DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

EVALUATION OF THE STRENGTH OF A THREE AXIS SERIAL ROBOT PERFORMING A TASK BY INTEGRATED DYNAMIC ANALYSIS

by
Armin AMINDARI

January, 2012
İZMİR

EVALUATION OF THE STRENGTH OF A THREE AXIS SERIAL ROBOT PERFORMING A TASK BY INTEGRATED DYNAMIC ANALYSIS

A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of Dokuz Eylül University
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Mechanical Engineering, Machine Theory and Dynamics Program

by
Armin AMINDARI

January, 2012
İZMİR

M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled **"EVALUATION OF THE STRENGTH OF A THREE AXIS SERIAL ROBOT PERFORMING A TASK BY INTEGRATED DYNAMIC ANALYSIS"** completed by **ARMIN AMINDARI** under supervision of **PROF. DR. HİRA KARAGÜLLE** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Prof. Dr. Hira KARAGÜLLE

Supervisor

Asst. Prof. Ahmet ÖZKURT

(Jury Member)

Asst. Prof. Levent ÇETİN

(Jury Member)

Prof. Dr. Mustafa SABUNCU

Director

Graduate School of Natural and Applied Sciences

ACKNOWLEDGMENTS

I would like to express my sincere indebtedness and gratitude to my thesis consultant Prof. Dr. Hira KARAGÜLLE, for all his time and effort. His guidance and input at every stage of my work truly helped me navigate this endeavor.

I would also thank Dr. Murat AKDAG (Engineering Faculty of Dokuz Eylül University) ,my friends Şahin YAVUZ and Serkan GÜLER for their encouragement and intellectual input during the entire course of this thesis.

Finally, I would like to thank my parents and my sister who have supported me all the way since the beginning of my studies. Their love gave me forces to make this work.

Armin AMINDARI

EVALUATION OF THE STRENGTH OF A THREE AXIS SERIAL ROBOT PERFORMING A TASK BY INTEGRATED DYNAMIC ANALYSIS

ABSTRACT

Three axis serial robots with different sizes are widely used for pick and place, welding and various operations in industry. Developments in mechatronics, which is the synergistic integration of mechanism, electronics and computer control to achieve a functional system, offer effective solutions for the design of such robots. The mechatronic design process involves solid modeling, assembly, rigid body dynamics, finite element rigidity analysis, motion control and computer programming. The integrated dynamic analysis of robots is usually used in design stage. In this study, it is offered that it can also be used in the application stage.

In this study SolidWorks, CosmosMotion, ABAQUS is used with an integrated approach. The integration software is developed in VisualBasic by using the application programming interface (API) capabilities of these programs. ABB-IRB1400 industrial robot is considered for the study. Different trajectories are considered. Each task is evaluated by kinematic analysis first. If the task is out of the workspace then the task is canceled. This evaluation can be done also by robot programs like RobotStudio. It is offered in this study that the task must be evaluated by considering the limits for velocities, motor actuation torques, reaction forces, natural frequencies, displacements and stresses due to the flexibility. The results are shown on kinematic, kinetic and rigidity evaluation charts. The results of this work can be used for optimal usage of robots.

Keywords: Robot programming, dynamic analysis, trajectory planning

BİR İŞ YAPAN ÜÇ EKSENLİ SERİ BİR ROBOTUN ENTEGRE DİNAMİK ANALİZ İLE MUKAVEMETİNİN DEĞERLENDİRİLMESİ

ÖZ

Farklı boyutlu üç eksenli seri robotlar sanayide taşıma, kaynak ve farklı işlemler için yaygın olarak kullanılmaktadır. Mekanik, elektronik ve bilgisayar kontrolünün sinerjik entegrasyonu olan mekatronikteki gelişmeler, bu tür robotların tasarımı için etkin çözümler sunmaktadır. Mekatronik tasarım süreci, katı modelleme, montaj, katı cisim dinamiği, sonlu eleman analizi, hareket kontrolü ve bilgisayar programlama gerektirir. Bu çalışmada, robotların entegre dinamik analizinin sadece tasarım aşamasında değil, uygulama aşamasında da kullanılması önerilir.

Bu çalışmada, SolidWorks, CosmosMotion, ABAQUS entegre bir yaklaşım ile kullanılır. Bu programların uygulama programlama ara yüzü (API) özellikleri kullanılarak VisualBasic ortamında entegrasyon yazılımı geliştirilmiştir. ABB-IRB1400 endüstriyel robot analiz için ele alınmıştır. Farklı yörüngeler analiz edilir. Önce Her yörüngenin kinematik analizi değerlendirilir. Yörünge, çalışma alanının dışında ise görev iptal edilir. Çalışma alanı değerlendirmesi RobotStudio gibi robot programlar kullanarak yapılabilir. Bu çalışmada robotun yaptığı işin, hız limitleri, motor çalışma torkları, reaksiyon kuvvetleri, doğal frekans analizleri, esneklik nedeniyle oluşan deplasman ve gerilmeler dikkate alınarak değerlendirilmesi önerilir. Sonuçlar kinematik, kinetik ve rijitlik değerlendirme grafiklerinde gösterilmiştir. Bu çalışmanın sonuçları, robotların optimum kullanımı için kullanılabilir.

Anahtar sözcükler: Robot programlama, dinamik analiz, yörünge planlama

CONTENTS

Page

CHAPTER ONE
INTRODUCTION

A robot is a reprogrammable multifunctional manipulator designed to move material, parts, tools, or specialized devices through variable programmed motions for the performance of a variety of tasks (Robot Institute of America).

Robotics studies, attempt to minimize the behavior of human function by the use of revolute joints, sensors, actuators, controllers and computers. There is much research being pursued in different aspects of the robotics field. The major areas include manipulator design, obstacle avoidance, artificial intelligence, trajectory planning, and computer vision. This thesis work focuses on dynamic analysis using integrated software which connects the last two major areas of robotics fields.

1.1 Literature survey

A significant amount of research has been reported concerning trajectory planning in robot manipulators (D.E Whitney,1969). Most of them are based on the calculation of inverse kinematics employing a pseudo-inverse of the Jacobian matrix. Trajectory planning using the minimal-time criterion was proposed under the B-Spline assumption of the Cartesian path (J.E Borrow,1988). A new method for time-optimal motion planning based on improved genetic algorithms was proposed, which incorporates kinematics constraints, dynamics constraints and control constraints of the robotic manipulator (W.M.YUN &Y.G Xi,1990).

Pires and Machado proposed an evolutionary method which optimizes the robot structure and the required manipulating trajectories. They described how an optimal manipulator minimizes both the path trajectory length and the ripple in the time evolution, without any collision with the obstacles in the workspace.

In recent years the concepts of dynamics and control of manipulators have gained tremendous attention from a number of researchers. The emphasis of research in the area of the manipulators has been on improvement of the generality of mathematical models and the formulation of equations of motions that are amenable to computer solutions and real-time controls for the same. Pasic, Williams and Hui proposed a new solution for the nonlinear differential equations of any order for multibody dynamics and control problems. A planar 2-DOF manipulator is considered for study to validate the algorithm against the popular shooting method.

In the area of robotics, one of the major subjects of research is to evaluate the kinetic and rigidity parameters of the robot during the path which has a great roll in view of life span of the robot. Wang and Ravani showed that the maximum allowable load of axed base manipulator on a given trajectory is primarily constrained by the joint actuator torque and its velocity characteristic (Wang& Ravani,1988). Korayem and Ghariblu determined the maximum allowable load of wheeled mobile manipulators for a desired trajectory. Yue computed the maximum payload of kinematically redundant manipulators using a finite element method for describing the dynamics of a system. Korayem and Shokri developed an algorithm for finding the MADL of the 6-UPS Stewart platform manipulator. Korayem and Heidari presented a general formula for finding the maximum allowable dynamic load of flexible link mobile manipulators. The main constraints used for the proposed algorithm are the actuator torque capacity and the limited error bound for the end-effector during motion on a given trajectory.

The dynamic stress of elastic mechanisms or flexible robots has been studied by a few researchers.Zhaocai studied the dynamic stress of the flexible beam element of planar flexible manipulators.Considering the effects of bending-shearing strain and tensile compression strain, the dynamic stress of the links and its position are derived by using the Kineto-Elastodynamics and the Timoshenko beam theory.An integrated design process presented by Karagülle and applied to a hexapod robot whichcan be used for medical operations.

1.2 Thesis organization

In this study the evaluation of the kinetic parameters is achieved with a main program developed in Visual Basic, which uses the application programming interface (API) capabilities.

Chapter II introduces the history of industrial robots followed by classification of the robots and robot softwares used in industry.

Chapter III explains the flowchart of the evaluation and modeling of the robot.Finite element analysis, inverse kinematics and forward kinetic analysis of the robot is discussed .A sample path is evaluated in the later parts of the chapter.

Chapter IV presents various simulation results for different cases.

Chapter V presents experimental results of the modal testing of the robot.

