

Design and Analysis of Multifunctional Robot by using CATIA/PATRAN

Sruthi K V¹ Ramesh Kumar K R²

¹M.Tech Student ²Associate Professor

¹Department of Industrial Automation and Robotics ²Department of Mechanical Engineering

^{1,2}Srinivas Institute of Technology, Valachil, Mangalore, Karnataka, India

Abstract— In the real world, any continuous medium deforms under the application of force. Rigid body simulations do not capture such deformations and may lead to inaccurate results. As virtual prototype replaces physical prototype, tests can be carried out to validate the design of product. Virtual prototype of multifunctional robot is developed to simulate the system level working with live stress distribution, which is designed to lift a design payload. The design and analysis of a PC Controlled “Pick and Place” Robotic vehicle has been presented in this paper. The project of Design and Analysis of Robot Gripper for 500gm payload is divided into two parts. The main objective of the project is to design and analysis of Robot Gripper for 500gm Payload. In this project, design and analysis will rely heavily on design software, namely CATIA V5R19 and PATRAN for the analysis, simulation and animation of the gripper’s design.

Key words: Simulation, Virtual Prototype, Payload, Analysis

I. INTRODUCTION

The multifunctional robot consists of several parts namely base frame, vertical column, horizontal column, end effector, gripper and wheels. The design and drawing of a multifunctional robot is carried out in the software called CATIA V5R19 (Computer Aided Three-Dimensional Interactive Application). Inclusion of flexible bodies in MBD (Model Based Design) simulations accounts for flexibility. All individual parts of a multifunctional robot are designed with the suitable dimensions. Once all the parts of a multifunctional robot is developed in CATIA then it will be assembled and simulated. As virtual prototype replaces physical prototype, tests can be carried out to validate the design of product. Virtual prototype of multifunctional robot is developed with flexi body concept to simulate the system level working with live stress distribution, which is designed to lift a design payload of 500gm. The robot arm is converted in to flexi body as it is a critical component to validate the design of robot manipulator.

II. LITERATURE REVIEW

This chapter describes the literature review that is done to gain more information on this project. The beginning part of this chapter explains the history of Industrial Robots and types of robots that are available. This will be followed by information pertaining robot grippers or end effectors. Finally, information on gripper design consideration will be available at the end of this chapter.

An industrial robot is officially defined ISO as an automatically controlled, reprogrammable, multipurpose manipulator programmable in three or more axes. The modern concept of industrial robotic manipulators was only introduced in late 1950s by G. C. Devol (U.S. Patent

2988237).[1] and later joined by J. Engelberger.[4] to start up Unimation Inc. They are the originators of the first industrial robot by Unimation Inc. in the 1959. The first installation of the Unimate robot.[1]. for loading/unloading a die-casting machine at GM was in 1961.[2]. Early industrial robots were developed to perform operations in hostile environments such as inside radioactive chambers. Later, robots were applied to perform work in undesirable environments and in applications which were dull and monotonous. Today industrial robots can be found in almost all manufacturing applications, ranging from machine servicing to welding to painting. Usage of robots for pick and place applications is the fastest growing segment of robotic

There are many types of industrial robots that are being used in industries nowadays. A simple example is an industrial robot manipulator with six degrees of freedom. It has the features of a human chest, upper arm, forearm and wrists respectively. Therefore, it can be said that robots are designed to emulate human hand. The only difference is that it is much bigger compared to human hand. Same as humans, industrial robots needs some sort of device so that it can interact with the world around it in a form of grasping, manipulating parts and so on. This device is called robot gripper or sometimes called end effectors or end-of-arm tooling. End effectors should not be considered as accessories, but as a major component in any industrial robot application.

According to the Robotic Industry Association.[5], there are currently over 118,000 robots being used in the United States alone. The industry has seen significant growth over the past five years, having record sales in 1999 and 2000 with new unit sales exceeding one billion dollars in the year 2000. Since cost of grippers may be as high as 20% of a robot’s cost, depending on the application and part complexity no wonder robotic grippers are a \$ 31M industry. Engineers and designers are striving very hard to keep the cost of robotic grippers as low as possible. Therefore, developing a multipurpose gripper is not a good solution because it involves exaggerated cost. This is one of the main reasons why grippers are design for specific application only. This is to ensure that simple design can be produced, thus cutting the initial cost for an automation system.

