaluation of stresses, total deformation, and natural frequency. Model analysis revealed that the natural frequency of the pedestal ranged from 69.536 Hz to 446.36 Hz. This research demonstrated the effectiveness of the ANSYS program in predicting the vibration behaviour of robot bases. This confirm ANSYS software reliability as analysis tool. The modelling approach presented in this study holds promise for improving substrate designs in practical applications.

These previous works collectively highlight the evolving nature of FEA and design optimization in the realm of robotic welding systems. By understanding and addressing the challenges posed by structural dynamics and loading conditions, these studies have significantly contributed to the development of more robust, efficient, and reliable welding robots. This literature review serves as a platform to understand the depth and scope of research in this field, setting the stage for the current study's contribution to the ongoing development of welding robot technology.

Poruba et al.'s study [4] focuses on the force analysis and optimization of emergency stop mechanisms in welding robots, which is extremely crucial for safety and performance. Dynamic simulations were employed to track stress trajectories. Researchers identified peak stress values caused by dynamic loading. The study led to design improvements by reinforcing the lower horizontal rail with more stiffening plates. This is significantly reducing maximum stress levels in the construction. However, it's essential to note that under dynamic loading, stress levels were found to be about 30% higher than those under static loading. This highlighted the need to consider dynamic loading's impact on structural design. These requirements as the exceeding of material's yield point, which could result in plastic deformation or cracks. This study underscored the importance of optimizing emergency stop mechanisms. It is also highlighted the importance of taking account for dynamic loading in structural design for welding robots.

III. METHODOLOGY

A. Development of Geometric Model for the Original Basic Structure

The base structure of the welding robot is designed with a cylindrical shape. To precisely model this structure, SOLIDWORKS, a 3D modelling software, was employed. The detailed geometric model created serves as the foundation for subsequent analyses and was efficiently imported into ANSYS Workbench, a robust FEA software. This importation besides initial setup are captured in Figure 1, showcasing the model ready for further analysis.

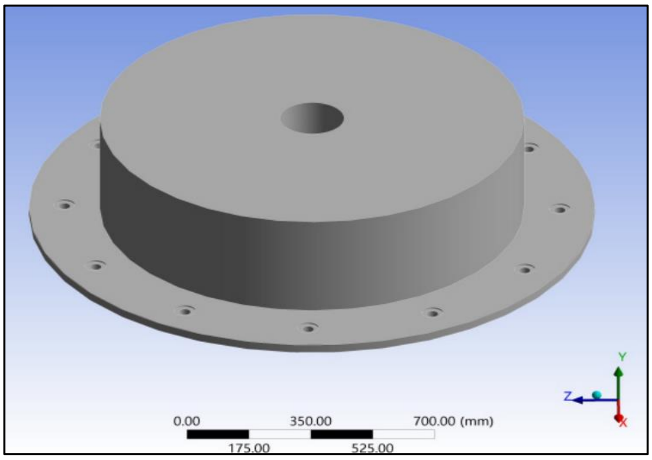

Fig 1: The Geometry of the Welding Robot for the Original Base Structure

B. Specification of Material Properties

Constructed from Q235 steel, the base structure's material is designated for its favourable mechanical properties suitable for rigorous operational demands. The material properties defined for the analysis contain an elastic modulus (E) of 2.1 MPa, which dictates the stiffness of the material under stress. Furthermore, a Poisson's ratio (μ) of 0.3 is applied, reflecting the material's ability to resist deformation in directions perpendicular to the applied force. The density is set at 7,900 kg/m^3, offering the gravitational forces calculations within the model.