CHAPTER TWO
INDUSTRIAL ROBOTS

2.1 History of industrial robot

Visions and inventions of robots can be traced back to ancient Greece. In about 322 BC the philosopher Aristotle wrote: "If every tool, when ordered, or even of its own accord, could do the work that benefits it, then there would be no need either of apprentices for the master workers or of slaves for the lords." Aristotle seems to hint at the comfort such'tools' could provide to humans. In 1495 Leonardo da Vinci designed a mechanical device that resembled an armored knight, whose internal mechanisms were designed tomove the device as if controlled by a real person hidden inside the structure. Theterm 'robot' was introduced centuries later by the Czech writer Karel Capek in his play R. U. R. (Rossum's Universal Robots), premiered in Prague in 1921. 'Robot' derives from the Czech 'robota', meaning forced labor, and 'robotnik,' a slave or servant. Isaac Asimov, the ingenious science fiction author, is generally credited with the popularization of the term 'robotics.' He used it in 1941 to describe the study of robotsand predicted the rise of a powerful robot industry. The term was first published in his short story 'Runaround' in 1942, and then in 1950 in the collection I, Robot, which also introduced his famous Three Laws of Robotics (Asimov, 1950).

Denavit and Hartenberg (1955) applied homogeneous transformations for modelingthe kinematics of robotic manipulators. The advent of automated flexible manufacturingsystems (FMS) in the 1960s established robotics as a scientific discipline. The primary objectives for FMS are reduced labor costs, a high product mix, and factory utilization near factory capacity. A typical FMS combines industrial robots, an automated warehouse, automated material handling, and complex software systems for simultaneouslymodeling, operating, and monitoring the factory. Industrial robots are a critical factor inthis strategy that minimizes the role of human labor, allowing rapid changes to assemblylines, avoiding costly equipment replacements, and enabling the economical production of customized lots.

2.2The global robotics industry

The International Federation of Roboticsestimates that the worldwide operational stock of industrial robots had reached almostone million in 2007 (figure 2.1). This number is estimated to increase to almost 1.2million by 2011. Estimates depend on the assumed average service life, typicallybetween 12 and 15 years, and do not include about 550 000 older robots that had alreadybeen decommissioned.

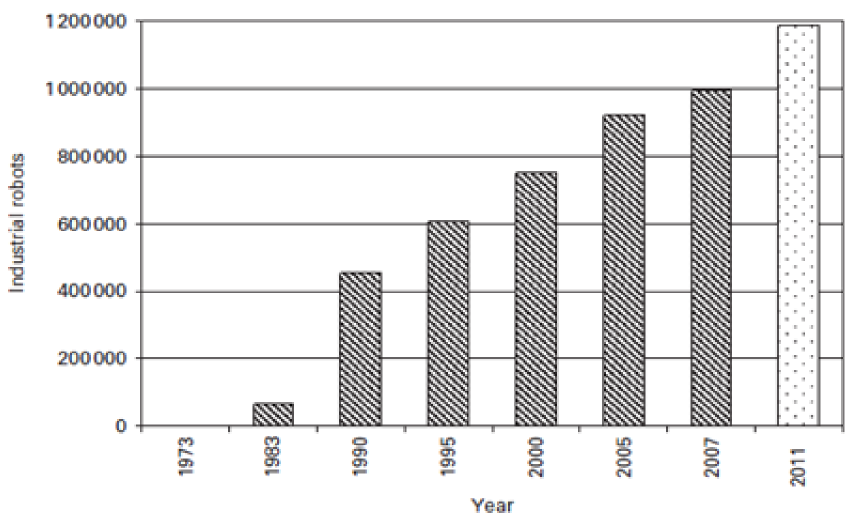


Figure 2.1Operational stocks of industrial robots (World Robotics 2008)

This success can be attributeto three characteristics ofindustrial robots:

- Industrial robots are programmable automation devices and are, as a consequence, flexible and versatile (unlike special-purpose automated machines).
- Industrial robots exceed the physical and mechanical abilities of humans during extended work periods and in uncomfortable or hazardous environments.
- Industrial robots perform with high fidelity and accuracy and in compliance with their programmed instructions.

2.3 Standard robot types

The main characteristics of industrial robots are:

- Number of axes of motion
- Kinematic structure
- Work envelope
- Maximum payload
- Maximum speed
- Accuracy
- Drive train (actuators, remote vs. direct-drive).

Eight robot types are defined in the ISO standard 8373:1994 based on their kinematicstructure and the coordinate frame that spans the workspace, Cartesian robots, cylindrical robots, spherical(or polar) robots, pendular robots, articulated (or anthropomorphic)robots, SCARA robots, spine robotsand parallel robots. The preferred kinematic structure of industrial robots depends on the application at hand and is influenced by the required motion, payload, end-effector orientation, and other factors.

Figure 2.2Scararobot (Toshiba MachineTH650 A)

Figure2.3Articulated robot (C3 Series,Epson)

Figure 2.4 Cartesian robot (Lexium MAX, Schneider Electric)

2.3.1 Cartesian robots

Cartesian robot is having three area of movement which is also called Gantry robot. Cartesian/Gantry robot is having three axes for controlling which are linear axes.

Advantages:

Linear motion in three dimensions

Simple kinematic model

Rigid structure

Easy to visualize

Can use inexpensive pneumatic

Drives for pick and place operation

Disadvantages:

Requires a large volume to operate in

Work space is smaller than robot volume

Unable to reach areas under objects

Guiding surfaces of prismatic joints

Must be covered to prevent ingress of dust

Applications:

Controlling cylindrical coordinate system

Handling at machine tools

Gearing and belt system

Assembly operations

Grinders and grinding machines

Spot welding

Wind Tunnel

die-casting machines

2.3.2Cylindrical robots or spherical robots

A robot, which is used to control cylindrical mechanism and cylindrical coordinate system is called cylindrical robot. Cylindrical robot can work in four degree mechanism.

Advantages
Simple kinematic model
Easy to visualize
Good access into cavities and machine openings
Very powerful when hydraulic drives used

Disadvantages
Restricted work space
Prismatic guides difficult to seal from dust and liquids
Back of robot can overlap work volume

Figure2.5 Cylindricalrobot(R19 ATHENA series, ST ROBOTICS)

2.3.3Articulated robots

An articulated robot is a robot with rotary joints. Articulated robots can be used to lift small parts with great accuracy. They are also known as a jointed-arm.

The articulated robot arm has a trunk, shoulder, upper arm, forearm, and wrist.With the ability to rotate all the joints, a majority of these robots have six degrees of freedom.

- Axis 1 - Arm sweeps from side to side.
- Axis 2 - Shoulder moves forward and backward.
- Axis 3 - Elbow moves up and down.
- Axis 4 - Middle of forearm pivots up and down.
- Axis 5 - Wrist moves up and down.
- Axis 6 - Wrist sweeps from side to side.

Movement is made in three ways: pitch is up and down movement, yaw is right and left movement, and roll is rotation. This mobility allows articulated robots to be used for tasks such as welding, painting, and assembly.

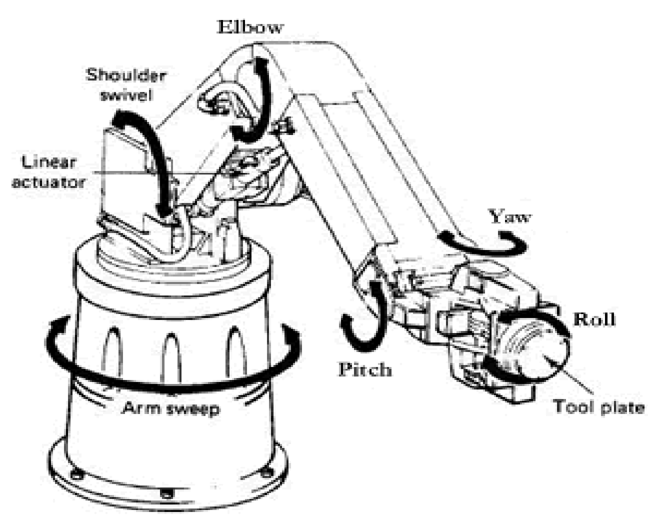


Figure2.6 Axes of articulated robot (Robotmatrix.org)

Advantages:

- maximum flexibility
- covers a large work space relative to volume of robots
- revolute joints are easy to seal
- suits electric motors
- can reach over and under objects

Disadvantages

- complex kinematics
- difficult to visualize
- control of linear motion is difficult
- structure not very rigid at full reach

Application areas

- Die-casting
- carrier robot
- gas welding
- arc welding
- Synchronous Positioning Control Technique
- fettling machines
- assembly operations
- spray painting

2.4Robot programing

Traditionally a robot program is generated by teaching the robot one point at a time using a teach pendant (online programming). This process is time consuming and also errors can come out during the teaching process. In order to overcome these limitations, off-line programming systems can be used. These kinds of systems are mainly based on Virtual Reality (VR) technologies.The restrictions encounteredrequire the necessity to create prior simulation of a virtual world which has to integrate all the information related to the real working environment.

Therefore, the applications and operational environments are limited to those in which a virtual model of the working environment can be reconstructed.

2.4.1 Methodology for industrial robots off-line programming

The methodology to generate robot softwares involves the next steps:

• Modeling of the virtual robot 3D mode
• 3D virtual robot model data conversion
• Development of the inverse kinematics robot mode
• Simulation of the virtual robot actions by using interaction devices (for example a keyboard) and generation of robot instruction by storage of robot control points.

