ANALYSIS OF ROBOTIC END EFFECTOR

Abstract

In this study analysis of end effector of a pick and place robot is carried out to analyze its strength and develop aroboticend-effect or for machining and part handling. Industrial robotic manipulators require an end effector to perform tasks. The device eliminates the need for separate robots to perform part handling and machining operations. Robots are very important in the field of automation of machinery. Performance of the robotic manipulator is dependent on the work done by the end effectors. The selection of end effector is based on the type of task that has to be performed. For picking, holding, and placing tasks to the specified end effector is selected and various different types of workshop operations tools are fixed on the manipulator for different applications e.g., welding electrode holder, painting spray gun etc. In the present scope of work, design and development of robot end effector by using Solid Works software is carried out. End effector model is imported to ANSYS Workbench to analyze the strength. Analysis for different materials with same joint displacement are carried out and compared with the results equivalent deformation, equivalent strain and equivalent stress for both titanium alloy and structural steel. Selectingthe best material or an application in material handling is very important. Further for the best material, design of end effector will beo ptimized for the industrial robot application in the future scope of work.

Keywords: End effector, robotic manipulator, Solid Works, ANSYS Workbench, Structural Steel, Titanium Alloy, Connections, Joint displacement**.**

Authors

Venkatesh Murthy K
PG Scholar
Machine Design Engineering
Department of Mechanical Engineering
Ghousia College of Engineering
Ramanagara
venky14murthy@gmail.com

Dr. Mohammed Imran
Assistant Professor,
Department of Mechanical Engineering
Ghousia College of Engineering
Ramanagara
mechmdim4an@gmail.com

I. INTRODUCTION

An end effect or is a component that attaches to a robot's manipulator/wrist,which allows the robot to perform its task. In general end effectors uses mechanical or electro mechanical source and work as grippers, process tools, or sensors. They come in wide variety of range from simple two-fingered grippers for pick-and-place tasks to complex sensor systems for robotic inspection. Without an end effect or, it is difficult or merely impossible for robots to work. We can make a robot to move to a particular location but without end effect or it has no way to perform any operation.

Robot manipulators work from a sequence of links, joints and contact combinations. These links are rigid members that are connected to the joints, or axes. There lative motion between adjoining link scan be obtained by the axes of the movable components of the robotic manipulator. There are five principle types of mechanical joints that are used to construct the robotic arm manipulator out of those five types, two joints are linear and three joints are rotary. The linear joints have non-rotational relative motion between adjacent links whereas the rotary types have rotational relative motion between links. There are six robotic configurations namely Cartesian, spherical, Selective Compliance Articulated Robot Arm (SCARA), Cylindrical, Delta (Parallel) and articulate. The arm-body section of robotic manipulators is based on one of these configurations. Depending upon differentapplications each and every configuration of these anatomies gives a different work envelope [7]. Articulate type of robotic manipulator configuration boundary conditions are used to analyze the end effector with a displacement of 5mm [1].

II. LITERATUREREVIEW

Anurag Sharma [1] studied the end effectors' where he is selecting various grippers, like3finger gripper, 2 finger gripper, vacuum gripper, etc. also he did study on robotic tools likedrilling spot welding after all he concluded that, the selection of correct end effector is veryimportant and necessary considering the robot manipulator work space limit and type ofoperations to be performed on the type of works piece. In this study a co-relation and co-ordinationofdifferenttypesofendeffectorsisshownforcompletingthedesiredtaskcompletelyand satisfactorily.

P.Ferdinandlivi joseph et al,[2] modeled the robotic arm where everyone using same material, he focused on reducing the weight of the robotic arm using different alloys and compositematerials.Forexample,structuralsteel,aluminumalloy365andARAMID Epoxy. After modeling the model using Solid works, they analyzed all three-material using ANSYS Workbench and concluded that structural steel and ARMID Epoxy Broke while analysis, where as the Aluminum 365 withstand.

Treveljian et al.[3] has studied about how the Robot wrists are designed to provideorientationtotheendeffector.Whichhasa"sphericaljoint",becausetheresultingconfiguration is more dexterous and less cumbersome than the other configurations. Robotwrists should have a low degeneracy level that represents the region where some rotationsaroundcertainfixedaxesintheCartesianspacearerequirehighspeed actuators.

Ho Choi & Koc, 2006 [4] based upon the feasible test results of a flexible gripper design, it was designed, analyzed, produced and testedwith the use of compliant materials (i.e., rubber) with pneumatic inflation.FEA Parametric analyses were conducted to investigate the effects of process and design parameters, such as rubber material, pressure, initial jaw displacement and friction. Based on these results obtained by the FEA analyses, a simple, single rubber pocketed flexible gripperwas designed and built.

Biswal,1997[5]designed and built a two-jawactuated gripper with the help of pneumatic cylinders using air pressure, force and torque which have been calculated for different set of conditions in which the jaws have controlled movement which was different from the conventional cam and follower gripper.

Kiran kumar B M et al [6], this paper describes the design and development of a robotic arm with five degrees of freedom (DOF) that is used to feed the elderly or individuals with disabilities. The user may regulate the position of the joints. The robotic arm is operated using MATLAB and robotic kinematics concepts. Stick diagrams are used to verify the algorithms. The actuators, which in turn control each joint of the robotic arm, are controlled via a graphic user interface. To verify the correctness of the method, a mockup of the produced robotic arm is created in MATLAB and simulated using forward and inverse transformation.

Ali Murtatha Shuman etal [7] the creation of a joint space inverse dynamic controller and a method to describe a 6-axis robot utilizing its dynamical characteristics are both covered in this thesis. Different methods will be used to test the controller. In the beginning, by introducing noise to the measured data. Assessing the control model's resilience while taking into account attributes unique to the simulated model from those utilized for the controller itself. Payloads and link inertia are two examples of the many attributes. Afterwards assessing the accuracy of the path that was taken based on the operational space trajectory. The findings of these trials are encouraging. The outcomes demonstrate the controller's capacity to regulate joint angle and joint velocity noise. It also demonstrates how a little inaccuracy in the joint angles caused by a payload data error leads to an acceptable error for the end-effector in the operational space. Additionally, when following a trajectory in the operational space, the controller is able to maintain a minimal maximum error in the joint angle.