C. Meshing of the Model

For the FEA, the model was discretized utilising tetrahedral elements with each element measuring 15 mm. The generated mesh included a total of 69,472 elements besides 120,816 nodes. This setup achieved a mesh skewness value of 0.4, demonstrating a high-quality mesh with elements well-suited for precise simulation results. The meshing process and its high fidelity are represented in Figure 2:

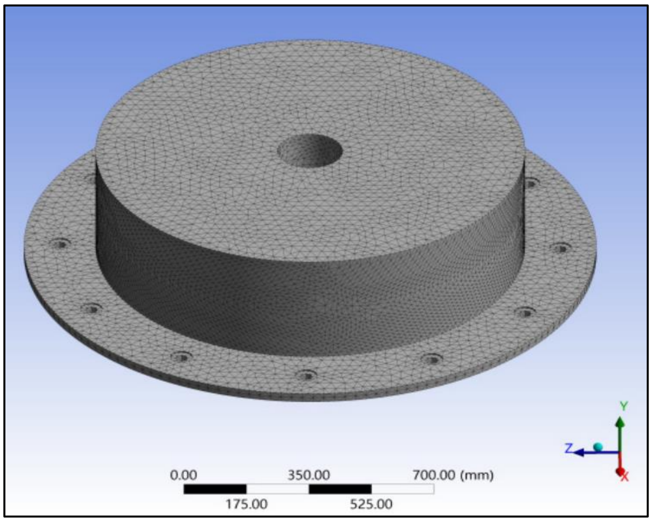

Fig 2: Modelling of Mesh of the Welding Robot for the Original Base Structure

D. Application of Boundary Conditions and Loading Scenarios

The base model incorporates twelve bolt holes on its base plate, which facilitate a secure bolted linking to the ground, ensuring stability during operation. Fixed constraints were applied to the internal cylindrical surfaces of the holes of the bolt on the bottom base plate, effectively eliminating all degrees of freedom associated with rotation and translation.

Mounted atop this base is the robot's main assembly, subjecting the base to a combination of static and dynamic loads. This contains the robot's own weight and operational loads such as inertia generated during movement in addition emergency manoeuvres. Specifically, vertical and horizontal inertia moments and forces are critical during scenarios such as rotational emergency stops and forward tilting. The precise distributions of these loads are illustrated in Figures 3 and 4 correspondingly:

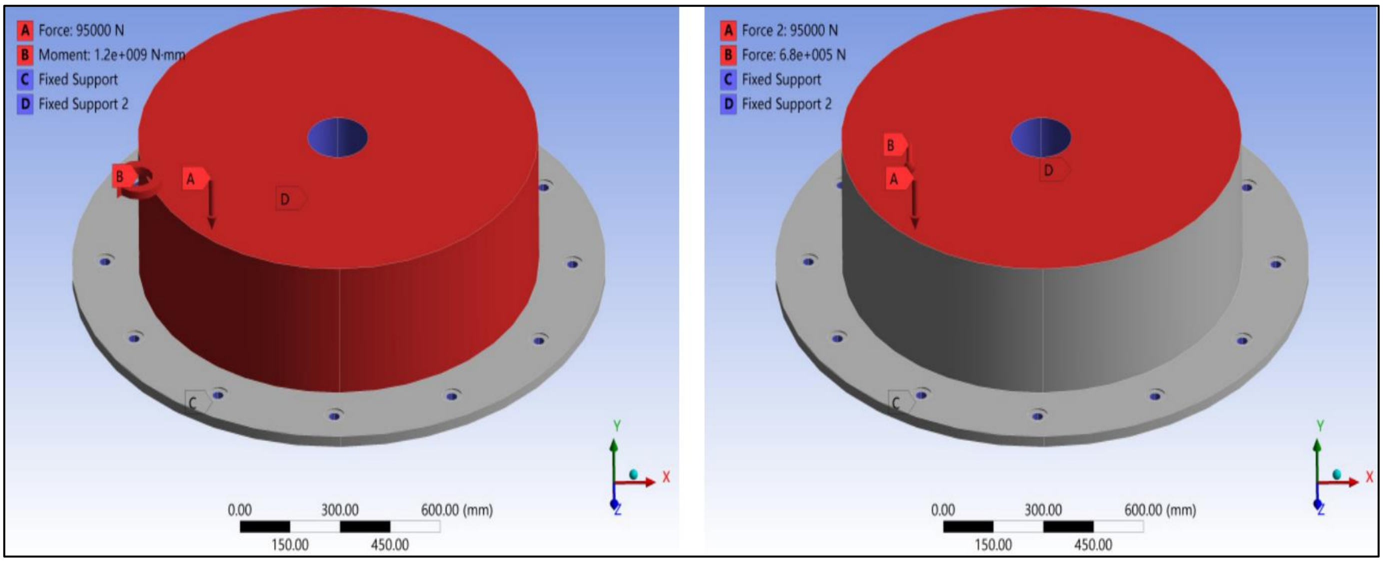

Fig 3: (a) Under the Influence of the Inertia Vertical Moment, (b) Under the Influence of Vertical Force