Robot program is the coded commands that tell a mechanical device (known as a robot) what tasks to perform and control its actions. Robot software is used to perform tasks and automate tasks to be performed. Programming robots is a non-trivial task. Many software systems and frameworks have been proposed to make programming robots easier such as RobotStudio which is developed by ABB™ Robotics Company.

2.4.2 RobotStudio

RobotStudio is a simulation and offline programming software. The software, allows robot programming to be done on a PC in the office without shutting down production. It also enables robot programs to be prepared in advance, increasing overall productivity.

RobotStudio provides the tools to increase the profitability of robot system by letting operator, perform tasks such as training, programming, and optimization without disturbing production. RobotStudio is built on the ABB™Virtual Controller, an exact copy of the real software that runs robots in production. It thus allows very realistic simulations to be performed, using real robot programs and configuration files identical to those used on the shop floor.

Two options are available, offline and online programming. The term online programming can be defined as, programming when connected to the control module. This expression is also implies using the robot to create position and motion. The term offline programming can be describes as, programming without being connected to the robot or control module.Offline programing is possible with RAPID language which is written in text document figure 2.7.Robotstudio simulates the program in PC and transfers to robot controller.

```
MODULE MainModule
                    CONST
        jointtargetjpos10:=[[0,0,0,0,0,60],[9E+09,9E+09,9E+09,9E+09,9E+09,9E+09]];
CONST jointtarget
jpos40:=[[30,0,0,0,0,0],[9E+09,9E+09,9E+09,9E+09,9E+09,9E+09]];
CONST speeddata speed1:=[1,10,1,1];
CONST speeddata speed2:=[1,1,1,1];
  PROC main()
           MoveAbsJ jpos10\NoEOffs, speed1/T:=60, z50, tool0;
           MoveAbsJ jpos40\NoEOffs, speed2/T:=0.1, z50, tool0;
ENDPROC
```

Figure 2.7 Sample of rapid code for ABB™ industrial robots

CHAPTER THREE
INTEGRATED DYNAMIC ANALYSIS

3.1 Flow chart

In this section, developed integrated dynamic analysis method for strength evaluation of mechatronic systems developed by Dokuz Eylul University BATUL Laboratory is presented. The BATUL integrated dynamic analysis method (BIDAM)which uses different engineering software with application programming interface(API) capabilities and integrationsoftware (IS), has been developed. SolidWorks is used for solid modeling and assembly, CosmosMotion for the rigid body dynamics, ABAQUS for the vibration andstress analysis. The motion of robot controlled with robot studio. The flow chart of studyis described in figure 3.1.

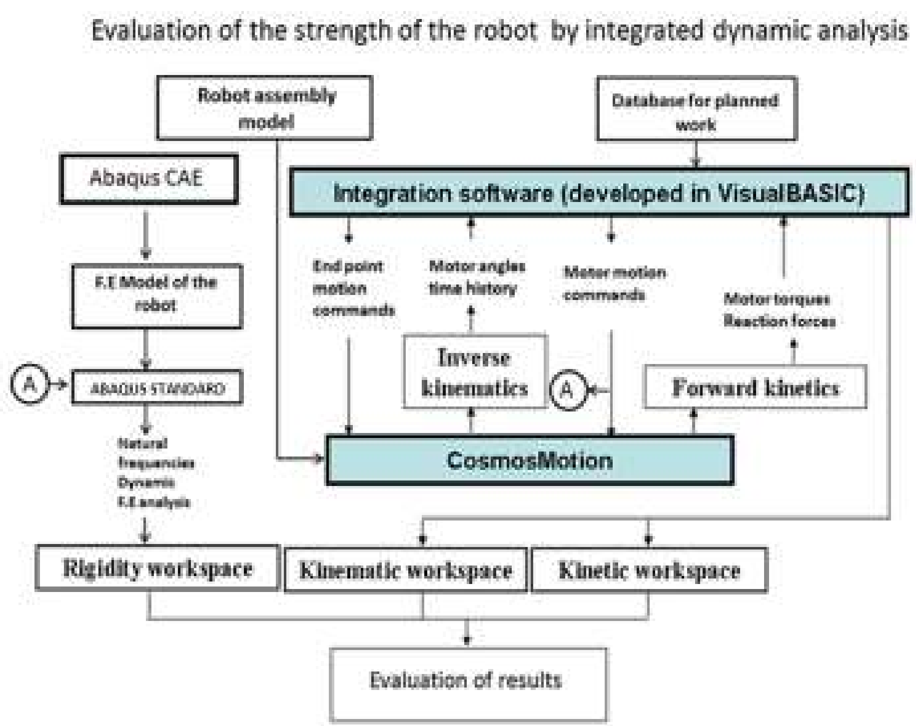


Figure 3.1 Flow chart of integrated dynamic analysis process in this study

Thedetails of the software to achieve the tasksin the flow chart can be found in the manuals.This flow-chart is applied toABB™ IRQ 1400 industrial robot in this study.The database of parts of the robot and the information on technicalspecifications such as bearings, motors, and gears etc. are limited. For the analysis, the specifications are considered as the same as the specification of the parts commonly used in industrial applications. The database for planned work consists of the information on the end-point motion inputs.

3.2BIDAM methodology

SolidWorks + CosmosMotion +Abaqus package is used in this study. Solid modeling of parts andassembly can are done using the Graphical User Interface (GUI) capabilities ofSolidWorks.

The integration software (IS) developed in Visual Basic has “Inverse Kinematics” and “Forward Kinetics”.IS reads the database for theplanned work and sends the end-point motion commands to CosmosMotion using API, when“Inverse Kinematics” option is clicked. The motor motions are assigned as free at this stepand the motorangles are found. The time histories of the motorangles aretransferred back to IS. The results are evaluated whether the motion is possible considering thekinematic limits (angle and velocity limits) of the motors.

When “Forward Kinetics”option is clicked the time histories of the motor motion generator torques and reactionforces in bearings are found. The results are evaluated whether the kinetic limits (the motortorques and bearing force limits) are exceeded or not.After the rigid body dynamics analyses are done, the finite element (FE) natural frequencystatic and dynamic analyses are performed in Abaqus.

Abaqusdoes not have API capabilities in the software package used in this study and these analysesare performed using Abaqus CAE and python scripting. “Rigidity Workspace Evaluation” option in IS is inserted as areminder. The lowest natural frequency (fmin), maximum displacement (umax) and maximumVon Mises stress (smax) are evaluated whether if the rigidity limits of the robot structure areexceeded.

The higher values of fmin and the lower values of umax and smax indicate higher staticand dynamic rigidity. The precision of the work performed by the robot is related to theserigidity values.

The database for planned works are arranged for various works and simulations areperformed for the kinematic, kinetic and rigidity workspace evaluations. The databases for paths are changed and the simulations and evaluations are repeated until a final trajectory isselected.

The parameters considered in the evaluation are reachability, maximum angular velocity limits, maximum forces in the bearings, maximum torques generated by motors, maximum von mises stress (S_{max}), maximum displacement and the lowest natural frequency of the robot during the path Table 3.1.

Table 3.1 Parameters considered in evaluation

Inverse kinematics	Reachability Velocity limits
Forward kinetics	Motor torques Forces in bearings
Rigidity	Lowest natural frequency Maximum displacement Maximum von mises stress

Limits of the parameters are defined in a text document. Due to some unknown specifications about the robot, some limits are referenced according to similar parts available in market. The limits are listed below:

Angle limit for motor 1: 90 degrees
Angle limit for motor 2: 80 degrees
Angle limit for motor 3: 80 degrees

Velocity limit for motor 1: 100 deg/sec

Velocity limit for motor 2: 100 deg/see

Velocity limit for motor 3: 100 deg/sec

Torque limit for motor 1: 300 N.m

Torque limit for motor 2: 300 N.m

Torque limit for motor 3: 300 N.m

Reaction Force limit for bearing 1: 5000 N

Reaction Force limit for bearing 2: 5000 N

Reaction Force limit for bearing 3: 5000 N

Maximum displacement: 250 μ

Maximum von mises stress: 30 Mpa

The database for the path is written in text document figure 3.2.

```
Path 1a
nsim,100
0,0,0
inverse
0,0,0
0,2,0,163.3,-500
3,7,0,0,1000
-1
```

Figure 3.2 Database for planed work

This database contains information about start and end point coordinates, simulation time and number of simulations in CosmosMotion. The database shown in figure 3.7 indicates that the number of simulation is 100, first step incremental displacement of the end point is$\{0,163.3,-500\}$, time is $\mathbf{t_i}$= [0, 2] and in the second step incremental displacement of the end point will $\{0,0,1000\}$ and the time will $\mathbf{t_i}$= [3, 7]. The robot will not move in $\mathbf{t_i}$= [2, 3].

3.3 Assembly model of robot

The model of the robot with un-detailed parts is shown in figure 3.3. The model of the robot assembly consists of motion subassemblieswhich are called as members.

Each member consists of models of parts or subassemblieswhich have a common rigid body motion. Members are connected by joints. Themodels of the members with un-detailed parts are shown in figure3.4.

Figure 3.3Model of the robot with un-detailed parts

The robot can be modeled by using GUI, after starting the recording of Visual Basic macros in SolidWorks then, the macros can be examined to determine the commands to develop codes for the modeling software, which uses API. The software performs the following tasks with sub-routines. In this study robot is modeled in SolidWorks GUI.