III. METHODOLOGY

Step by step procedure followed in this work is listed below:

- 1) Requirement collection of the robot
- 2) Design/Drawing
- 3) Modelling of individual parts using CATIA software
- 4) Assembly of all the parts
- 5) Simulation
- 6) Analysis

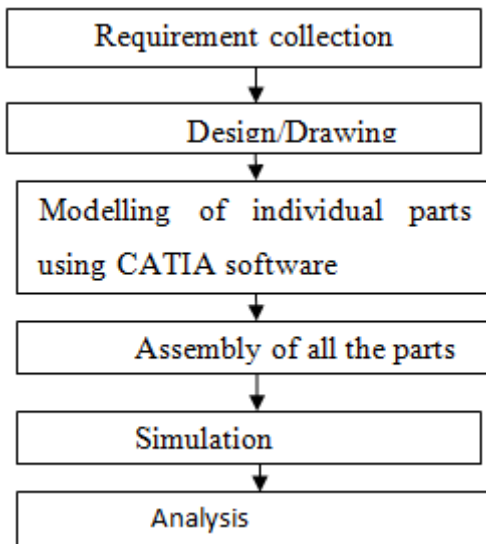


Fig. 1: Methodology

A. Requirement Collection of the Robot:

- 1) Number of axes.
- 2) Degrees of freedom.
- 3) Working envelope.
- 4) Kinematics.
- 5) Carrying capacity or payload.
- 6) Speed.
- 7) Acceleration.
- 8) Accuracy.
- 9) Repeatability.

B. Design and Drawing:

The two dimensional drawing and three dimensional CAD models of multifunctional robot is drawn using CATIA software. Design of all the parts such as Drawing of Base with L column, CAD model of Base with L column, Drawing of vertical column, CAD model of Vertical column, Drawing of Gripper, CAD model of Gripper, Drawing of Link assembly, CAD model of link assembly, Assembly of Multifunctional robot and virtual prototyping of robot are designed as shown in the figure.

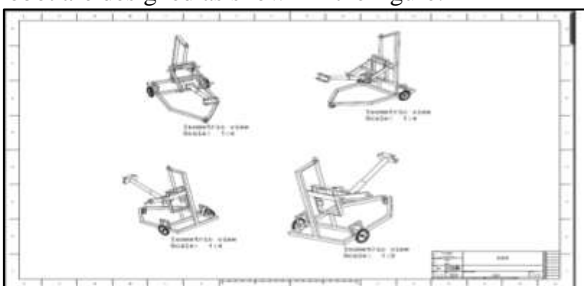


Fig. 2: Design and Drawing

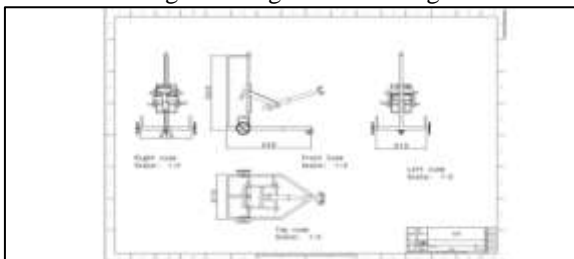


Fig. 3: Isometric View

C. Modelling of Individual Parts Using CATIA Software:

1) Base Frame:

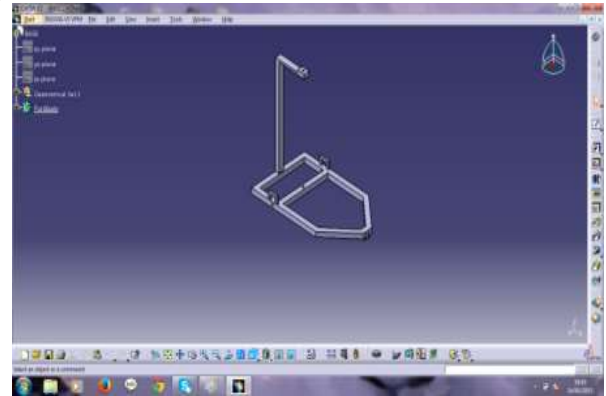


Fig. 4: Base Frame

The Base frame is designed with the help of CATIA which can be viewed in 3D. The Base frame of the multifunctional robot is the main component of entire robotic model.