Gob-Soon Kim et al [8]to measure force and torque during hip-joint rehabilitation exercises utilizing a lower rehabilitation robot, a two-axis sensor with parallel-plate beams (PPBs) and single beams was designed and made. The force sensor and torque sensor, which measure z-direction force and z-direction torque, respectively, make up the two-axis force/torque sensor. The two-axis force/torque sensor was created using strain gages and built using FEM. The two-axis force/torque sensor's characteristics experiment was conducted. The test results reveal that the two-axis force/torque sensor's interference error, repeatability error, and non-linearity were all less than 0.64% and less than 0.03%, respectively. The created two-axis force/torque sensor is envisioned as being used to a lower rehabilitation robot.According to

Jianqiang Wang et al. [9], intelligent grasping may be accomplished by gripper design, automatic component detection, an intelligent algorithm for gripper control, and real-time decision-making using sensory input. Based on an industrial robot from ABB, a general framework is built to combine sensory input, component recognition, decision-making, and gripper control to accomplish intelligent grasping.The system software is created using object-oriented programming techniques built on Visual C++ MFC to assure compatibility, expandability, and modular programming architecture. It is explained how to divide the robot's capabilities into high-level (modeling, recognizing, planning, and perceiving) levels and low-level (sensing, interacting, and executing) layers using a hierarchical design for intelligent grasping. The intelligent grasping system is made up of separate system elements that have been flawlessly combined.

Anurag singh et al [10]in this study, the stresses and overall deformation induced for a certain robot payload are analyzed and investigated. Solid works was used to create the design and model for the five-DOF robot arm, and ANSYS software was used to evaluate the entire structure. With the use of the ANSYS program, a robotic arm is simulated, and various systemic features of a robotic arm are investigated. The primary goal is the evolution of a design that can accurately predict the robot arm's position in specific arm postures and high-stress situations. Further structure upgrading should be conceivable if disparate nozzle weights are pressed and final data from diverse conditions are compared to identify the weak areas.

Manikandan Ganesan et al [11] this research focuses on developing and studying a flexible robotic arm that can perform tasks related to material management. The investigation of different components and loading scenarios was done using the SOLIDWORKS® software for design and simulation, and the ANSYS® software workstation was utilized for numerical simulation analysis. To choose the right material and confirm that the articulated robotic arm was practical, the investigation's findings are analyzed. This robotic arm might be used for tasks including picking and putting, assembling, and other tasks in a number of industries. The arm appears to match design requirements and be able to support a range of payloads. This will help with productivity and is suited for locations that are hazardous.

Tanmay Deshpande et al [12] a robot's index finger sits in its palm and is used to carry out certain tasks or contact objects to sense them. Evaluation of the structural analysis of the Index Finger Assembly is the paper's primary goal. Total deformation, Stress and Strain graphs will be the results. SOLIDWORKS and ANSYS Workbench are used to build the model and conduct the analysis. Index Finger Assembly underwent structural study using ABS (Acrylonitrile-Butadiene-Styrene) and structural steel to confirm equivalent elastic strain, equivalent stress, and total deformation.

Supriya Sahu et al [13] The aim of the current study is to use the finite element method to measure the deformation, stresses, shear elastic strain, and strain energy at various sites on an industrial robot in order to assess its safety. ANSYS software is used to create the robot model, and finite element analysis is carried out. In order to identify the weak points in the design and make future design improvements, various values of typical gripper loads are applied and results obtained under various scenarios are compared. The finite element study produced remarkably good findings, with thorough evaluation of maximum deformation, maximum stress, and maximum strain. To replicate the behavior of an industrial robot, a FE-based model must be put into practice.

Harihara. S et al [14] the modeling and analysis of an articulated robotic arm that may be used for painting jobs are key to this research. SOLID EDGE software is used to develop and simulate the articulated robotic arm. Utilizing the ANSYS software workbench with varied loading circumstances and different size ranges, the finite element analysis was performed. The analysis findings are examined in order to choose the ideal structural parameter and to confirm the articulated robotic arm's viability. The findings unmistakably demonstrate that an articulated robotic arm is stronger and has a reduced vibration impact when applying a transitory weight of 3 kg at different angular velocities. If the payload is the same but various dimensions are used, there will be a minor increase in distortion. According to the results of the program, the end effector may be used to perform painting tasks effectively with a payload of up to 3 kg without experiencing any deviations. With this approach, complex and difficult tasks might be completed more quickly and precisely. Therefore, this manipulator satisfied the requirements for the job and is acceptable for carrying out work in hazardous areas in sectors that qualify for future research.

Jeevan et al [15] a finite element analysis (FEA) design optimization process is examined, and the resulting arm design is given. High structural vibration frequencies are predicted by FEA simulations of the final design throughout the workspace for the arms. The robot arm's various structures are examined using modal analysis, and it is discovered that the natural frequency of the structure is higher for circular-shaped robot arm structures, whether hollow or solid. This means that circular-shaped robot arm structures can withstand more vibrations than square-shaped robot arm structures. The findings of the structural study indicate that the equivalent stress and deformation of a robot arm with a circular shape are superior in comparison because the stresses raised are lower. The study is considered valid since the findings fall within safe design parameters.

Adrian Ghiorghe et al [16] Using a structural optimization and topology technique, the current research demonstrates a methodology for choosing the best values for design parameters while taking into account the need to minimize the amount of material utilized to construct an industrial robot's structure. This study, which is based on the finite element method (FEM), entails finishing the design model by employing the dimensions data as parametric design variables to which limitation criteria have been applied to accomplish the object function. To determine the ideal composition of the object function, a recurrent FEM analysis was conducted utilizing various settings for the design variables.

III. OBJECTIVE AND METHODOLOGY

1. Objective

- Analysis carried out to predict the strength and dynamic behavior of end effector for different materials with asame load in terms of joint displacement of puller rod.
- The dynamic behavior of the end effectors with respect to 5mm displacement of the push rod will be observed to identify the behavior of end effect or according to the present condition.
- The results of the two different materials are compared and concluding the factors of using it within certain parameters.