First, the un-detailed parts are modeled using the specifications of the parts. Then the parts are insertedto the assembly file for the member to be modeled. The parts are located to their positions. After modeling all the members,the assembly files of the members are inserted to the assembly file of the robot. The mates are definedbetween the members for joints. The model is the assembly of 4 members.

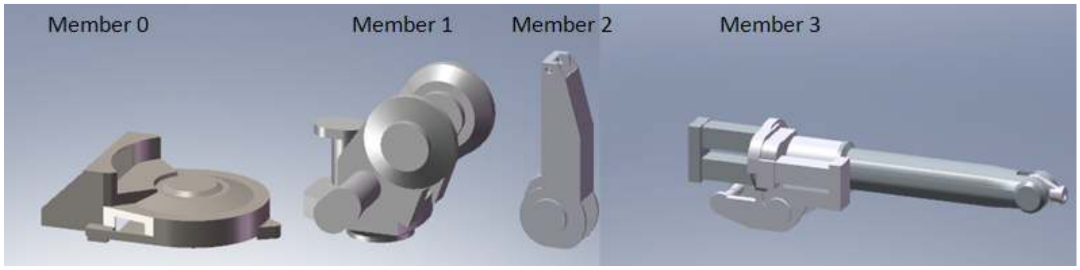


Figure 3.4Members of the robot

Reference planes and axes are created on the relevant partsand these reference objects are mated in SolidWorks to createthe revolute joints in CosmosMotion automatically. Therevolute joints at the motor output shaft bearings are named asmotor1, motor2, and motor3 for the axes 1, 2, and 3respectively.The payload is attached to Member-3 at the end-point.The origin of the global coordinates is at the center of thecircular face at the bottom of Member-0.Member-0 is fixed tothe ground from its bottom face. Material of members 0, 1 and 2 is steel (AISI 1020). Also, member 3 is composed of steel (AISI1020) and aluminum (alloy1060). The inertialproperties of the members are given in Table 3.2.

Table 3.2 Inertial properties

	Member 1	Member 2	Member 3
l.o[a]	$[0,0,0]^T$	$[156,462,-55.04]^T$	$[153.01,912.56,106.79]^T$
c.o.g[b]	$[33.16,217.02,-6.96]^T$	$[13.44,124.94,67.14]^T$	$[21.35,88.01,-56.02]^T$
Mass (kg)	88.3	20.367	50.516
I_G[c]	$\begin{bmatrix} 761.4 & 1170 & -164 \\ 1170 & 3653 & -105 \\ -164 & -105 & 646 \end{bmatrix}$	$\begin{bmatrix} 9054 & 662 & 217 \\ 662 & 173 & 1823 \\ 217 & 1823 & 8187 \end{bmatrix}$	$\begin{bmatrix} 685 & 110 & 0.4 \\ 110 & 511 & -98 \\ 0.4 & -98 & 633 \end{bmatrix}$

a: Position of local origin with respect to global coordinates

b: Position of center of gravity w.r.t. local coordinates at l.o.

c : Moment of inertia with respect to local coordinates at c.o.g. (103 kg-mm2)

The total weightof the robot without the payload is 225 kg. The payload is 5 kg.The locations of the revolute joints (motor1, motor2, andmotor3) are given in table 3.3.

Table 3.3 Locations of revolute joints (w.r.t global coordinates.)

Revolute joints	Coordinates
Motor 1	$[0,118.90,0]^T$
Motor 2	$[151.21,462.65,-70.2]^T$
Motor 3	$[151.27,912.68,-52.7]^T$

3.4 Finite element model of the robot

The robot is modeled with some assumptions because of insufficient information about the parts of the robot in ABAQUS CAE. The assumptions are listed below.

- Approximate rotational spring elements have been introduced to define joints flexibility
- Inner parts of the members of the robot such as wires and gears have been ignored

The whole model consists of linear tetrahedral elements of type C3D4 in ABAQUS. The element size is selected between 8.5 mm to 16 mm according to the geometry of members. The translational displacements (and/or rotational angles) of bottom of Member-0faces (or ground-fixed node) are assumed to be equal to zero.The model has 104938 elements and 32074nodes. The meshed model of the robot is illustrated in figure 3.5.

The linear perturbation frequency analysis is performed to find natural frequencies. Mode shapes can be observed by animation. Let fmin be the lowest natural frequency. Also, the static analysis is performed. The gravity is in –y direction. The displacements and vonMises stresses are found. Let umax be the maximum displacement and Smax be the maximum vonMises stress. The robot is more rigid for higher values of fmin, and for lower values of umax and Smax. Experimental modal analysis is done on the robot to verify that the finite element model operates according to real robot, which is explained in chapter 5.

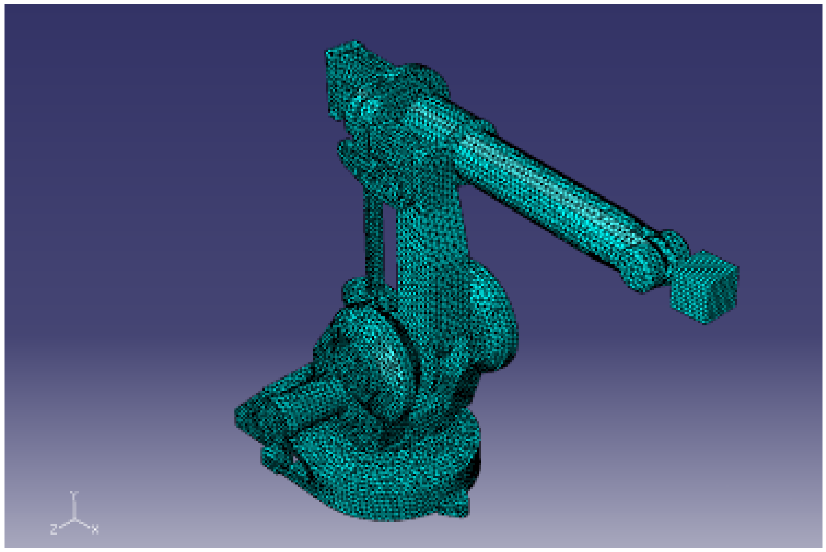


Figure 3.5 F.E model of the robot

Next part discusses the procedures used in ABAQUS to analyze the finite element model of the robot and simulation results. Two different points in the robot model is selected as reference point. The von mises stress and logarithmic strain component for different paths in case 2 are evaluated during the movement.

3.5 Dynamic analysis theory in ABAQUS

Contrary to the static analysis a dynamic analysis has load and responses that vary with time and the duration of loads are small. When the inertia effects in a system are important a dynamic analysis must be performed,ABAQUS has several methods for solving such problems. When nonlinear dynamic response is studied direct integration must be used. ABAQUS can use both implicit direct integration and explicit direct integration. The equations of equilibrium governing the nonlinear dynamic response of a system of finite elements are:

$$M\ddot{U}_t + C\dot{U}_t + KU_t = R$$

$$C\dot{U}_t + KU_t = I$$

Where: M, C, K are the mass, damping and elemental stiffness matrices.R is the external load vector. I is the internal forces and is the sum of CU+ KU

For solving the system off differential equation described in this equation, direct integration is used. That means that prior to the numerical integration; no transformation of the equations onto a different form is carried out. The application of this method is based on two ideas:

• Trying to satisfy equation 2 only at discrete time intervals "Δt", instead of any time "t"
• Assume the variation of the displacements, velocities and accelerations within each time interval "Δt". Obviously, the choice criteria on these assumptions determine the accuracy, stability and cost of the solution procedure.

3.5.1 Explicit method

In this method, known also as central difference method, the kinematic conditions at one increment are used to calculate the kinematic conditions at the next one. At the beginning of each increment the program solves the dynamic equilibrium with respect to accelerations according to equation 4.

$M\ddot{U} = R - I \quad (4)$

Since the acceleration of any node is determined by its mass and the net force acting on it the nodal calculations is very inexpensive. The accelerations are integrated through time using the central difference rule, which calculates the change in velocity from the middle of the previous increment to determine the velocities at the middle of the current increment. After that the velocities are integrated through time and added to the displacement at the beginning of the increment to determine the displacements at the end of the increment. Thus satisfying dynamic equilibrium at the beginning of the increment again provides the accelerations for the new loop.

The explicit method requires small time increment to produce accurate results depending on that the accelerations in the central difference formula are assumed to be nearly constant. Simulations generally take on the order of 10000 to 1000000 increments, but the computational cost per increment is relatively small as explained previously.

3.5.2 Implicit method

The general direct-integration method provided in Abaqus/Standard, called the Hilber-Hughes-Taylor operator is an extension of the trapezoidal rule. In this method the integration operator matrix must be inverted, and a set of simultaneous nonlinear dynamic equilibrium equations must be solved at each time increment. This solution is done iteratively using Newton's method. This nonlinear equation solving process is expensive, and if the equations are very nonlinear, it may be difficult to obtain a solution. However, nonlinearities are usually accounted more simply in dynamic situations than in static situations, because the inertia terms provide mathematical stability to the system. Thus, the method is successful in all but the most extreme cases.

3.6Moving end point tostart position for a path

The position of the robot is defined by the position of the end point shown in figure3.6.The global coordinates are x, y and z and x_b, y_b and z_b are the local coordinates of the end point attached to the robot.