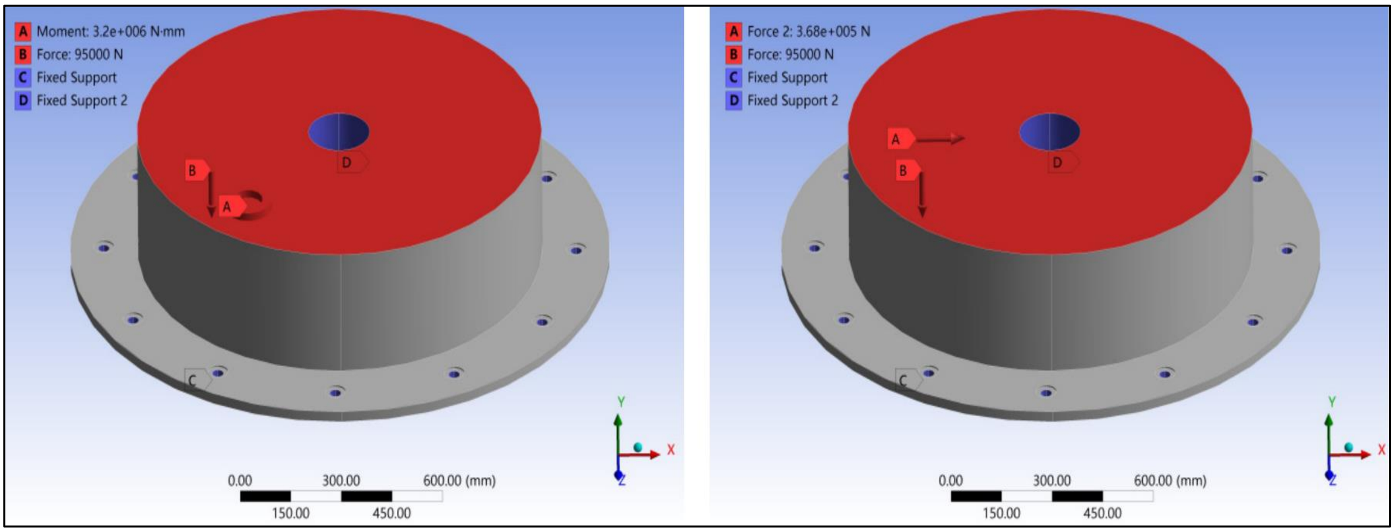

Fig 4:(a) Under the Influence of the Inertia Horizontal Moment (b) Under the Influence of Horizontal Inertial Force

IV. RESULTS AND DISCUSSION

A. Finite Element Analysis (FEA) Results

As the primary support structure for the body of robot, the base must be capable of enduring several types of loads whereas maintaining stable performance. Initially, it is essential that the base meets the necessary strength requirements, meaning that the maximum stress experienced should not exceed the material's allowable stress. Additionally, the base should fulfil the static stiffness criteria by diminishing overall deformation. Beyond these static characteristics, the base also needs to show robust dynamic performance, typically characterized by favourable low-order modal properties of the construction.

The von-Mises stress, the of shape 1st order mode, and total deformation results for the basic base under several loading situations are depicted in Figure 5, Figure 6, Figure 7, and Figure 8. The figures reveal that the highest total deformations and total equivalent stress occur under sheer inertia moments during scenarios such as forward tilting of the robot body besides emergency stopping. The peak von-

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24SEP877

Mises stress reaches 105 MPa at a hole of the bolt on the bottom plate, whereas the maximum value of deformation is measured at 0.0368 mm at a critical load-bearing area on the higher surface. Additionally, the frequency of the base 1st order modal is determined to be 703.95 Hz. With the material yield strength of Q235 steel being 235 MPa in addition the value of safety factor is 1.2, the FEA displays that the maximum value of stress that's experienced by the base is below the acceptable limit of 195 MPa, thus fulfilling the required strength criteria.