2. Methodology

- Modeling of robot end effect or using solid works software.
- Analysis by using ANSYS workbench.
- Analysis carried out for different materials with as a me joint displacement and compared with the results.
- Analyzing the behavior of the component with different material application.
- Selection of best material for industrial applications.

- **Modeling of robot end effector using solid works software.**

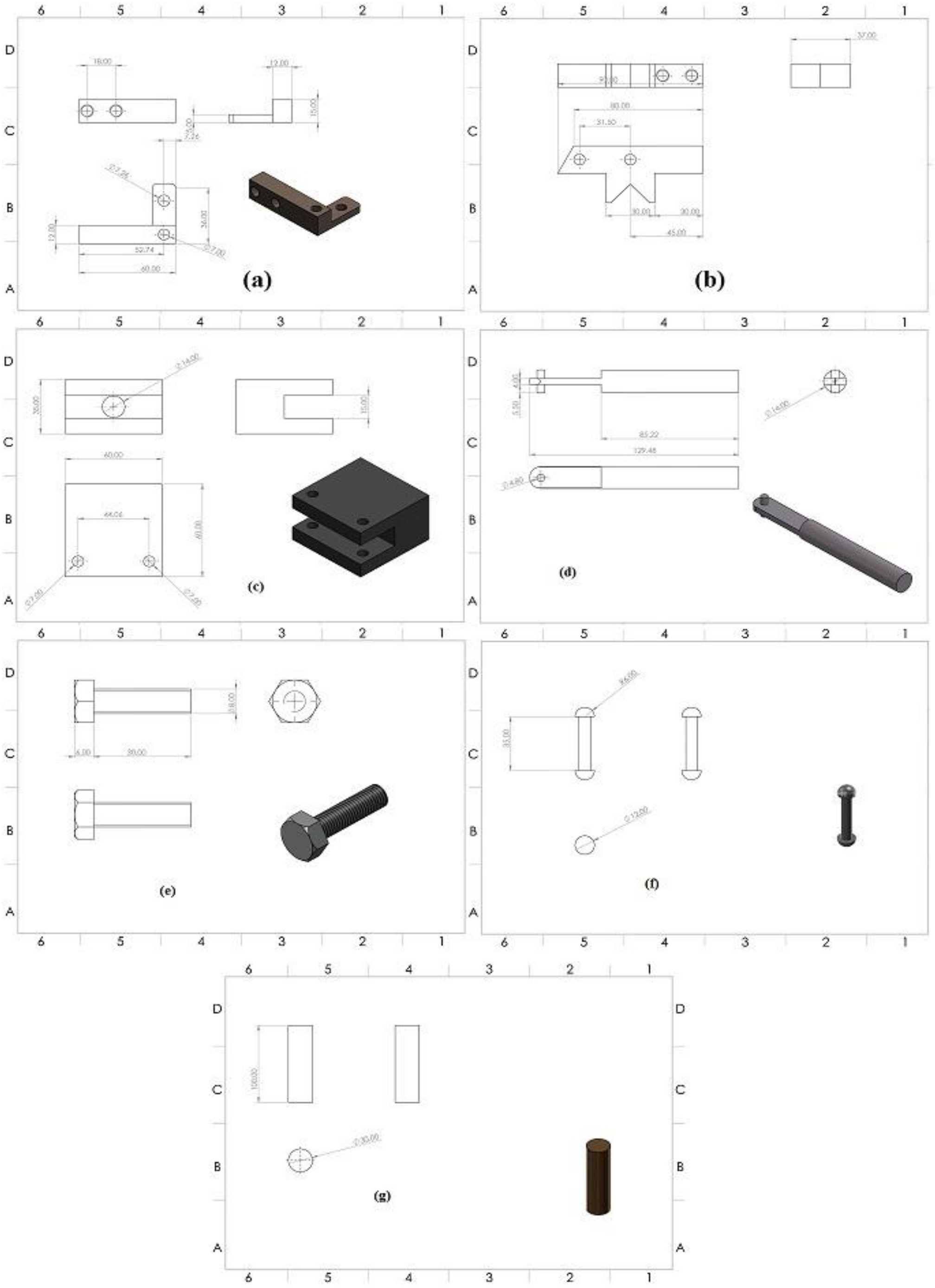
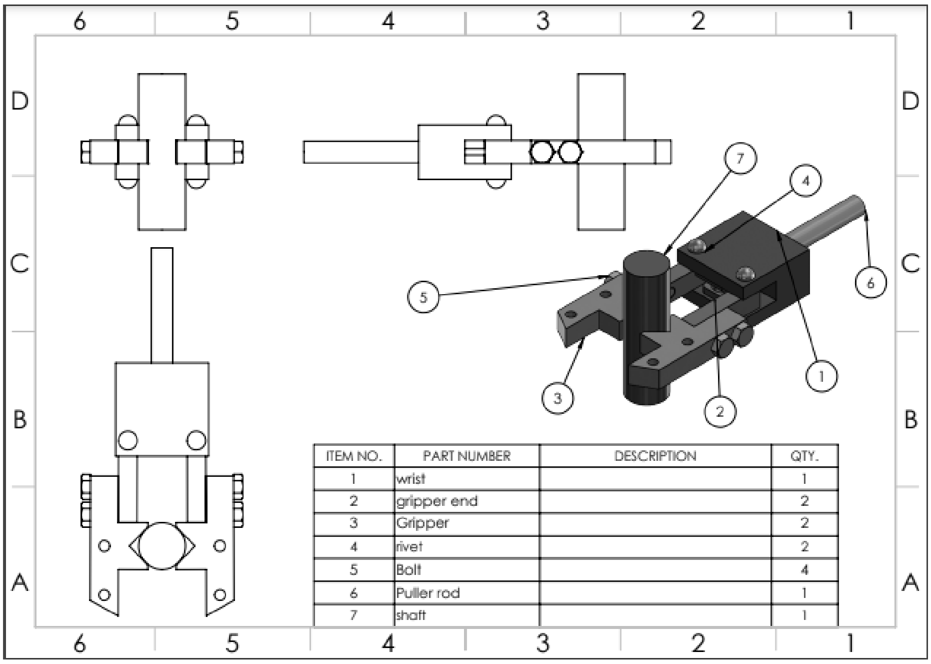