Let the position vector of the end-point with respect to the global coordinates be $\mathbf{q_e}= [x_e, y_e, z_e]^T$. $\mathbf{q_{ea}}=[872.74,1031.70,0]^T$ is for the assembly position figure 3.6. The motor rotations are θ_1, θ_2, and θ_3 for the axes 1, 2, and 3 respectively. The motor rotation vector is given as $\mathbf{q_m}=[\theta_1,\theta_2,\theta_3]^T$. $\mathbf{q_m}=[0,0,0]^T$ for the assembly position. The displacements are given in mm, and the rotations are given in degrees, unless otherwise stated. Initially, all the motion objects of the joints are set to “Free” mode in CosmosMotion. The motion objects of motor1, motor2 and motor3 have the RotateZ components only and these components are set to “Displacement” mode.

The values of θ1, θ2, and θ3 are assigned to the corresponding RotateZ components with GUI or with the motion program developed in VisualBasic using API of CosmosMotion. The end-point moves to the last position after running the simulation. The values of x_e, y_e, and z_e can be read for the last position.

Figure 3.6 Position of the end point in rest position

3.7Inverse kinematics

3.7.1 Theory of inverse kinematics

Kinematics studies the motion of bodies without consideration of the forces or moments that cause the motion. Robot kinematics refers the analytical study of the motion of a robot manipulator. Formulating the suitable kinematics models for a robot mechanism is very crucial for analyzing the behavior of industrial manipulators.

The inverse kinematic equations give the joint angles that will place the end effector in the goal frame. The inverse kinematic equations are a set of many equations which must be solved simultaneously. Solution of these equations is complicated by the existence of multiple solutions and by the possibility of singularities. The inverse kinematics problem of the serial manipulators has been studied for many decades. It is needed in the control of manipulators. Solving the inverse kinematics is computationally expansive and generally takes a very long time in the real time control of manipulators. Tasks to be performed by a manipulator are in the Cartesian space, whereas motors work in joint space.

Cartesian space includes orientation matrix and position vector. However, joint space is represented by joint angles. The conversion of the position and orientation of a manipulator end-effector from Cartesian space to joint space is called as inverse kinematics problem.There are two solutions approaches namely, geometric and algebraic used for deriving the inverse kinematics solution, analytically:

- Geometric solution approach
- Algebraic solution approach

3.7.2 BIDAM inverse kinematics

The incremental displacement vector of the end-point isdefined as $\mathbf{q_{ei}}= [x_i, y_i, z_i]^T$. x_i, y_i, and z_i are the displacements ofthe end-point from its previous position in the x, y, and zdirections, respectively. The incremental motor rotation vectoris defined as $\mathbf{q_{mi}}=[\theta_{1i},\theta_{2i},\theta_{3i}]^T$. θ_{1i}, θ_{2i}, and θ_{3i} are the incrementalmotor rotations for the axes 1, 2, and 3 respectively. In theinverse kinematic analysis, the components of $\mathbf{q_{ei}}$are given andthe components of $\mathbf{q_{mi}}$are found.The starting position of the end-point is defined as$\mathbf{q_{es}}=\mathbf{q_{ea}}+\mathbf{q_{is}}$, where the first term is the assembly position vector.The second term is the incremental displacement vector for thestarting position, which is defined as $\mathbf{q_{is}}=[x_{is},y_{is},z_{is}]^T$. The timeinterval for an incremental displacement is defined as $\mathbf{t_i}=[t_1,t_2]$,where t_1 is the starting time and t_2 is the ending time for theincremental motion.The RotateZ components of motor1, motor2, and motor3 areset to “Free” for the inverse kinematic analysis.A motion object is assigned to Member-3. The location ofthe object is at the end-point figure 3.6. The variable name “motione”is assigned to the motion object. motione has 6 degrees offreedom titled as TranslateX, TranslateY, TranslateZ, RotateX,RotateY, and RotateZ. The TranslateX, TranslateY, andTranslateZ components are set to the “Displacement” mode.The RotateX, RotateY, and RotateZ components are set to the“Free” mode. Successive incremental displacement values can be assignedto the TranslateX, TranslateY, and TranslateZ components. TheVisualBasic command for the assignment to the TranslateYcomponent is given below.

“Call motione.Motions.TranslateY.Function.SetExpression (expy)”

An example of the string variable expy, is given as:
expy = “-595+STEP(TIME,0,0,2,600)+STEP(TIME,3,0,6,-600)”

Here is y=-595 mm, y_i=600 mm, and t_i= [0, 2] for the first step. y_i=-600 mm, and t_i=[3,6] for the second step. So, y_e=-595+600mm at t=t_1, and y_e=-595+600-600 mm at t=t2. There is nomotion from 2 s to 3 s.

The time histories of the displacement, velocity, and acceleration for a step in CosmosMotion are shown in figure 3.7. Knowing d_m, t_1, and t_2 for a step, a_0 can be calculated by takingd (t_2) = dm. So, d_m=600 mm and a_0=900 mm/s^{2}for the first step above.

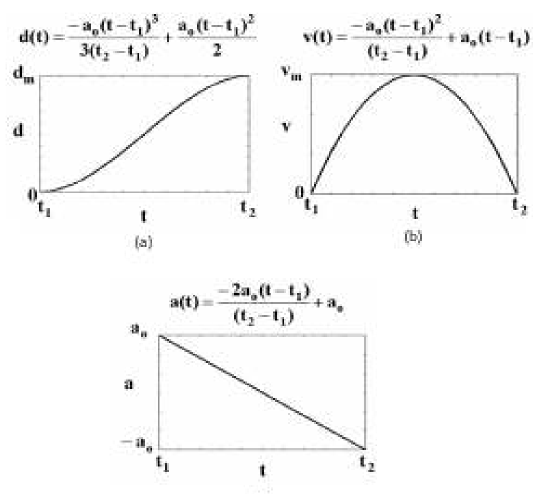


Figure 3.7Time histories of a) displacement, b) velocity and c) acceleration for step function in CosmosMotion

By clicking on the inverse kinematics button, the motor angles times history and angular velocity time history results will be transferred to VisualBasic And will evaluated if the angles of the motors, exceed angle limits.Red color next to reachability option in results form indicates that the point is not reachable and green shows that the point is in workspace of the robot figure 3.8.

If the any motor exceeds the maximum angular velocity limit the cycle next to that motor turns to red in results form. Therefore the simulation time must be increased and tested again. In most of robot softwares which is used in industry this option is not available.

Figure 3.8 Work space evaluations

3.8Forward kinetics

3.8.1 Theory of Forward kinematics

The forward kinematics problem is to determine the position and orientation of the end-effector, given the values for the joint variables of the robot. The joint variables are the angles between the links in the case of revolute or rotational joints, and the link extension in the case of prismatic or sliding joints.

In order to have forward kinematics for a robot mechanism in a systematic manner, one should use a suitable kinematics model. Denavit-Hartenberg method that uses four parameters is the most common method for describing the robot kinematics.

3.8.2 BIDAM forward kinetics

All the components of the end-point motion (motione) are setto “Free” and the RotateZ components of motor1, motor2, andmotor3 are set to “Displacement” in the forward kineticanalysis. The values of θ_{1i}, θ_{2i}, and θ_{3i} found in the inversekinematic analysis are assigned to the corresponding RotateZcomponents. Plot objects are defined to observe the kineticoutputs such as motor torques and reaction forces.

By clicking on forward kinetics figure 3.9,the values of motor angles are assigned to motors and by clicking on the results, the results for maximum reaction forces in bearings, maximum motor torques, are divided to their reference limit values and multiplied by 100. The results will be shown in the results form with lines. Each line shows the parametersvalue as a percentage of its reference limit. If the value is under 50% the line is shown in green color. If it exceeds the maximum limit then the linescolor turns to red. For the values between 50% and 100% brown color is selected.

Figure 3.9 Forward kinetics

CHAPTER FOUR
SIMULATION RESULTS

4.1 Casestudies

In all of the cases it was supposed that the robot will weld a 1 meter long line on a work piece in 4 seconds.Depending on the position of the work piece the robot will pass different trajectories to weld the line on work piece. We can position the work piece so that the line would be vertical, horizontal or diagonal and also in each condition we can choose the distance of the work piece from the robot. In this study three different distances considered for each three cases (vertical, horizontal and diagonal) so we will have 9 different paths. Strength of the robot in each path is evaluated and acceptable paths are selected for the task.

Case 1(horizontal paths), in this case 3 paths parallel to -z axis are selected as shown in figure 4.1. Coordinates of the start and end points of the paths are given in the table 4.1.

Case 2 (vertical paths), in this case 3 paths parallel to -y axis are selected as shown in figure 4.12. Coordinates of the Start and end points of the paths are given in the table 4.5.

Case 3(diagonal paths), in this case 3 paths parallel to ZY plane are selected as shown in figure 4.23. Coordinates of the start and end points of the paths are given in the table4.9.