The stress distribution across the robot base differs significantly depending on the type of loading condition. Under vertical moments of inertia and forces, the stress is cantered and diminishes outward, representing central concentration. Horizontally, stress is distributed more towards the periphery, particularly under moments of inertia, highlighting the base's response to twisting and bending forces.

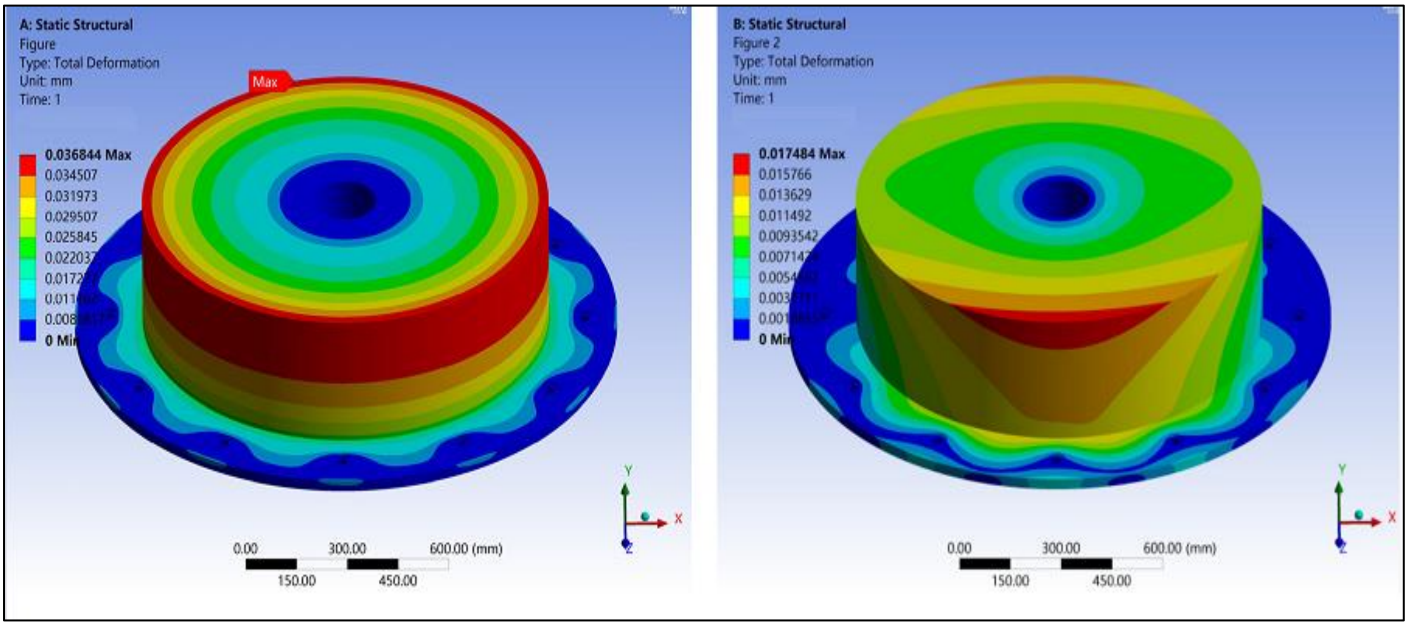

Fig 5: Complete Von-Mises Stress Distribution Across the Base Subjected to Various Operational Settings, (a) Subjected to Inertia Vertical Moment (b) Subjected to Vertical Force.

Deformation patterns on the robot base also reflect the nature of applied loads. Vertical loading conditions result in more centralized deformation, mainly under moments, suggesting significant bending at the centre. In contrast, horizontal loading spreads deformation towards the edges, suggesting the base undergoes flexing and potentially faces higher stress at these points under rotational dynamics.