Figure 1: Dimensions of all the components (a) Gripper end (jaw grip) (b) gripper (jaw) (c) wrist (main body) (d) puller rod (push rod) (e) bolt (f) rivet (g) shaft

The model of end effector is createdin Solid Works software with all dimensions including 2D and 3D dimensions, to minimize the errors model has created from 2D to 3D as shown in figure 1. Figure 2 and figure 3 shows that the assembly of components and 3D view of end effector respectively.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	wrist		1
2	gripper end		2
3	Gripper		2
4	rivet		2
5	Bolt		4
6	Puller rod		1
7	shaft		1

Figure: 2 Assembly of components

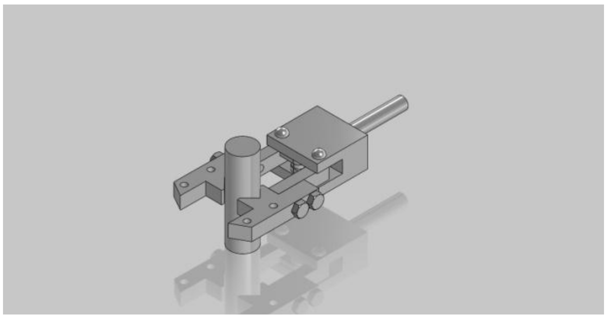


Figure: 3 3D view of end effector

- **Analysis by using ANSYS workbench**

  - **Engineering Geometry:** As shown in Fig 3 the assembled components of end effectoris designed and created using the Solid Works software and saved in IGES format. Further the 3D model is imported to the analysis software ANSYS Workbench in IGES format.

  - **Materials Used**

Table 1: Properties of materials

Properties	Structural Steel	Titanium Alloy
Density ton/mm^3	7.85×10^{-9}	4.43×10^{-9}
Coefficient of Thermal Expansion /^{o}C	1.2×10^{-5}	8.5×10^{-6}
Specific Heat MJ /ton ^{o}C	4.34×10^{8}	5.44×10^{8}
Thermal Conductivity W/ mm ^{o}C	6.05×10^{-2}	7.2×10^{-2}

Compressive Yield Strength MPa	250	1080
Tensile Yield Strength MPa	250	862
Tensile Ultimate Strength MPa	460	1200
Young's Modulus Mpa	2.0×10^5	1.2×10^5
Poisson's Ratio Mpa	0.3	0.35
Bulk Modulus Mpa	1.6667×10^5	0.96×10^5
Shear Modulus MPa	76923	45000

Material selection is an essential aspect of product design and development. The material selection should be done not only to meet the application requirements of the product, but it must be cost-effective also.Selection of suitable materials is very much significant in designing as it contributes to the enhancement of time and cost efficiency.

- **Connections and Joints:** The links in between the end effector components are Cylindrical, Revolute, Fixed to the partand Ground to body, Body to body. These are defined because during the joint displacementalong Z-axis dynamic behaviour are observedin order to analyze the model.The main body, jaws, bolts are fixed (fig 4), the rivets make revolute connections with the main body (fig 5), whereas cylindrical connections with the movable jaws (fig 7), the puller rod makes transitional connections (fig 6), and the shaft makes rough contacts with the fixed jaws (fig 8).

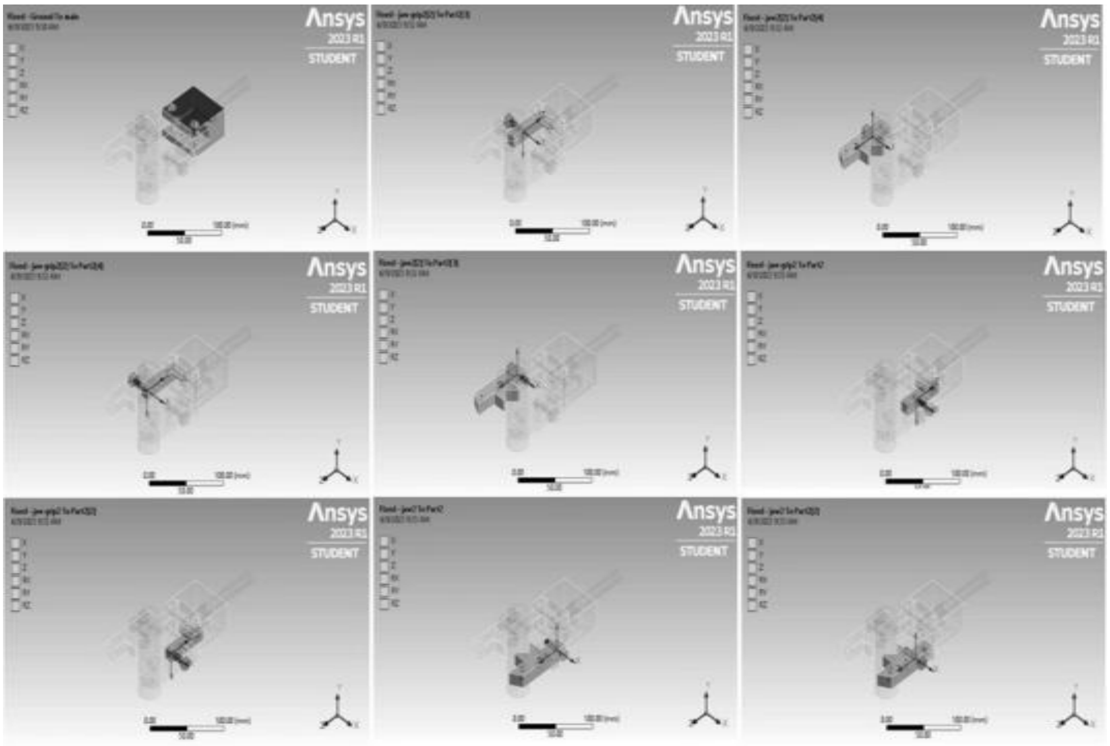
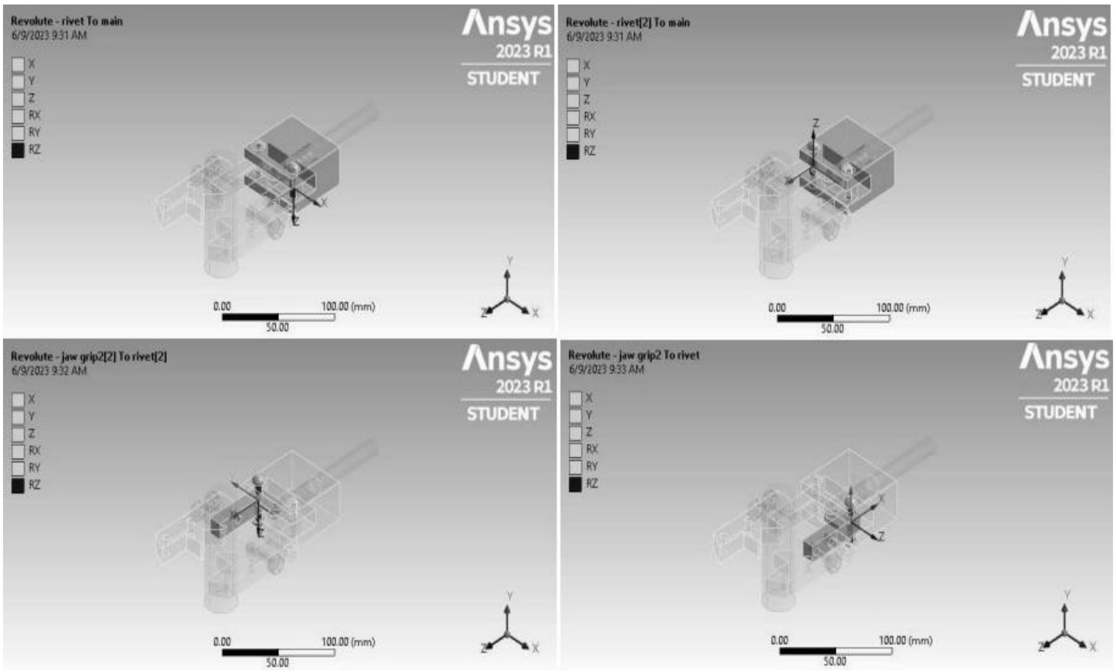