4.2 Case 1 (Horizontal paths)

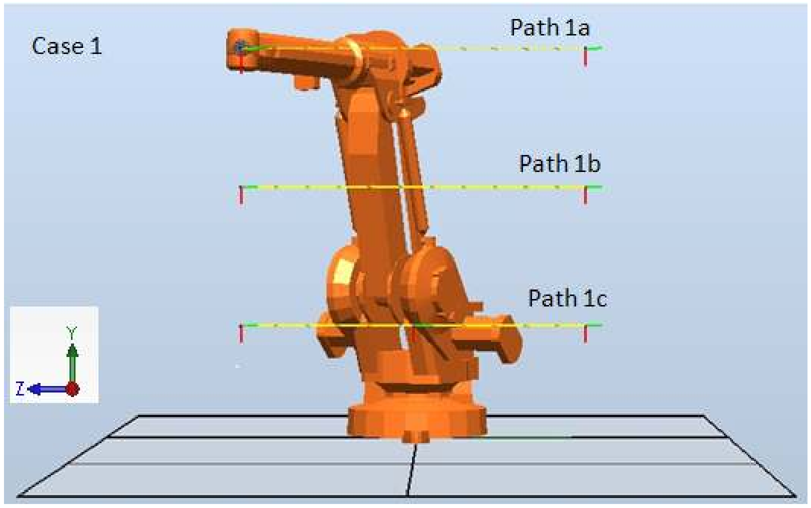


Figure 4.1Case 1 (paths 1a, 1b and 1c)

Table 4.1 Coordinates of start and end points of the paths

	Path 1a		Path 1b		Path 1c	
	S.P	E.P	S.P	E.P	S.P	E.P
X	955	955	955	955	955	955
Y	1195	1195	795	795	395	395
Z	-500	500	-500	500	-500	500

S.P: start point of the path

E.P: end point of the path

```
Path 1a
nsim,100
0,0,0
inverse
0,0,0
0,2,0,163.3,-500
3,7,0,0,1000
-1
```

```
Path 1b
nsim,100
0,0,0
inverse
0,0,0
0,2,0,-236.7,-500
3,7,0,0,1000
-1
```

```
Path 1c
nsim,100
0,0,0
inverse
0,0,0
0,2,0,-636.7,-500
3,7,0,0,1000
-1
```

Figure 4.2 Databases for the paths in case 1

4.2.1Simulation results for paths 1a, 1band 1c

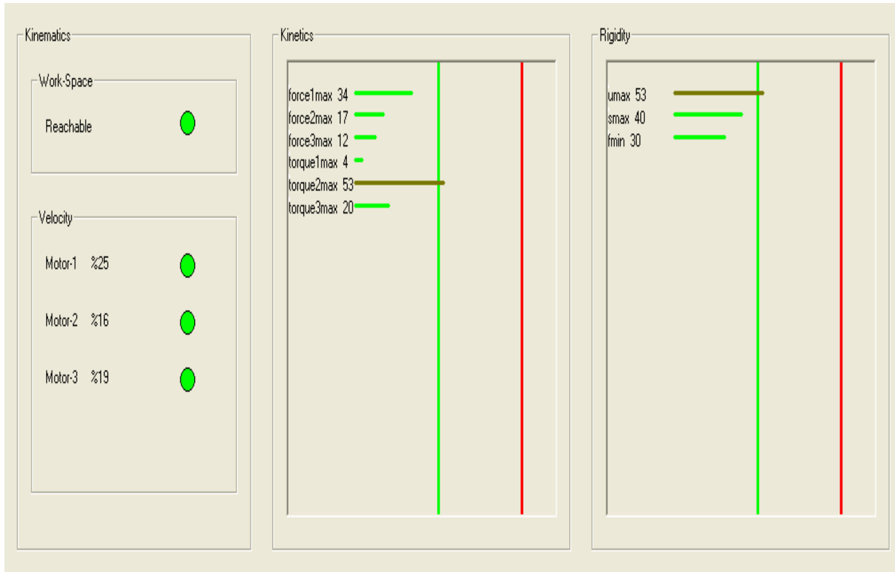


Figure 4.3 BIDAM results for path 1a

The reaction forces in bearings are mostly affected by the payload and sudden movements of the robot. The torque developed by motors varies with the speed of the motor when it accelerates from full stop or zero speed, to maximum operating speed. The Locked Rotor Torque or Starting Torque is the torque the electrical motor develop when its starts at rest or zero speed.

According to results for path 1a figure 4.3 the path is in robots workspace and the motorsdon't exceed velocity limits. The reaction forces are under limit of 50%. The second motor's torque line is brown which indicates that it has exceeded the 50% of its maximum value. The displacement line has exceeded the green area. This path is not recommended for welding tasks.

F.E results for path 1a

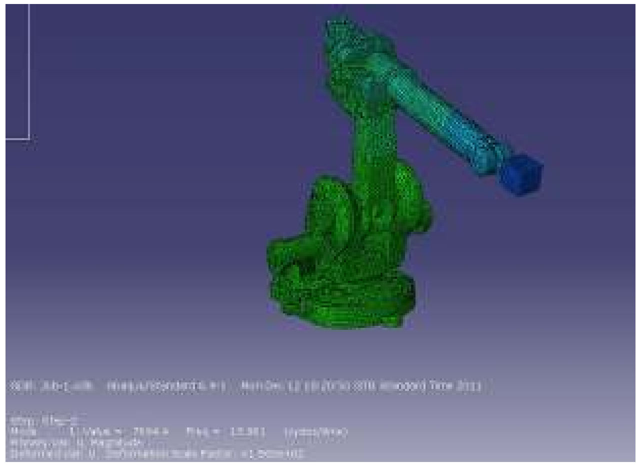


Figure 4.4 First mode of natural frequency 13.961 Hz

Figure 4.5 Second mode of natural frequency 16.998 Hz

Table 4.2 Static and modal analysis results in path 1a

S_{max} (maximum vonMises stress)	12 Mpa
U_{max} (the maximum displacement)	132.0 μ
f_{min} (the lowest natural frequency)	13.961 Hz

Figure 4.6 BIDAM results for path 1b

Results of evaluation for path 1b are shown in figure 4.6. All of the parameters are in green area. The line is in work space of the robot. This line would be the best path between horizontal paths in view of strength of the robot. In this path the payload can be increased or even the simulation time can be decreased if it's needed. The results for the F.E analysis are given in the next page.

F.E results for path 1b

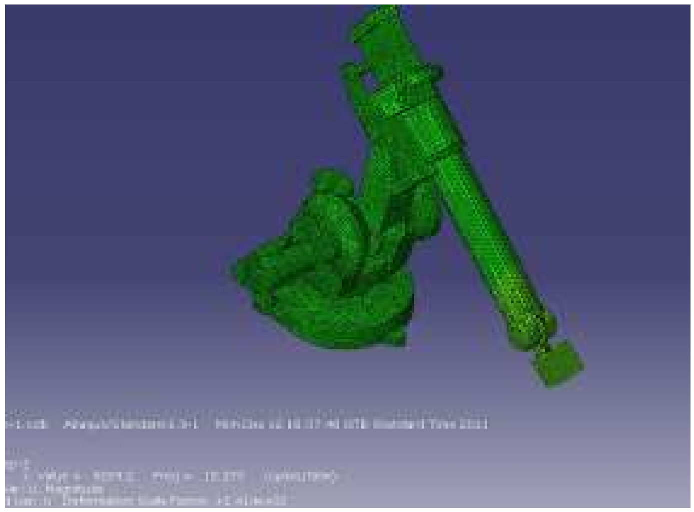


Figure 4.7 First mode of natural frequency 15.273 Hz in path 1b

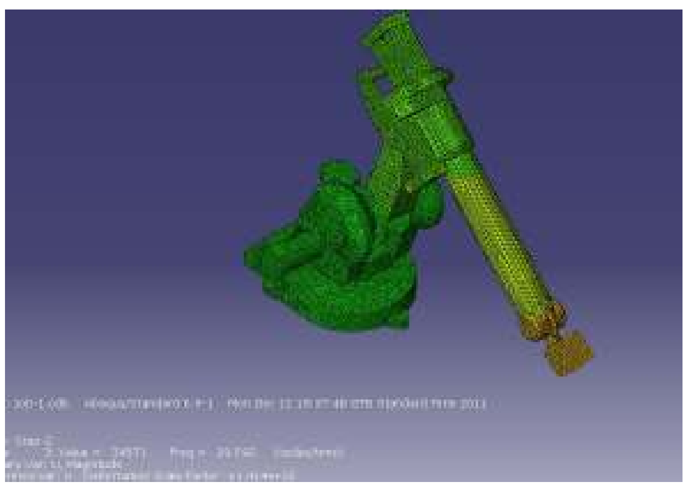


Figure 4.8 Third mode of natural frequency 29.592 Hz in path 1b

Table 4.3 Static and modal analysis results in point 1b

S_{max} (maximum von Mises stress)	11.24 Mpa
U_{max} (the maximum displacement)	126.2 μ
f_{min} (the lowest natural frequency)	15.273 Hz

Figure 4.9 Simulation results for path 1c

In this path motor 2 exceeds the green area but maximum displacement is the smallest compared to previous paths. If the application is welding, the robot would have the lowest displacement in this path. Second motors maximum torque can be decreased by increasing the simulation time. Depending on the application of the robot in industry one of these paths can be selected.