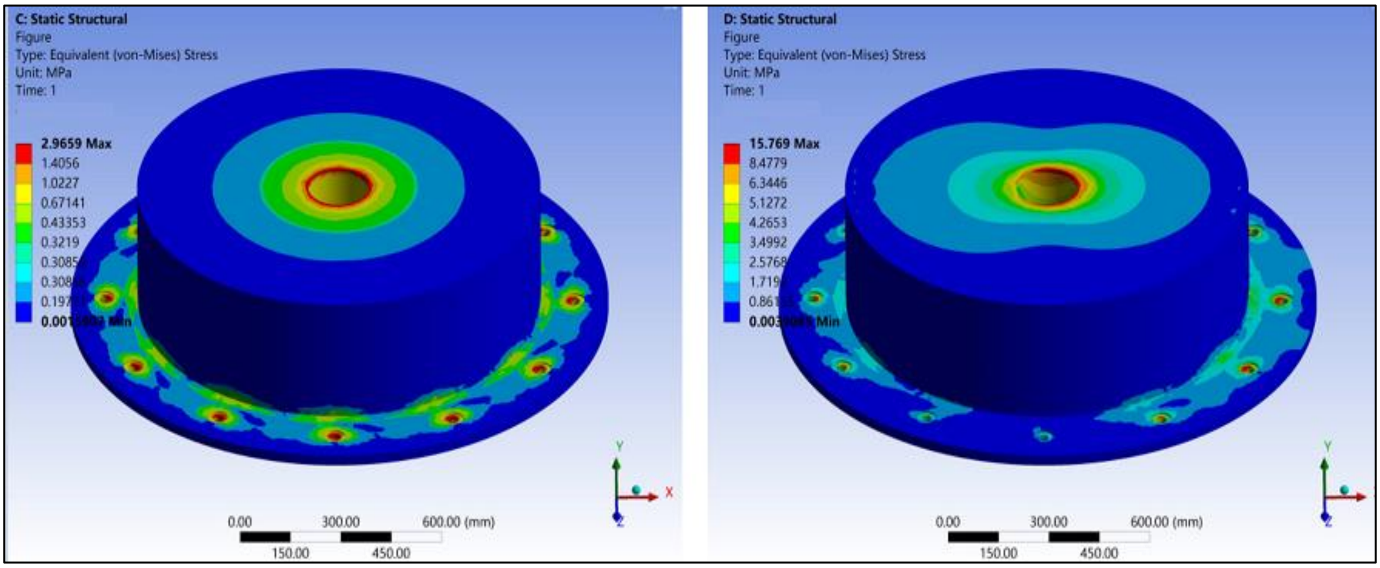

Fig 6: Complete Total Deformation Cloud Diagram of the Base Structure Subjected to Various Operational Settings (a) Subjected to Vertical Moment of Inertia (b) Subjected to Vertical Force