Figure 4: Fixed joints



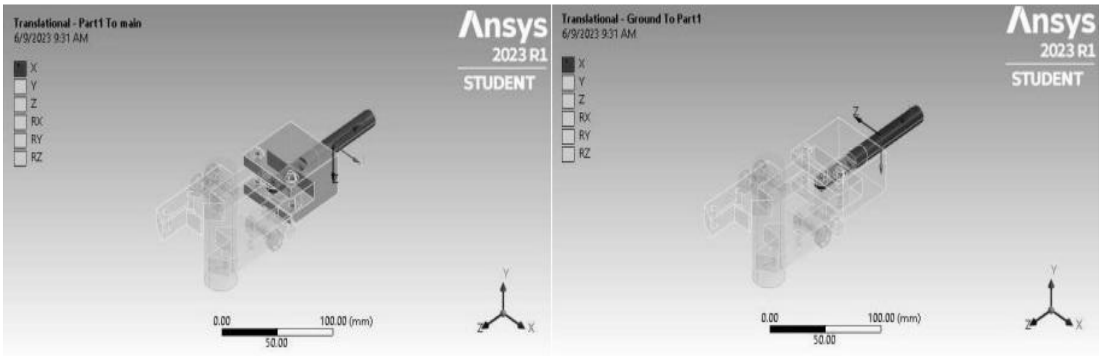
Revolute - rivet To main
6/9/2023 9:31 AM
☐ X
☐ Y
☐ Z
☐ RX
☐ RY
■ RZ