Path 1c F.E results

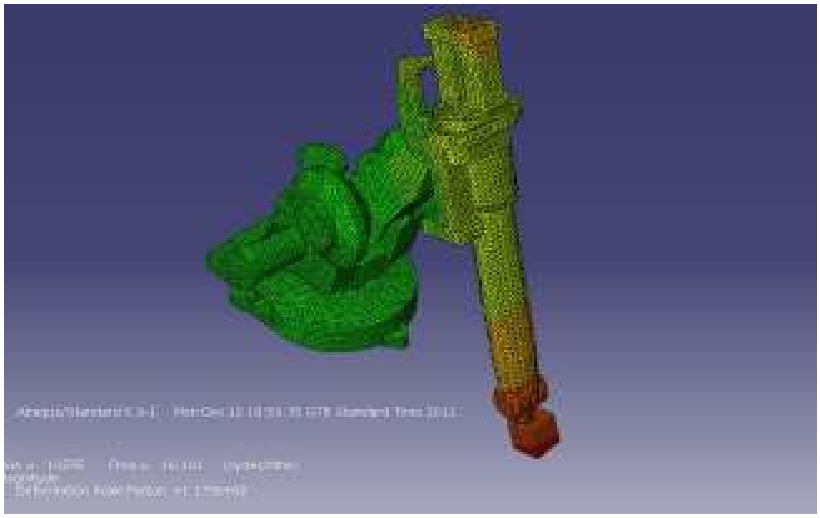


Figure 4.10 First mode of the natural frequency 16.101 Hz for path 1c

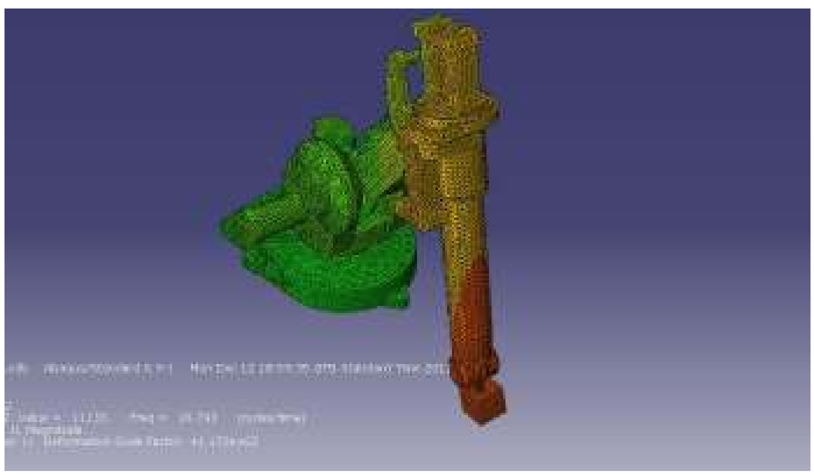


Figure 4.11 Second mode of the natural frequency 16.793 Hz for path 1c

Table 4.4 Static and modal analysis results in point 1c

S_{max} (maximum vonMises stress)	9.92 Mpa
U_{max} (the maximum displacement)	106.5 μ
f_{min} (the lowest natural frequency)	16.101 Hz

4.3 Case 2 (vertical paths)

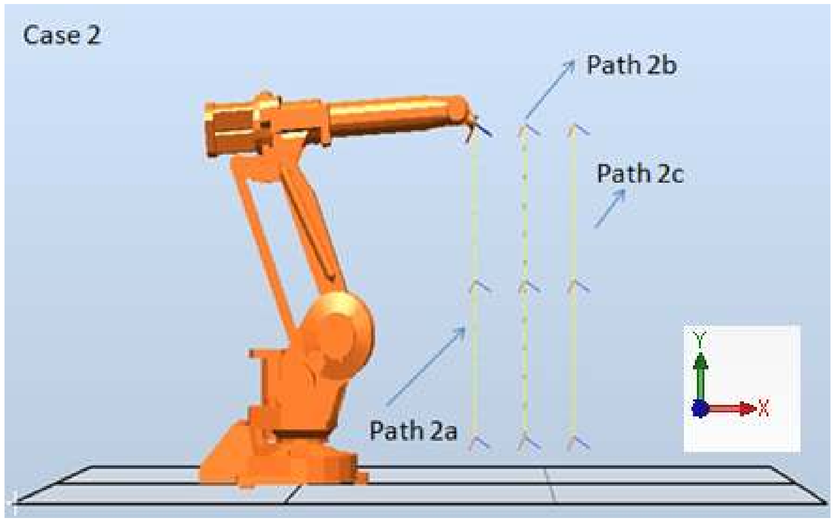


Figure 4.12 Case2 (paths 1a,1b and 1c)

Table 4.5 Coordinates of start and end points of the paths

	Path 2a		Path 2b		Path 2c	
	S.P	E.P	S.P	E.P	S.P	E.P
X	700	700	900	900	1100	1100
Y	150	1150	150	1150	150	1150
Z	0	0	0	0	0	0

S.P: start point of the path

E.P: end point of the path

```
Path 2c
nsim,100
0,0,0

inverse
0,0,0
0,2,227.66,-881.7,0
3,7,0,1000,0
-1
```

```
Path 2a
nsim,100
0,0,0

inverse
0,0,0
0,2,-172.74,-881.7,0
3,7,0,1000,0
-1
```

```
Path 2b
nsim,100
0,0,0

inverse
0,0,0
0,2,27.26,-881.70,0
3,7,0, 1000,0
-1
```

Figure 4.13 Databases for the paths in case 2

4.3.1 Simulation results for paths 2a, 2band 2c

Figure 4.14 BIDAM results for path 1a

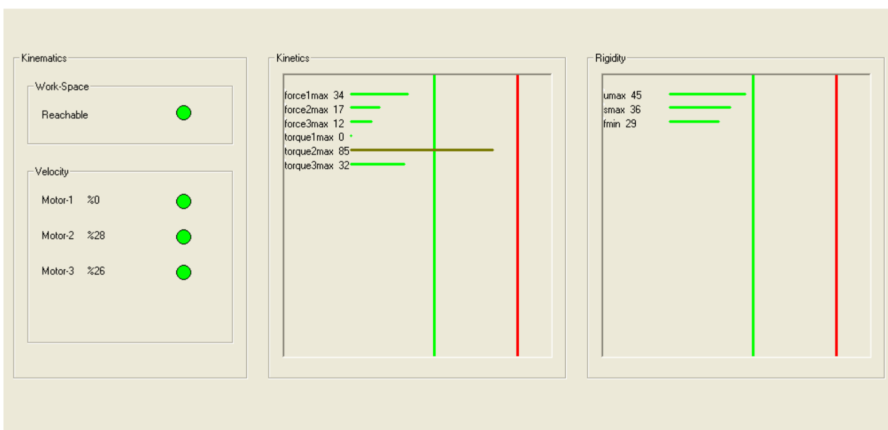


Figure 4.15 BIDAM results for path 2b

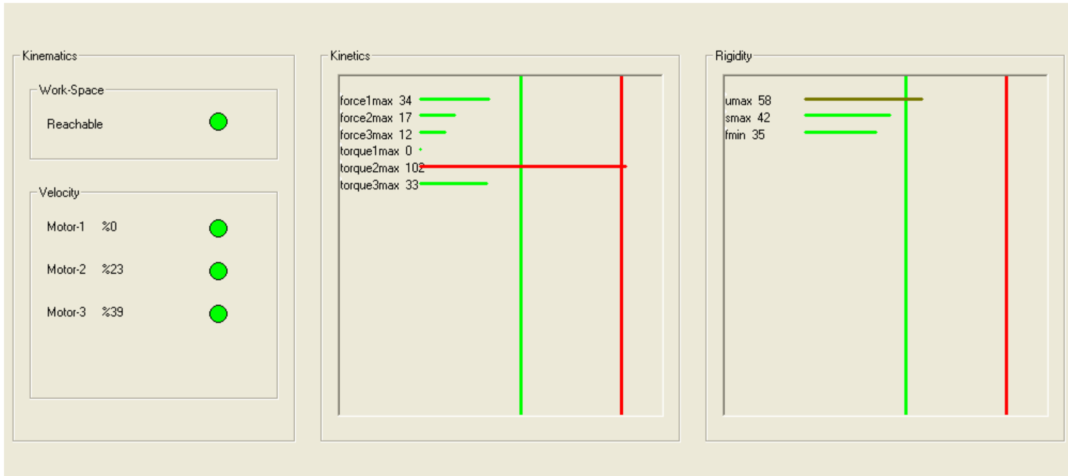


Figure 4.16 BIDAM results for the path 2c

Although the rigidity of the robot in green area the second motors torque is exceeded the green area and the motor 1 is not working. In mechanical systems in view of life span it is more wanted that the motors generate equal torques. Therefore the vertical path can be changed to other paths. If there no chance of changing the path the time of simulation can be reduced.

