Finite Element Analysis of Extension Bar For Pneumatic Impact Wrench

Chaitanya Kulkarni[1], Dr. Santosh Patil[2]
[1]Dept of Mechanical Engineering
[2]Professor, Dept of Mechanical Engineering
[1, 2] K.E. Society's Rajarambapu Institute of Technology, Sangli, India.

Abstract- *Accurate torque measurement plays a vital role in ensuring optimal performance and safety of mechanical systems across industries. This research paper presents a detailed analysis of an extension bar utilized in a smart torque measurement system specifically designed for a pneumatic impact wrench by employing advanced software tools like CATIA and ANSYS, the paper highlights the modeling process, material selection, and stress analysis to evaluate the extension bar's structural integrity. This analysis aids in optimizing the design and ensuring the long-term functionality and reliability of the system.*

Keywords- Finite Element Analysis, extension bar, stress analysis, pneumatic impact wrench, ANSYS, CATIA.

I. INTRODUCTION

The ability to accurately measure and control torque is of paramount importance in various industries, especially in the field of mechanical engineering. Torque measurement plays a vital role in ensuring proper assembly, maintenance, and performance of numerous mechanical systems. Pneumatic impact wrenches, also known as air impact wrenches or air guns, are handheld tools powered by compressed air. They are commonly used in industries such as automotive, construction, and manufacturing. These tools are designed to deliver high torque output for fastening or loosening nuts and bolts.

In the project "Design and Development of smart torque measurement system for impact wrench" we focus on creating a robust and efficient torque measurement system that provides accurate real-time data and enables precise control over torque application. The developed system consists of a torque measurement mechanism integrated into the extension bar of an impact wrench, as well as a wireless control panel for data display and manipulation. The torque measurement system utilizes the principle of measuring the torque applied to the extension bar, which serves as a reliable proxy for the overall torque exerted by the impact wrench. By accurately measuring the torque at the extension bar, we can effectively monitor and control the torque output of the impact wrench during various applications.

One crucial aspect of this project involves the analysis of the extension bar to ensure its structural integrity under Torque. The extension bar is constructed using high-strength chrome molybdenum steel material, selected forits excellent mechanical properties.

The stress analysis is conducted using advanced software tools such as CATIA and ANSYS for modeling and stress analysis respectively.The stress analysis allows us to understand the distribution of stress within the extension bar and identify potential areas of concern where failure may occur under extreme torque conditions. By conducting a comprehensive stress analysis, we can ensure that the extension bar is capable of withstanding the expected torque loads and guarantee the reliability and safety of the torque measurement system.

In this research paper, we present the modeling and stress analysis of the extension bar and compare the results to validate its structural integrity and determine its maximum torque-carrying capacity.

A. *Extension bar*

An extension bar serves as a vital intermediary component between a power tool, such as a wrench or drill, and the target object or workpiece. To ensure a secure and stable connection, extension bars are equipped with attachment mechanisms at each end.

One end is designed to fit the power tool, typically using a square or hexagonal drive that matches the corresponding drive size of the tool. This ensures a precise and reliable connection between the power tool and the extension bar.

At the other end of the extension bar, there is an attachment point where the socket or other tool-specific attachments can be affixed. The socket attachment point is designed to accept various socket sizes, allowing users to work with different bolt or nut sizes, further enhancing the versatility of the extension bar.

The physical structure of the extension bar, typically a cylindrical bar, ensures its strength and durability. Extension bars are commonly manufactured from materials such as alloy steel, chrome vanadium steel, chrome molebdinum steel, or aluminum alloy, chosen for their robustness and resistance to deformation. These materials enable the extension bar to withstand the forces applied during operation, ensuring its reliability and longevity.

II. MODELING OF EXTENSION BAR

The software used for modeling the component is CATIA V5. CATIA is a renowned computer-aided design (CAD) software widely used in industries such as automotive, aerospace, and manufacturing, CATIA offers a comprehensive suite of tools for 3D modeling, simulation, and analysis. The CAD modeling provides an accurate representation of the extension rod's geometry and enables further analysis and simulation for evaluating its structural performance.

Figure 1. Front view of CAD modeling of the extension bar

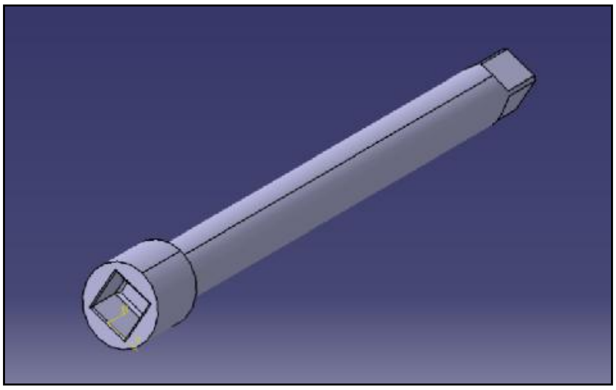


Figure 2. Isometric view of CAD modeling of the extension bar

Following are the material properties of the Impact socket (SAE 9840) that were entered into the ANSYS software for the analysis.