2) Vertical Column:

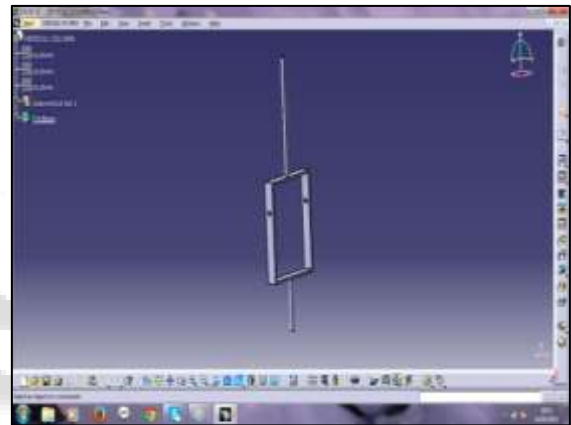


Fig. 5: Vertical Column

The 3D view of the vertical column is drawn using CATIA. The vertical column is attached with one end of the Base frame and the other end is been connected to topmost supporting column of the robot. The vertical column is simulated according to the degrees of freedom which we input.

3) Horizontal Column:

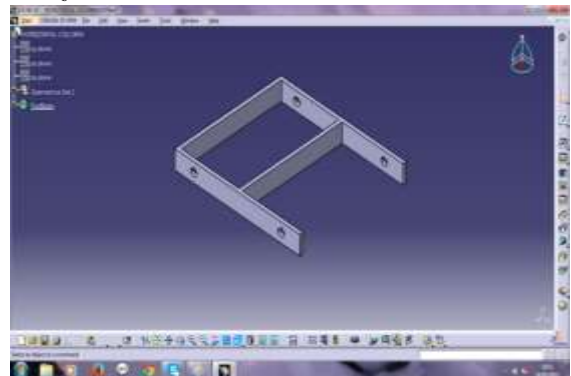


Fig. 6: Horizontal Column

The 3D view of the horizontal column is drawn using CATIA software. The horizontal column is an interface between the vertical column and end effector arm.

4) *End Effector Arm:*



Fig. 7: End Effector Arm

The end effector arm is drawn using CATIA software. The design of the end effector arm is suitable to pick a load of 500gm.

5) *Gripper:*

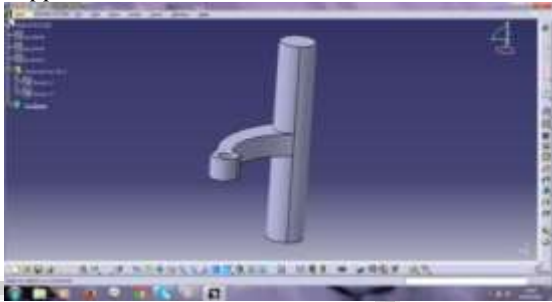


Fig. 8: Gripper

The 3D view of the gripper is drawn with the help of CATIA software. The grippers are the main components of the robot. The grippers are those which holds the material by opening and closing. The grippers does the mechanism of opening and closing which helps the gripper to hold the material.

6) *Front Wheel:*

The robotic model consists of three wheels one at the front and other two at the rear end. The front wheel will be freely moving. The front wheel is fixed to the Base frame and the provisions are provided to fix the wheels to it.



Fig. 9: Front Wheel

7) *Rear Wheel:*



Fig. 10: Rear Wheel

There are two rear wheels in this robotic model. The rear wheels are fixed to the Base frame.