Revolute - rivet[2] To main
6/9/2023 9:31 AM
☐ X
☐ Y
☐ Z
☐ RX
☐ RY
■ RZ

Revolute - jaw grip2[2] To rivet[2]
6/9/2023 9:32 AM
☐ X
☐ Y
☐ Z
☐ RX
☐ RY
■ RZ

Revolute - jaw grip2 To rivet
6/9/2023 9:33 AM
☐ X
☐ Y
☐ Z
☐ RX
☐ RY
■ RZ

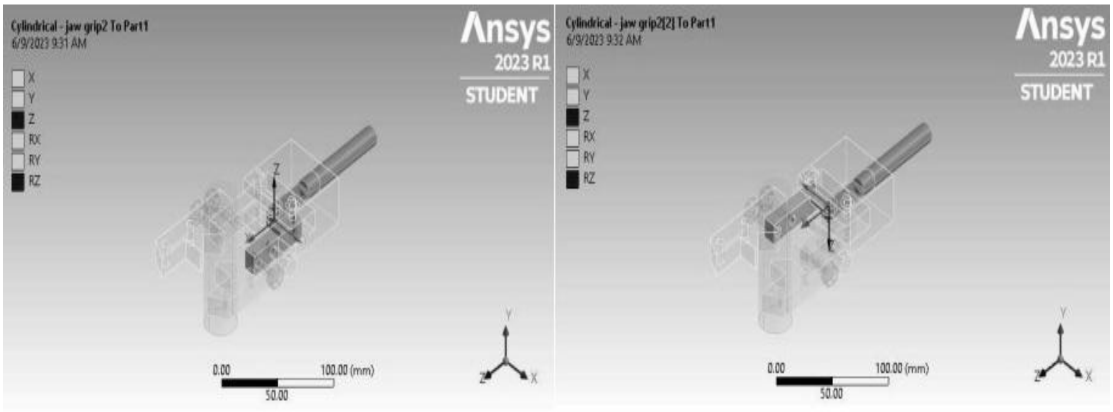
Figure 5: Revolute Connections



Translational - Part1 To main
6/9/2023 9:31 AM
■ X
☐ Y
☐ Z
☐ RX
☐ RY
☐ RZ

Translational - Ground To Part1
6/9/2023 9:31 AM
■ X
☐ Y
☐ Z
☐ RX
☐ RY
☐ RZ

Figure 6: Translational connections



Cylindrical - jaw grip2 To Part1
6/9/2023 9:31 AM
☐ X
☐ Y
■ Z
☐ RX
☐ RY
■ RZ

Cylindrical - jaw grip2[2] To Part1
6/9/2023 9:32 AM
☐ X
☐ Y
■ Z
☐ RX
☐ RY
■ RZ

Figure 7: Cylindrical connections

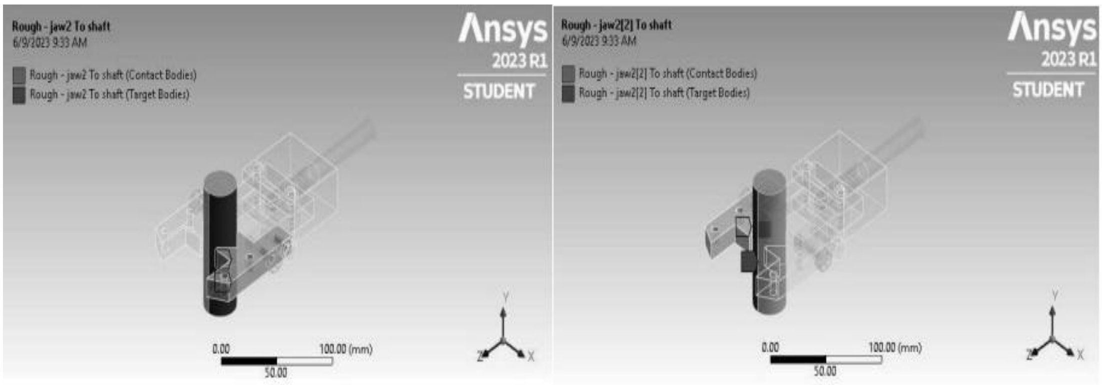


Figure 8: Rough Contacts

- **Meshing:** Meshing is the main step in performing an accurate simulation using FEA. Mesh consists ofnumber of nodes and elements that represent the geometrical modelling shape (fig 9 and table 2). The process of converting geometric modelling into FE modeling where the most complex and irregular shapes are discretized into more recognizable volumes (elements) is called meshing.

Table 2: Meshing Statistics

Statistics	
No of Nodes	23243
No of Elements	12571

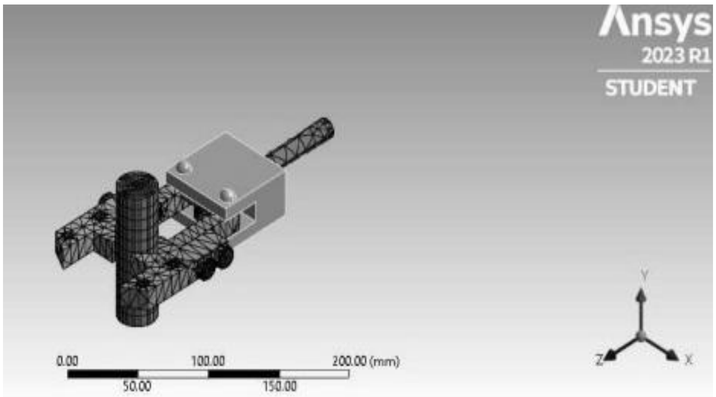


Figure 9: Mesh Plot

- **Analysis Settings and Loading:** Transient analysis involves the study of time-dependent behavior of physical systems or structures. Transient analysis of end effector in ANSYS Workbench allows us to study the dynamic response of end effector subjected to time-varying loads. It helps us to determine the stress, strain and deformation of the end effector under 5mm displacement with respect to 0 to 5 seconds (fig 10 and table 3).