Table 1. Extension bar material properties.

Properties	**Units**	**Value**
Density	kg/m^3	7700
Poisson's ratio	-	0.29
Young's modulus	GPa	200
Tensile ultimate strength	MPa	1240
Tensile Yield strength	MPa	1105

III. FINITE ELEMENT ANALYSIS OF EXTENSION BAR

The finite element analysis has been done in ANSYS using the extension bar's technical specification. Both a higher order 3-D, 10-node SOLID187 element with three degrees of freedom at each node with transitions in the nodal x, y, and z directions and a higher order 3-D, 20-node SOLID186 element were employed.

The meshing details for the bar are as follows:

Number of nodes: 327360
Number of elements: 180377
Types of elements:
Tetrahedral (SOLID 187)
Hexahedral (SOLID 186)
Behaviour of element: 3 Degree of Freedom.

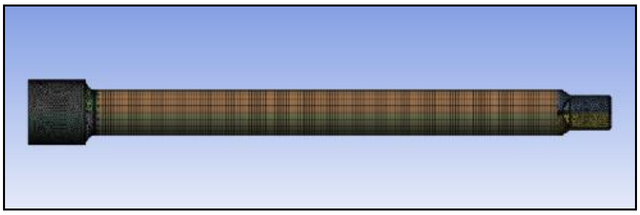


Figure 3. Meshing of extension bar

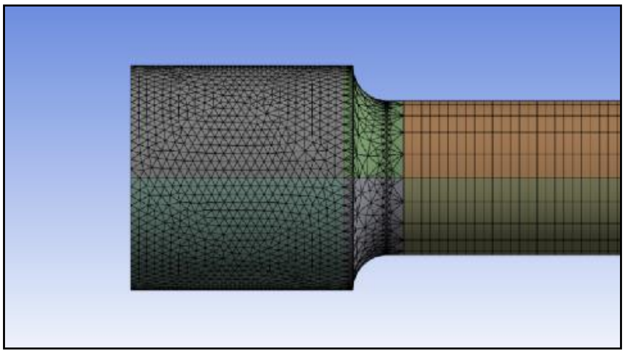


Figure 4. Meshing of drive end.

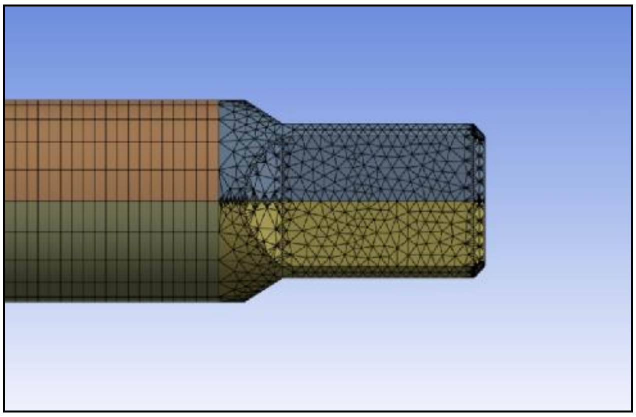


Figure 5. Meshing of socket end.

In the analysis, the extension rod was subjected to specific boundary conditions. One end of the rod was fixed to simulate a rigid support, restraining its translation and rotation. At the other end, a moment of 180 Nm was applied, representing the operational torque being exerted on the bar. These boundary conditions were employed to accurately simulate the real-world behaviour of the extension rod.

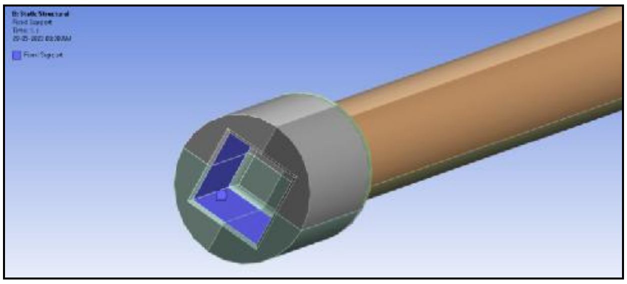


Figure 6. Fixed support applied at the drive end.

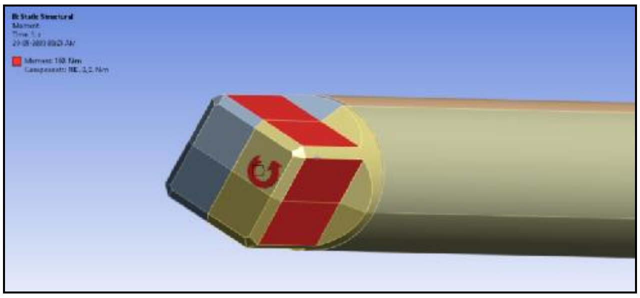
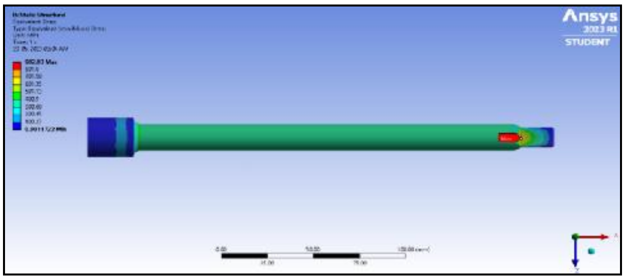


Figure 7. Moment applied at socket end.