8) *Assembly of All the Parts:*

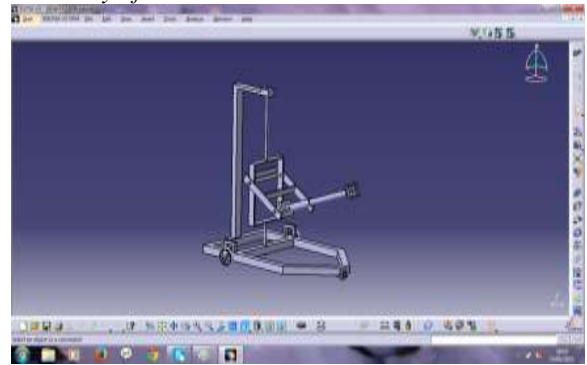


Fig. 11: Assembly

The assembly of the robotic model is been shown in the figure. Once all the parts of the robotic model is been designed and drawn with the help of CATIA software then we need to assemble it so that to form a robotic unit.

IV. SIMULATION

A. *Simulation of Vertical Column:*

In the figure shown the vertical column is highlighted.



Fig. 12: Vertical Column Highlighted

On clicking on the constraints of vertical column it can be simulated as per our requirements. Here we have taken 900 1800 and 2700 input value to simulate the vertical column.



Fig. 13: Constraints of Vertical Column

Above figure constraints of vertical column is highlighted

a)



Fig. 14: 90° Entered

Above figure degrees of freedom angles are entered ie 90°

b)

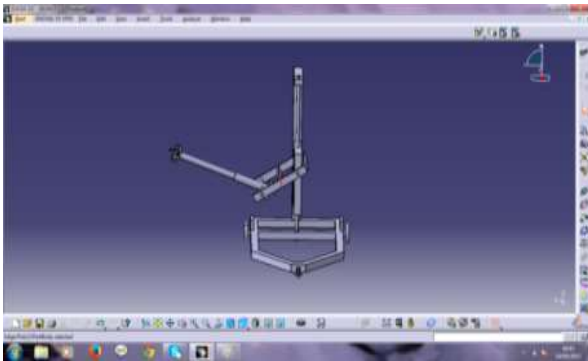


Fig. 15: 90° Moved

Above figure vertical column is been moved with 90°

c)

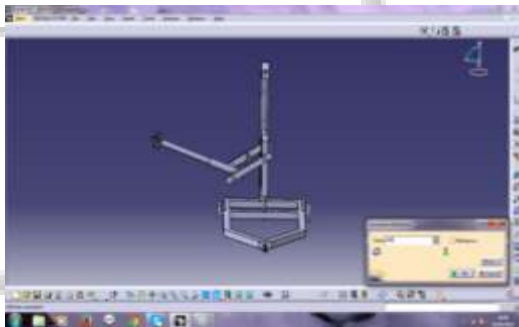


Fig. 16: 180° Entered

In the above figure degrees of freedom angle which is 180° is entered

d)

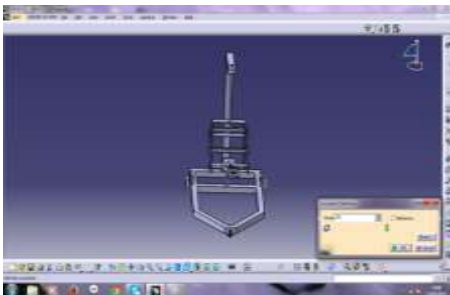


Fig. 17: 270° entered

In the above figure 270° entered

e)



Fig. 18: 270° Moved

Here 270° moved.

B. Simulation of Horizontal Column:

Horizontal column is highlighted



Fig. 19: Horizontal Column Highlighted



Fig. 20: Constraints of Horizontal

a)



Fig. 21: Angle 30° is Entered

Above figure degrees of freedom with an angle 30° is entered

b)



Fig. 22: 30° moved

c)



Fig. 23: Angle 60° is Entered

Above figure degrees of freedom with an angle 60° is entered

d)

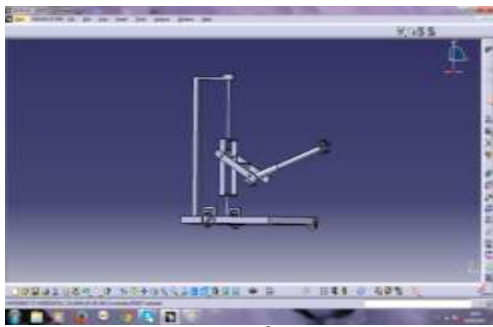


Fig. 24: 60⁰ Moved

e)