Table 3: Joint Displacement

Steps	Time (s)	Displacement (mm)
1	0	=1
	1	1
2	2	=2
3	3	=3
4	4	=4
5	5	5

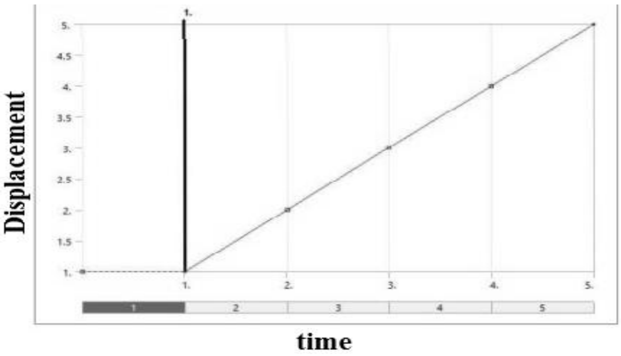


Figure 10: Joint Displacement

IV. RESULTS AND DISCUSSIONS

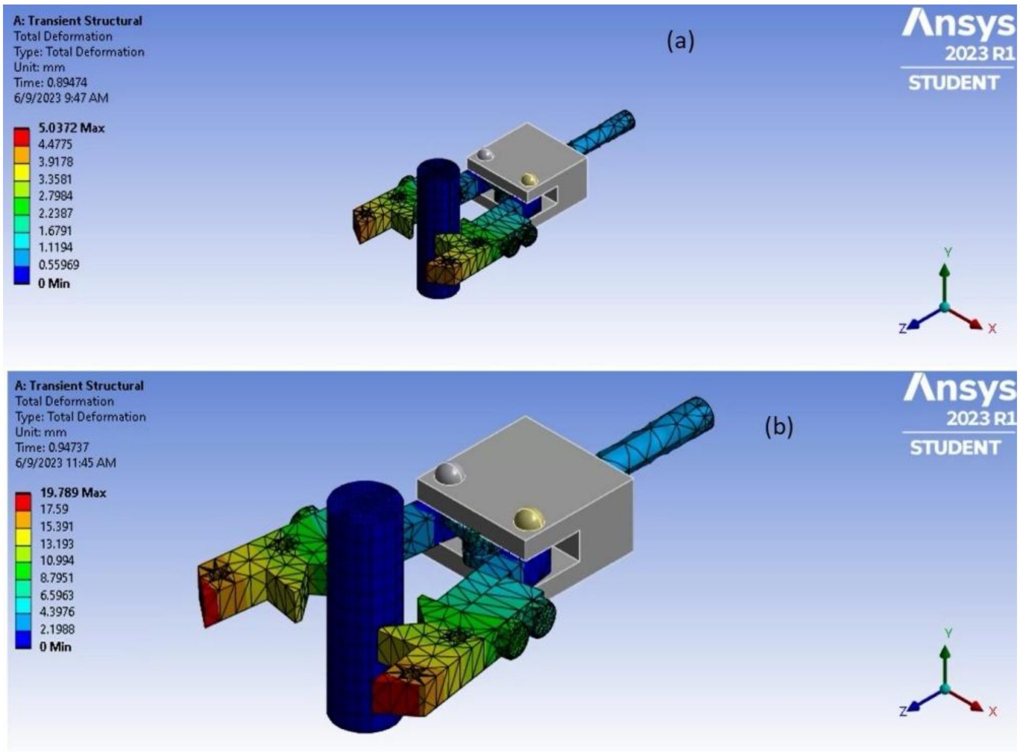


Figure 11: Deformation Results (a) Structural steel (b) Titanium alloy

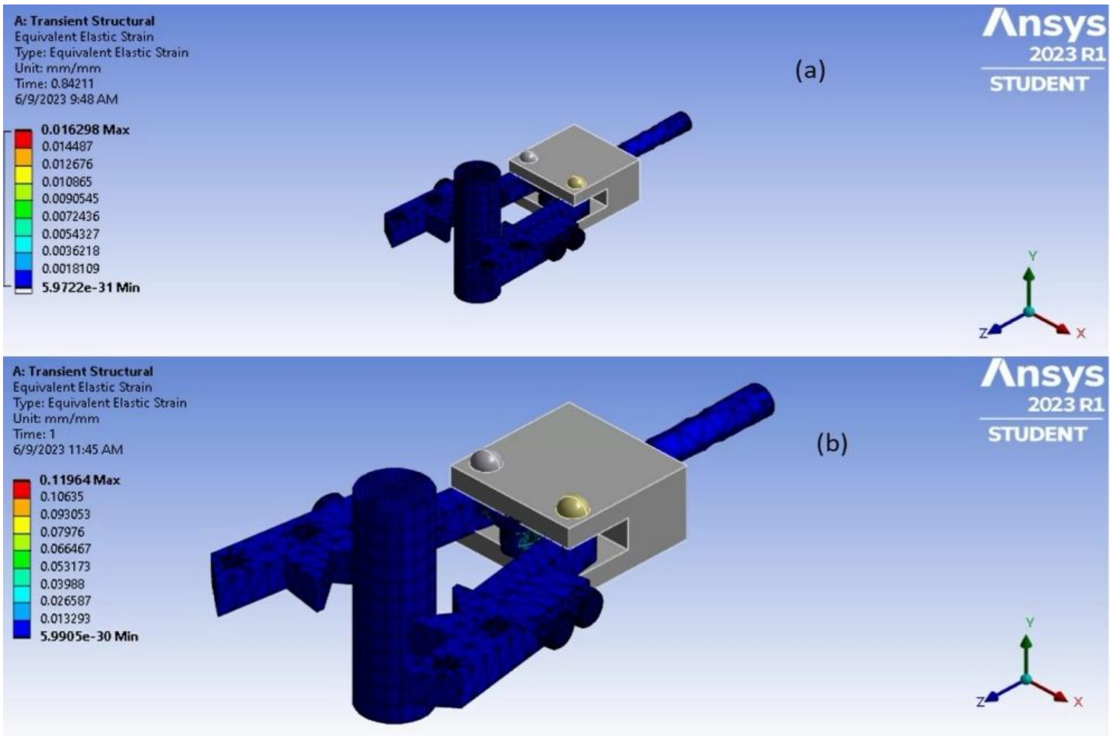


Figure 12: Strain Results (a) Structural Steel (b) Titanium Alloy

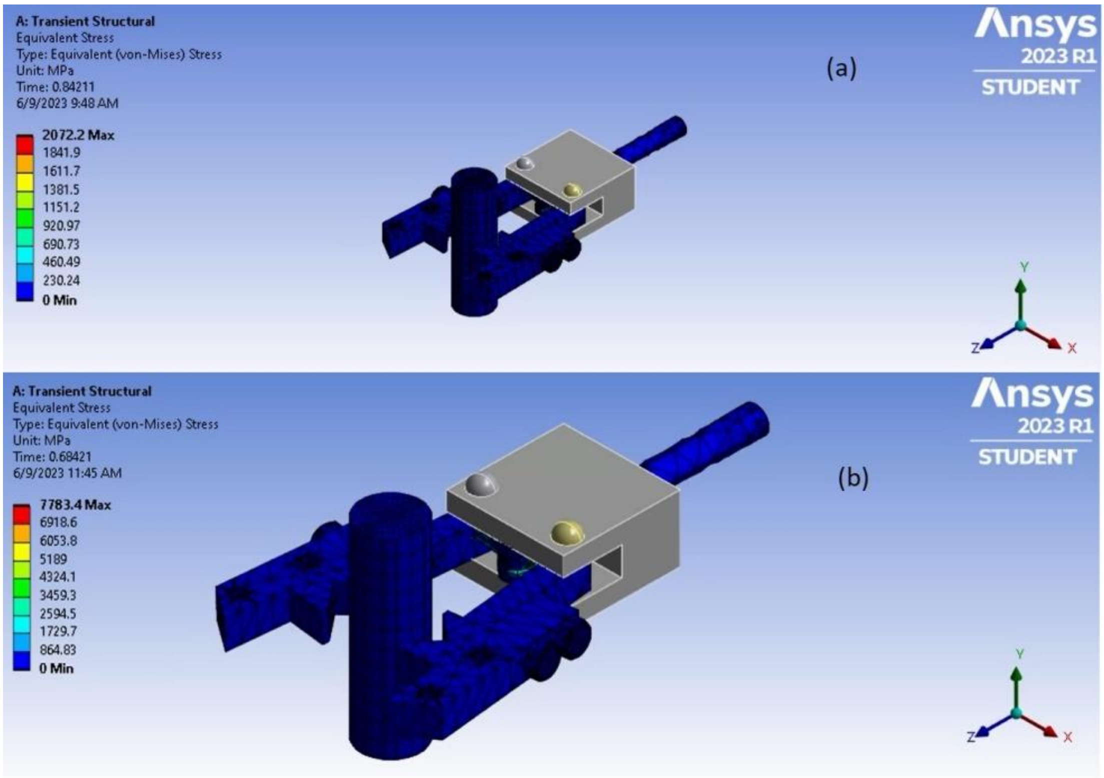


Figure 13: Stress Results (a) Structural Steel (b) Titanium Alloy

Table 4: Results

Type of material	Average deformation (mm)	Average Equivalent Strain	Average stress (Mpa)
Structural steel	3.309	7.3576e-004	110.49
Titanium Alloy	3.3344	7.8052e-004	78.395
Percentage variation	0.76%	5.7%	29.04%

In table 4, it is shown that structural steel has an average deformation of 3.309 mm and a range of 0 to 5.0372 mm, while titanium alloy has an average deformation of 3.3344 mm and a range of 0 to 19.789 mm. Figure 11(a) shows the deformation results of structural steel, which has a deformation range of 0 to 5.0372 mm and 3.309 mm.