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24SEP877

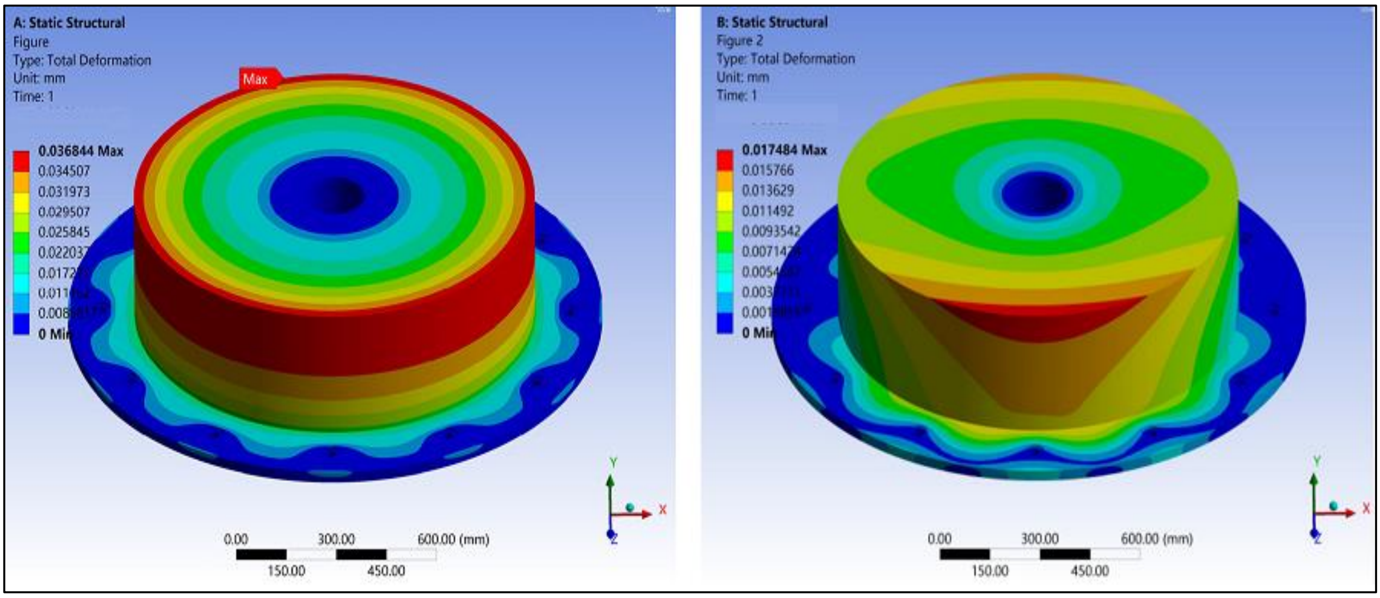

Fig 7: Complete Total Deformation Cloud Diagram of the Base Structure Subjected to Various Operational Settings (c) Subjected to Horizontal Moment of Inertia (d) Subjected to Horizontal Inertial Force

B. Optimization of the Robot Base Dimensions

From the FEA discussed in figures above, the base generally sustains minimal stress across most conditions, with significant exceptions under sheer inertia moment loads where stress concentrates around the holes of the bolt of the bottom plate. To enhance the welding workshop layout, facilitate the addition of more robots, as well as boost the automation rate, it's feasible to optimize the original structural dimensions. This contains reducing the thickness and external diameter of the cylinder sidewall and the plate of the bottom base plate, specifically downsizing the bottom plate diameter to 1,180mm, based on stress distribution and structural characteristics.

C. Finite Element Analysis (FEA) Results for the Revised Base Framework

The studied base component underwent the same finite-element procedures, yielding deformation and stress analyses presented in Figures 9 and 10. The modifications have slightly reduced stress and deformation levels; the maximum stress dropped to 102.36 MPa, and the maximum deformation decreased to 0.0302 mm. Notably, the 1st order natural-frequency became 1348.4 Hz, suggesting enhanced structural dynamics. These changes also adjust the robot base's bolt connection locations, further influencing the overall structural efficacy under several operational conditions. The analysis of performance, both static and dynamic, ahead of and after optimization is detailed in below points, demonstrating the significant improvements achieved through this optimization process.