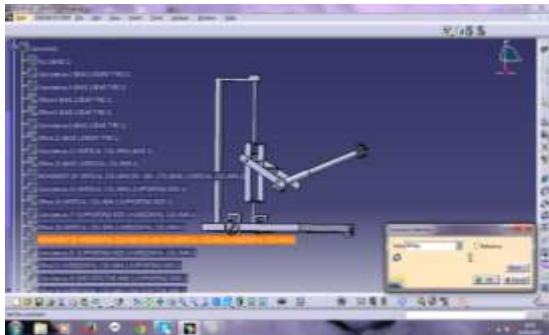


Fig. 25: 90⁰ Entered

d)

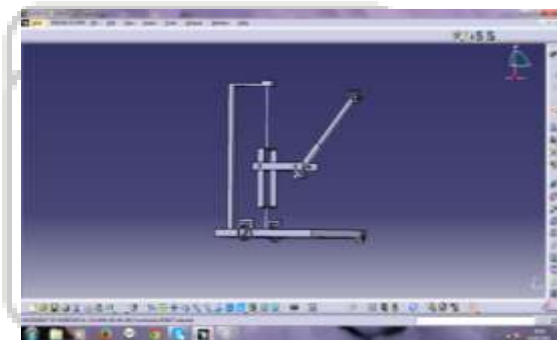


Fig. 26: 90⁰ Moved

C. Simulation of End Effector Arm:

End effector arm is highlighted



Fig. 27: End Effector Arm Highlighted

a)



Fig. 28: Constraints Highlighted with 90⁰

b)



Fig. 29: 90⁰ Moved

c)



Fig. 30: DOF with angle 130⁰ Entered

d)



Fig. 31: 130⁰ Moved

e)



Fig. 32: 170⁰ entered

e)



Fig. 33: 170⁰ moved

D. Simulation of Gripper:

a) Gripper1 is highlighted

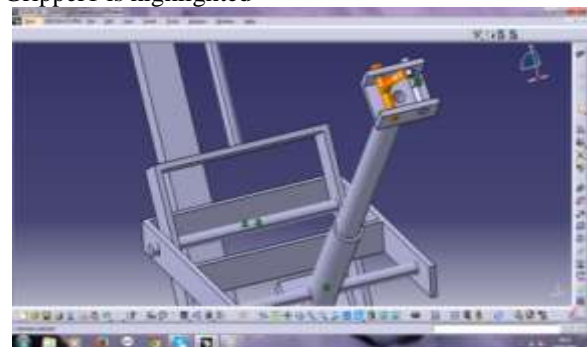


Fig. 34: Gripper1 is Highlighted

b) Gripper2 is highlighted

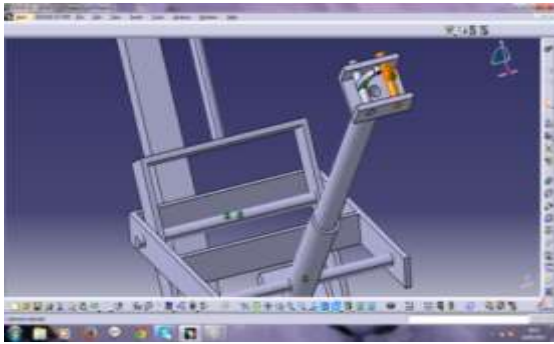


Fig. 35: Gripper2 is Highlighted

c) Gripper position with the gap of 1.5mm between two grippers (left and right)