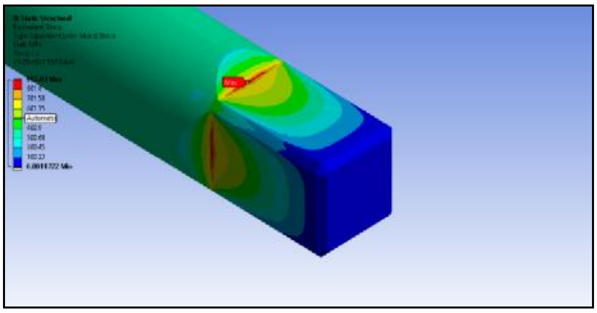
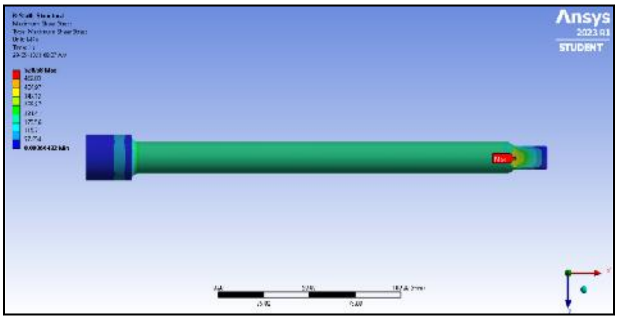


Figure 8. Equivalent Von-Mises stress distribution on the extension bar.



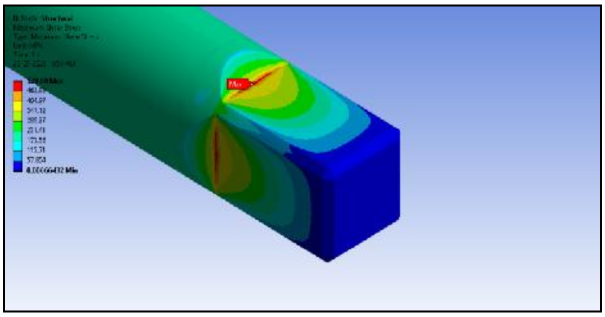


Figure 9. Shear stress distribution on the extension bar.

IV. RESULTS

Table 2. Result Table.

Stresses	**Ansys results**
Equivalent Von-Mises stress (MPa)	902.03
Maximum shear stress (MPa)	520.68

Based on the stress analysis results, the maximum Von-Mises stress and Maximum shear stress observed in the extension bar were measured to be 902.03 MPa and 520.68 MPa respectively. As we know the yield strength of the material is 1105 MPa. The maximum stress values are within the allowable stress limit. The factor of safety for the extension bar is approximately 1.2. This implies that the extension bar has a safety margin against yielding and is capable of withstanding the applied stresses without undergoing plastic deformation.

V. CONCLUSION

This study analyzed an extension bar made of SAE 9840 alloy steel that was used with a pneumatic torque wrench. Stress analysis performed using Ansys software revealed that the male part of the extension bar experienced the highest stress. The maximum Von-Mises stress observed in the extension bar was 902.03 MPa, while the maximum shear stress measured 520.68 MPa. Based on the stress analysis, it can be concluded that the extension bar demonstrates structural integrity and reliability for its intended use with a pneumatic impact wrench in the smart torque measurement system. These results contribute to the overall design assurance, ensuring the accuracy, performance, and safety of the torque measurement system.

REFERENCES

[1] Gopinath, D., &Sushma, C. V. (2015). Design and optimization of four wheeler connecting rod using finite element analysis. Materials Today: Proceedings, 2(4-5), 2291-2299.

[2] Raut, L. P. (2014). Finite Element Analysis of Impact Socket Used In Hydraulic Torque Wrench.

[3] Singh, P. K., Kumar, M.,& Mishra, S. (2022). Finite element analysis of functionally graded bar under torsional load. Materials Today: Proceedings, 56, 2960-2966.

[4] Lopatukhin, I., &Ber, A. (1997). Accuracy and strength of a joint loaded by suddenly applied torque. *The International Journal of Advanced Manufacturing Technology*, *13*(10), 692-695.

[5] Ali, H. M. M., &Haneef, M. (2015). Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end. Materials today: proceedings, 2(4-5), 2094-2103.

[6] Miao, R., Shen, R., Zhang, S., &Xue, S. (2020). A review of bolt tightening force measurement and loosening detection. Sensors, 20(11), 3165.

[7] Bansal, R. K. (2010). A textbook of strength of materials:(in SI units). Laxmi Publications.