Structural steel has a strain range from minimum 5.9722 e-31 to maximum 0.016298, and the average equivalent strain is 7.3576e-004 as shown in table 4. In contrast, titanium alloy has a strain range from minimum 5.9905 e-30 to maximum 0.11964 shown in fig 12(b), and the average equivalent strain is 7.8052e-04 as shown in table 4.

The stress results for structural steel are shown in Fig. 13(a), where the stress ranges from 0 MPa to 2072.2 MPa maximum, with an average equivalent stress of 110.49 MPa as shown in Table 4. The stress results for titanium alloy are shown in Fig. 13(b), where the stress ranges from 0 MPa to 7783.4 MPa maximum, with an average equivalent stress of 78.395 MPa as shown in Table 4. From the table 4 observed that the equivalent stress is less in Titanium alloy compared to Structural steel by 29.04%, the total deformation and equivalent total strain changes very slightly that is 0.76% and 5.7% respectively. Based on these results titanium alloy is preferable for this type of end effector since the stresses induced are less however taking the cost into consideration structural steel is preferable but only for small displacements that are less than 5mm.

Based on this analysis of end effector, the maximum stresses are acting at the pivot point where the push rod is making contact with the jaw which is shown if fig 7 , the pivot point of the push rod is making cylindrical contact with the jaw, the stresses acting here are maximum because of very less cross section hat is provided which can be increased in the future work since it is very much possible to increase the cross section by at least 2mm that can be accommodated in the present design itself.

From this analysis, it is concluded that titanium alloy shows better results when compared to structural steel material. For displacement of the push rod which are less than 5mm, then structural steel is sufficient considering the factor but when the displacement is more than 5mm titanium alloy is needed to withstand though the cost is on higher side compared to structural steel.

V. CONCLUSION

- According to this analysis of the end effector, he pivot point where the push rod makes cylindrical contact with the jaw is where the most stresses are acting. This is illustrated in

Fig. 7. Because this point has a very small cross section, it is very likely that the cross section can be increased in future work by at least 2mm, which can be accommodated

- This investigation leads to the conclusion that titanium alloy performs better than structural steel material. Structural steel is enough for push rod displacements under 5 mm, but when the displacement exceeds 5 mm, titanium alloy is required to endure, even if its cost is more than that of structural steel.
- The most affordable tool for simulation that uses a discrete method and produces excellent results quickly is ANSYS.
- The ability to safely simulate settings that are complex, hazardous, or challenging to recreate in a physical test environment is one of the major advantages of FE analysis.
- The results generated by FEA software give a large range of situations to test against and are incredibly precise and thorough

VI. ACKNOWLEDGMENT

First and foremost I would like to express my special thanks to Dr. Mohammed Imran for his time and efforts he provided throughout the project work.

I also owe my sincere thanks to all the staff of Ghousia College of engineering for their support.

I convey my sincere thanks to Solid Works and ANSYS software.

I wish to thank my wife, Dr. Harini, who has stood by me, gave me love, support and help throughout my project work.

Finally, I would like to express my gratitude to everyone who assisted me in this research, either directly or indirectly.

REFERENCES

[1] AnuragSharma,DesignStudyofEndEffectorsISSN:2249–8958,Volume-4Issue-3, February2015.pp1-5

[2] P.Ferdinandlivijosephetal,Design,modelingandanalysisofindustrialRoboticarmminimizingweight and capital invested.June2020. pp 1-6

[3] Paul and Stevenson (1983)], [Huang and Milenkovic (1987)], [Treveljian et al. (1986)].Robot Wrist Configurations, Mechanisms and Kinematics, A. Romiti, T. Raparelli, M.SorliZ Department of Mechanics Politecnico di Torino, Italy 2 Department of industrialEngineeringUniversity of Cassino (FR),Italy.

[4] Ho Choi, Muammer Koc,"Design and feasibility tests of a flexible gripper based oninflatablerubberpockets",internationalJournalofMachineTools&Manufacture2006:46; 1350–1361.

[5] BiswalPneumatics handbook,by Antonybarber, 8th edition,1997.

[6] KiranKumarB Metal. DesignandDevelopment ofaRoboticArmon22March2019.

[7] AliMurtathaShuman etal. Modeling andControl of 6-axis Robot,December16, 2020

[8] Gob-Soon Kim, "Design of two-axis force sensor of robots finger", vol.3, No.1, March,2001.

[9] JianqiangWang"Intelligentgripperdesignandapplicationforautomatedpartrecognitionand gripping", October2002

[10] Anurag singh et al **"**Design and Static Analysis of Robotic Arm using Ansys", May 2020.

[11] Manikandan Ganesan et al **"**Implementing Industrial Robotics Arms For Material Holding Process In Industries", September 2021

[12] Tanmay Deshpande et al "Structural analysis of index finger assembly of the robot", 2021.

[13] Supriya Sahu et al "Static Analysis of a 6 - Axis Industrial Robot Using Finite Element Analysis", Volume 8, Issue 3, March 2017.

[14] Harihara. S et al "Static and Dynamic Analysis of Articulated Robotic Arm for Domestic Purposes", January 2023.

[15] Jeevan et al "Modeling and Analysis of Robot ARM using ANSYS", August-2015.

[16] Adrian Ghiorghe et al "Optimization Design For The Structure Of An RRR Type Industrial Robot", 2010