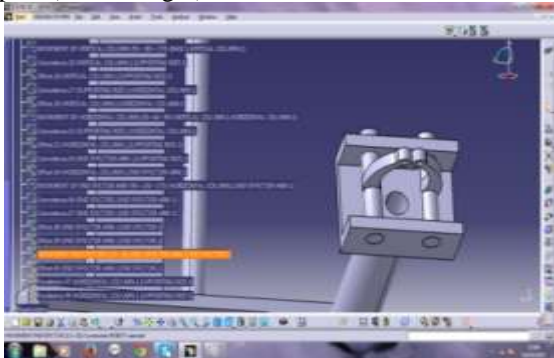


Fig. 36: Gripper Movement

d) Gripper movement after movement with the value 25mm

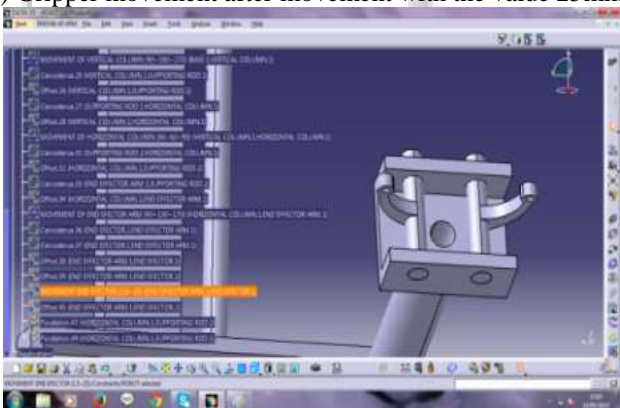


Fig. 37: Gripper Movement

V. ANALYSIS

A. Introduction of Finite Element Method:

Finite element method, sometimes referred as finite element analysis, is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Boundary value problem is mathematical problems in which one or more dependent variables must satisfy specific conditions on boundary of domain. Boundary value problems are sometimes called field problems. The field is the domain of interest and most often represents a physical structure.

B. Procedure of Finite Element Analysis:

1) Preprocessing

- Conversion of geometric model into finite element model.
- Material property data representation.
- Defining the boundary and loading conditions.

2) Processing:

- Solve the system of linear algebraic equation.
- Stresses are determined from the strains by hook's law.
- Strains are derived from the displacement functions within the element combined with Hooke's law.

3) Post Processing

- Output from finite element analysis in numerical form.
- It usually consists of nodal values of the field variables and its derivatives.

4) Meshing

A mesh is a network of line elements and interconnecting nodes used to model a structural system and numerically solve for its simulated behaviour under applied loading. Different element size can selected for different component cases.

5) Solid Mesh

Solid mesh is used for two dimensional components, when a dimension of two directions is greater than third direction than shall mesh can be used. Generally thick and thin components selected for solid meshing.

C. NASTRAN:

Nastran is software which is used for finite element analysis and computer aided engineering. It was invented by NASA. Nastran is used in the automotive, aerospace, and industrial products industries. Nastran also provides a good collection of multi-physics capabilities, such as coupled acoustic-structural, piezoelectric, and structural-pore capabilities, making it attractive for production-level simulations where multiple fields need to be coupled.

D. Linear Static Analysis:

In linear static analysis displacements, strains, stresses, and reaction forces under the effect of applied loads are calculated.

A series of assumptions are made with respect to a linear static analysis:

- 1) Small deflections
- 2) Material properties
- 3) Constant boundary conditions

E. Meshing of Robot for Linear Static Analysis:

Meshed model is done by using tetra mesh. In tetra mesh the thickness of the component is to be mentioned, so thickness of Robot is 1.6mm for steel.

F. Material Properties of ROBOT

Here we assumed steel and composite materials are isotropic in nature

Property	Mild steel
Density (Kg/m ³)	7.85g/cm ³
Modulus of elasticity(MPa)	2×10 ⁵
Poisson ratio	0.3
Tensile strength(MPa)	460

Table 1: Material properties

VI. RESULT AND DISCUSSIONS

A. Procedure for Analysis Using PATRAN Software:

Here in this project we have considered the weight of the motor, weight of the individual parts of robot and also counter weight. A node is created just above the robotic model and all the weights are added through the node on the desired parts of the robotic model as shown in the fig

Here the weights of different parts of robotic model is listed below,

- 1) Weight of the gripper - 187gm
- 2) Weight of the end effector arm – 1.5kg
- 3) Weight of the vertical column – 1.9kg
- 4) Counter weight – 8kg

In order to avoid the end effector arm free fall when the gripper picks up load of 500gm we need to add the counter weight on the other side which is fixed at the back end of horizontal column. The counter weight acts as a balancing the end effector arm from a free fall.