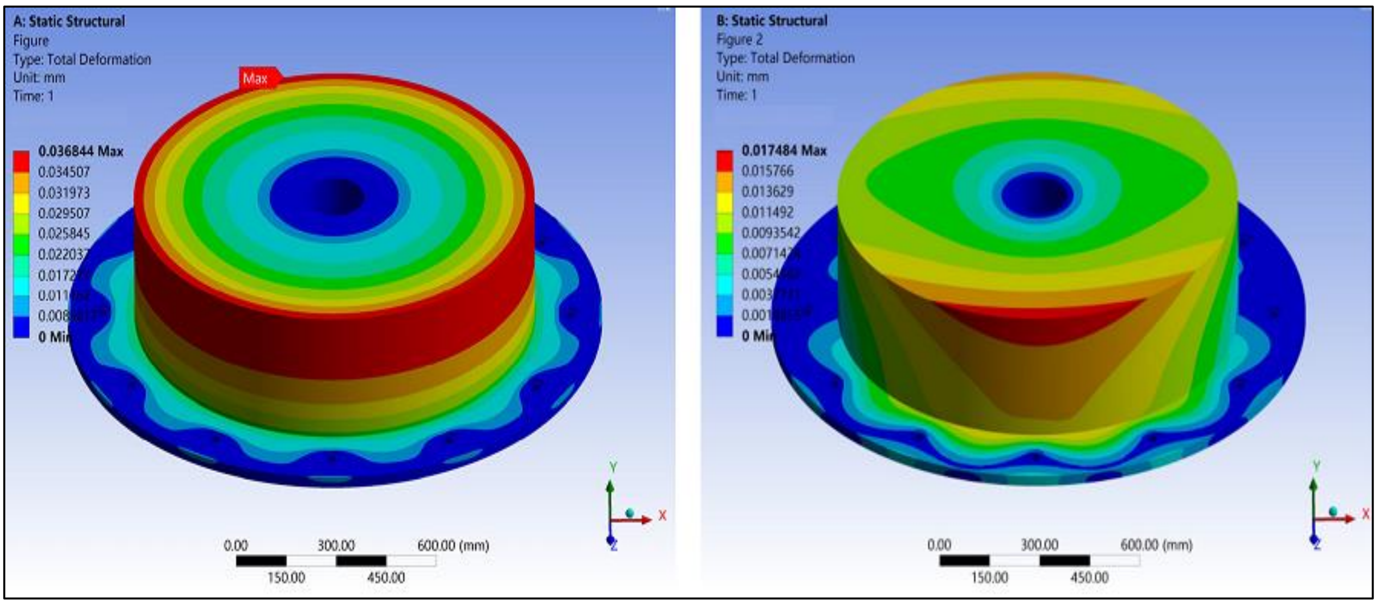

Fig 8: Comprehensive Diagram of Base's Von-Mises Stress Cloud Subjected to Vertical-Inertia Moment (a) Basic Configuration (b) Revised Configuration

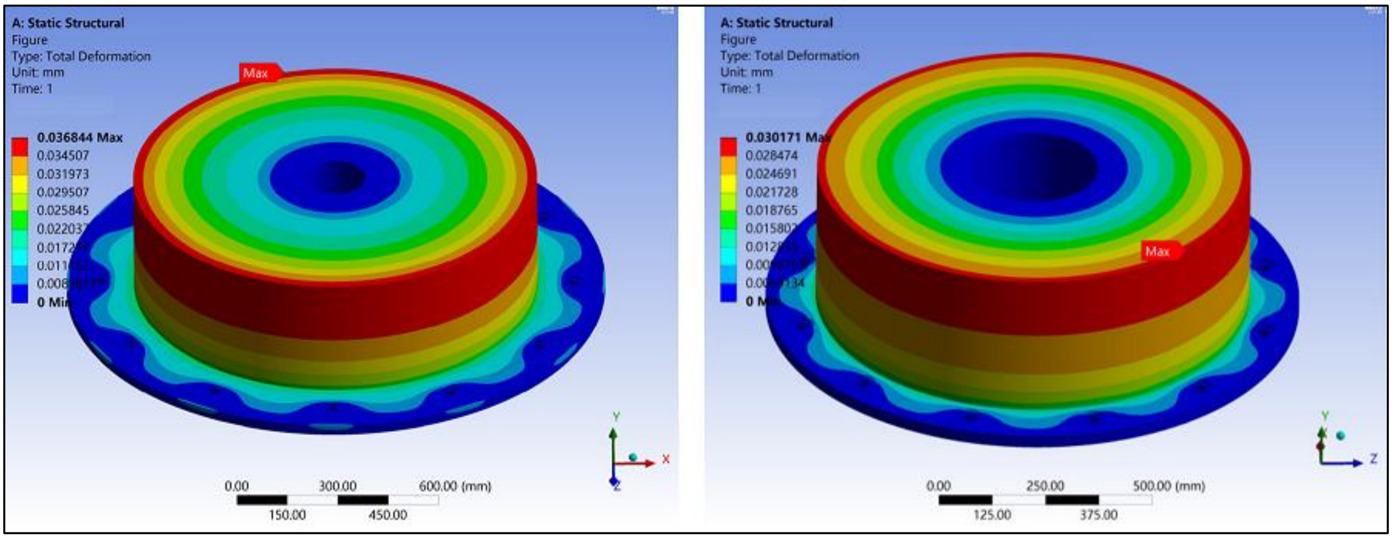

Fig 9: Deformation Cloud-Diagram of the Base Subjected to Vertical Inertial Moment Stress
(a) Initial Design (b) Improved Design