Considering all the weights then we import the files to PATRAN and NASTRAN software

First we develop the model in CATIA software. We save it in the IGES format. This file will be imported to PATRAN software for Finite Element analysis. In Finite element analysis we can perform meshing, apply loads, material properties, boundary conditions, add counter weight and motor weight. Once it is been done then the BDF file will be transferred to NASTRAN software for getting the results. The results includes files such as FO6, FO4,XDB,LOG which are read only file again sent back to PATRAN where we can get the results such as von mises stress, deformation, axial stress and bending stress.

B. NASTRAN Software files:

When we run the BDF files in NASTRAN we get the results to read in four different files namely, robot.f04, robot.f06, robot.xdb and robot.log.

Robot.log files fetches the information about how much memory the robotic model occupies, timings of execution and system performance etc as shown in the following figures.

C. Steps for Applying Load and Analysis:

1) Step 1- Tetmesh:

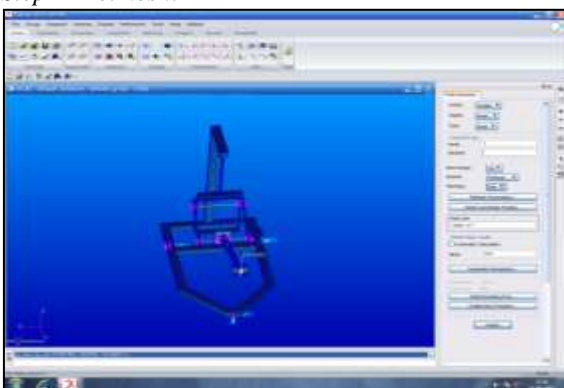


Fig. 39: Tetmesh

2) Step-2 Boundary Conditions:

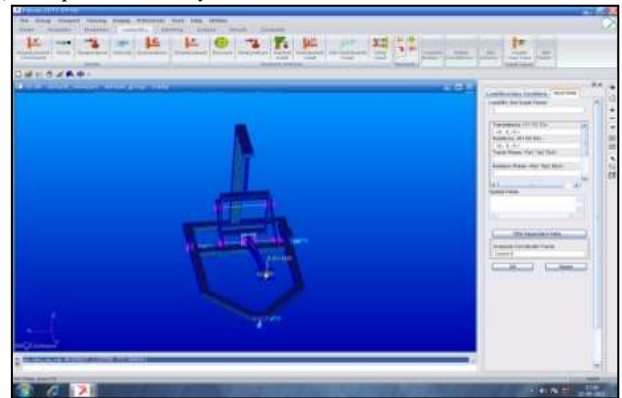


Fig. 40: Boundary Conditions

Every model has 3 translational and 3 rotational degrees of freedom. Here 3 translational motion is with respect to x, y and z and also 3 rotational motion with respect to RX, RY and RZ which needed to be fixed for meshing as shown in figure.

3) Step-3 Application of Load

Here we can input the weights and counter weight load and also can mention the boundary conditions on various parts of a robotic model as shown in figure.

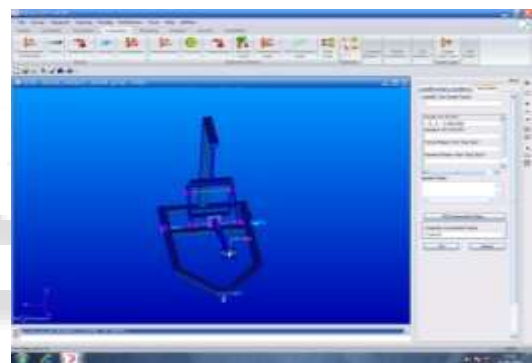


Fig. 41: Application of Load

Here $x=0, y=0$ and $z=-4.99N$ because the force applied on the gripper part is acting downward which is in 'z' direction.

4) Step-4 Material Properties:

In this we can assign the youngs modulus value, poisson ratio and also density of steel.

Here in this project we have assumed the value of youngs modulus as $2.5E^{-5} N/MM^2$ and poisson ratio as 0.33 and density of steel as $7.8E^{-3} \text{ tonn}/MM^3$

5) Step-5 MPC (Multipoint constraints):



Fig. 42: MPC

With the help of MPC we can create a node and connect all the weights of robots of different parts onto that single node.

In between MPC we have to connect bar element also.



Fig. 43: MPC with Bar Element

6) Step-6 Counter weight and motor weight



Fig. 44: Counter Weight and Motor Weight

7) Step-7 BDF file creation for results

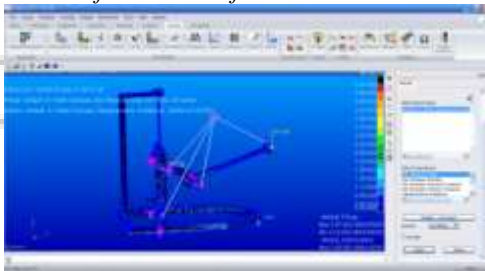


Fig. 45: BDF File Creation

For the results we need to have the BDF file from the linear static model of the robot. Once we get the BDF file from this software we can send this file to NASTRAN software for getting the results.

The results are obtained and fetched in read only format which can be read easily.

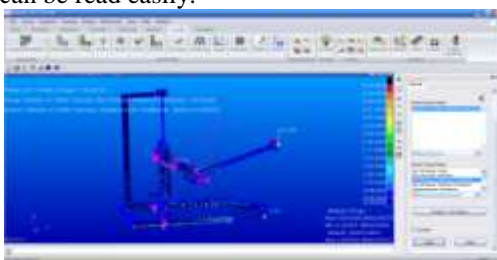


Fig. 46: BDF File Creation

8) Step-8 Deformation

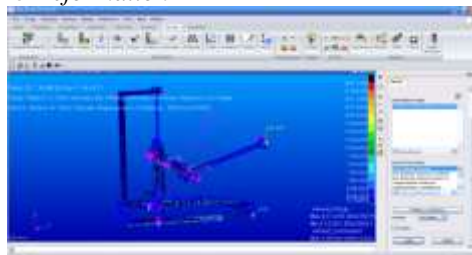


Fig. 47: Deformation

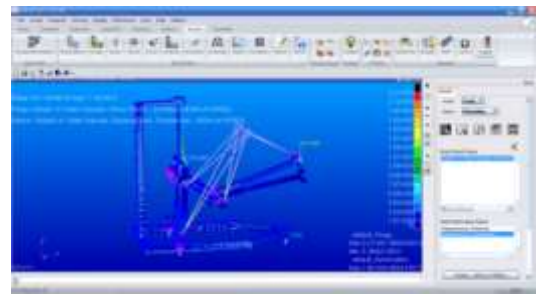


Fig. 48: Deformation

9) Step 9- Vonmises Stress

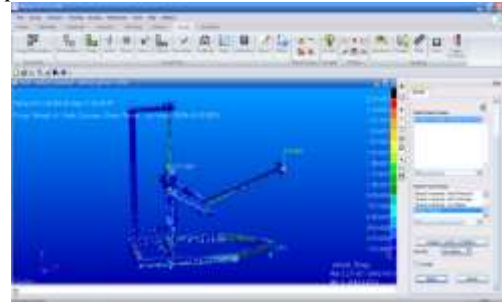


Fig. 49: Vonmises Stress

So the Vonmises stress we are getting is totally 22.7 N/MM^2 as shown in figure which is more than yield stress of a steel.

VII. CONCLUSION

In this project the different parts of the robotic model is been designed and assembled. The robotic model is been simulated using CATIA software. Analysis of the robotic model is done using PATRAN and NASTRAN software. The results which are shown here is of mainly vonmises stress which is found that 22.7 N/MM^2 which is not suitable for the fabrication process and to have the design fabricated the counter weight has to be lowered which should be less than 8kg.

REFERENCES

- [1] Rosen, Rebecca j. "Unimate: The Story of George Devol and the First Robotic Arm."
- [2] 1961: The First Robot.
- [3] Nof, Shimon Y. (1999). Handbook of Industrial Robotics (2nd Edition ed.). John Wiley & Sons. Pp. 3-5. ISBN 0-471-17783-0.
- [4] "The Top 50 inventions of the Past 50 Years." Popular Mechanics. Retrieved 2012-03-15.
- [5] Pearce, Jeremy. "George C .Devol, Inventor of Robot Arm, Dies at 99", The New York Times, August 15, 2011. Accessed August 14, 2011.