The following points present a comparison of the dimensions and both static and dynamic performance metrics for the base component before besides afterward optimization process:

- The size of Bottom Plate: Initially, the bottom plate size was 1650 mm, which was reduced to 1180 mm in the optimized design.
- Cylinder Wall Size: The cylinder wall size was reduced from 1200 mm to 1012 mm as part of the optimization.
- Distance from Base Plate Bolt to Centre: This measurement was reduced from 720 mm in the original plan to 560 mm in the optimized configuration.
- Maximum von-Mises Stress: The stress decreased slightly from 105 MPa in the original design to 102.36 MPa after optimization.
- Maximum Total Deformation: There was a reduction in deformation from 0.0368 mm initially to 0.0302 mm post-optimization.
- First Order Frequency: This frequency increased significantly, from 703.95 Hz in the original design to 1348.4 Hz in the optimized plan, indicating improved dynamic response characteristics.

V. CONCLUSION

In conclusion, the comprehensive study on the FEA and Design Optimization of a welding robot base utilising ANSYS has successfully demonstrated how structural integrity, operational efficiency, and lifespan of welding robots can be significantly enhanced. By meticulously refining the base structure's dimensions and employing advanced simulation tools, the study not only enhanced the robot's static and dynamic performance but also contributed to a more robust design capable of withstanding rigorous industrial demands. This research offers valuable insights that could be applied broadly across robotic applications, ensuring that robots continue to perform efficiently and reliably in demanding environments, thus bolstering safety and productivity in industrial settings.

REFERENCES

[1]. G. Yadav, A. Pirjade, M. Jadhav and V. Patil, “Design analysis and optimization of robot pedesta,” International Journal of Engineering and Technical Research (IJETR), 2016.
[2]. P. Kah, M. Shrestha, E. Hiltunen and J. Martikainen, “Robotic arc welding sensors and programming in industrial applications.,” International Journal of Mechanical and Materials Engineering, vol. 10, no. 1, pp. 1-16, 2015.
[3]. X. Li and J. Liang, “Optimization design and finite element analysis of welding robot base based on ANSYS Workbench,” In Journal of Physics: Conference Series, vol. 2383, no. 1, p. 012073, 2022.
[4]. Z. Poruba, J. Podešva, O. František, M. Fusek, R. Brázda and M. Sadílek, “Strength analysis and optimization of welding robot mechanism in emergency stop state,” Perspectives in Science, vol. 7, pp. 347-352, 2016.
[5]. X. Liao, C. Gong, Y. Lin and W. Wang, “The finite element modal analysis of the base of welding robot,” In 2010 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE), vol. 2, pp. 2-123, 2010.
[6]. G. Chung, D. Kim, H. Shin and H. Ko, “Structural analysis of 600Kgf heavy duty handling robot,” IEEE, pp. 40-45, 2010.
[7]. H. Varma and S. G., “Analysis and Optimization of Robot Gun Support Structure for Welding of Light Weight Vehicle Door Frame,” ISSN (Print), vol. 3, no. 6, pp. 2321-5747, 2015.
[8]. J. Niresh, M. Subramanian and S. Neelakrishnan, “Finite Element Analysis of Bridge Riser for Automobile Application Robots,” in In Proceedings of the First International Conference on Combinatorial and Optimization, ICCAP 2021, Chennai, India, 2021.
[9]. P. D. and W. W., “Optimal design of robot base in welding workshop based on finite element analysis,” J. China Water Transpo pp. 117-118, 2019.