F.E results for path 2a

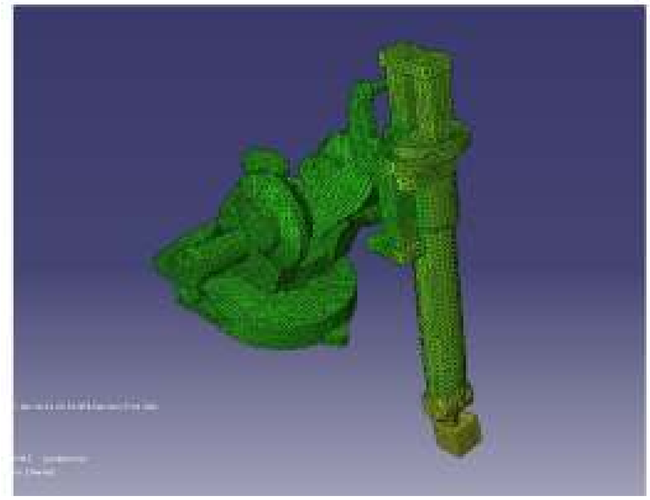


Figure 4.17 first mode of the natural frequency 15.923 Hz for path 2a

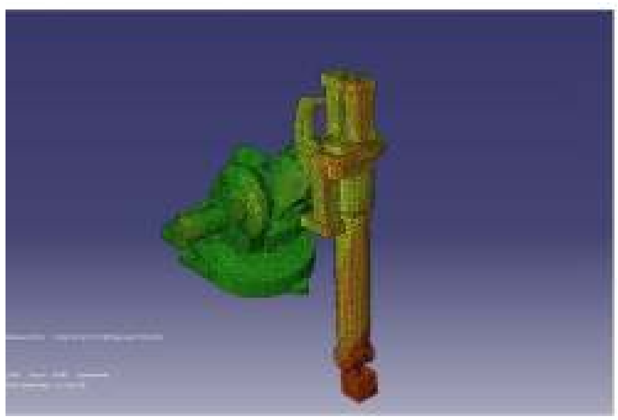


Figure 4.18Second mode of the natural frequency 16.587 Hz path 2a

Table 4.6 Static and modal analysis results in point 2a

S_{max} (maximum von Mises stress)	8.85 Mpa
U_{max} (the maximum displacement)	94.4 μ
f_{min} (the lowest natural frequency)	15.923 Hz

F.E results for the path 2b

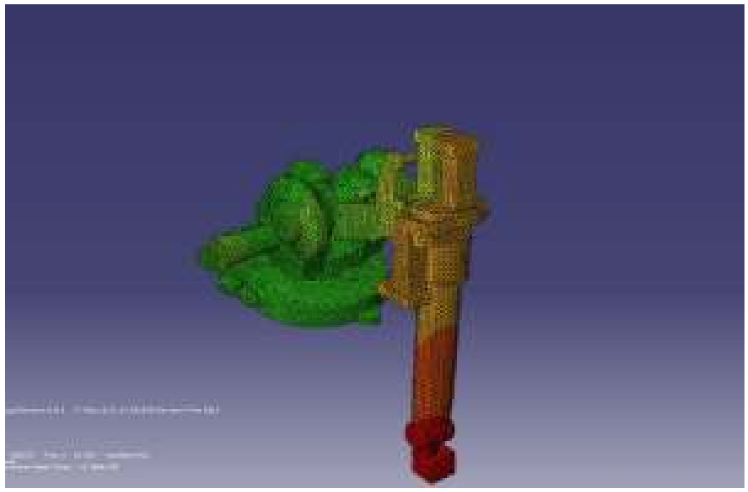


Figure 4.19 First mode of the natural frequency 14.44Hz for path 2b

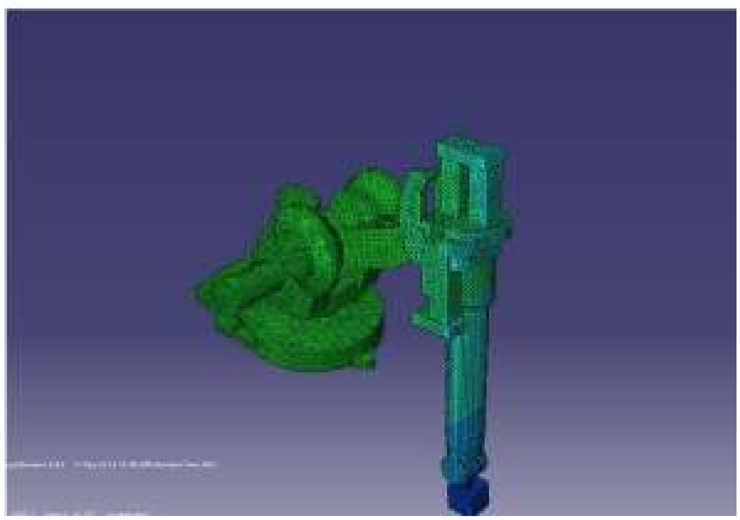


Figure 4.20Second mode of the natural frequency 15.252Hz for path 2b

Table 4.7 Static and modal analysis results in point 2b

S_{max} (maximum von Mises stress)	10.71 Mpa
U_{max} (the maximum displacement)	112 μ
f_{min} (the lowest natural frequency)	14.244 Hz

F.E element results for the path 2c

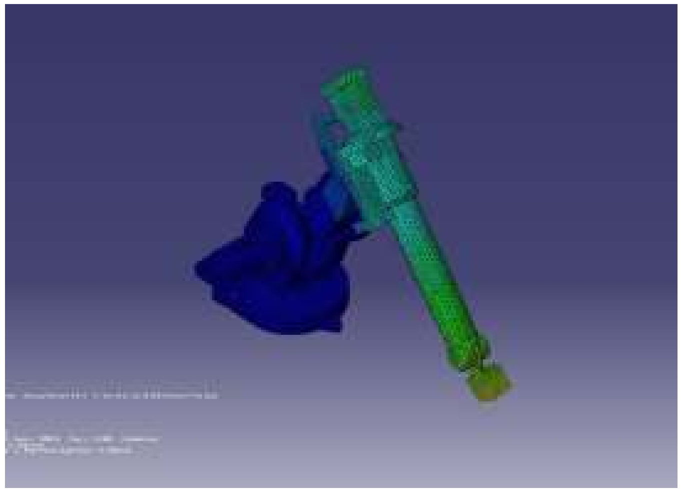


Figure 4.21 First mode of the natural frequency 13.051Hz for path 2c

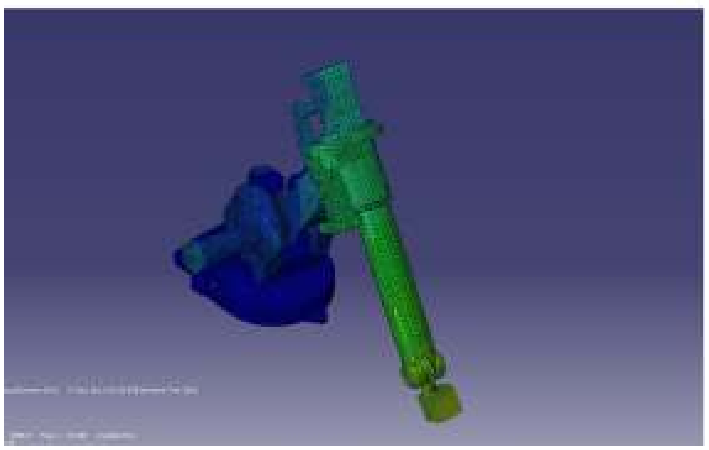


Figure 4.22 Second mode of the natural frequency 14.132Hz for path 2c

Table 4.8 Static and modal analysis results in point 2b

S_{max} (maximum vonMises stress)	12.61 Mpa
U_{max} (the maximum displacement)	144 μ
f_{min} (the lowest natural frequency)	13.051 Hz

4.4 Case 3 (diagonal paths)

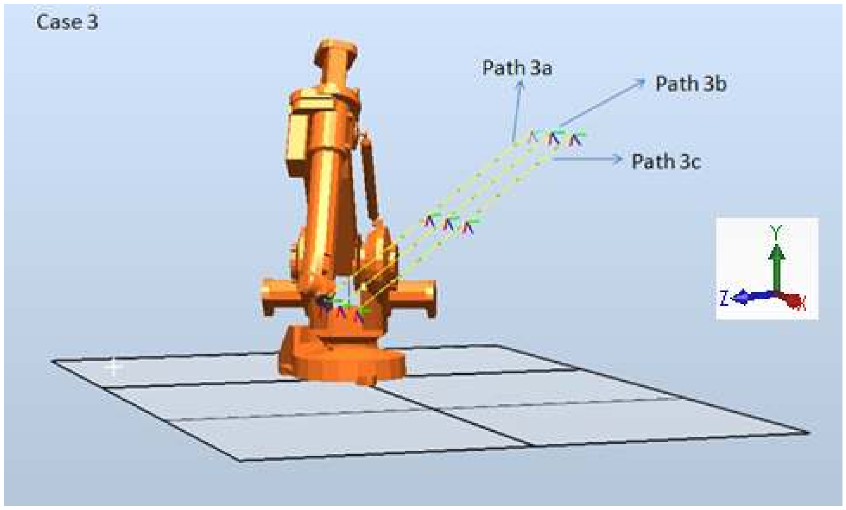


Figure 4.23 Case 3 (paths 3a, 3b and 3c)

Table 4.9 Coordinates of end and start points of the paths

	Path 3a		Path 3b		Path 3c	
	S.P	E.P	S.P	E.P	S.P	E.P
X	543.61	543.61	743.61	743.61	943.61	943.61
Y	1000	400	1000	400	1000	400
Z	400	-400	400	-400	400	-400

S.P: start point of the path

E.P: end point of the path

```
Path 3a
nsim,100
0,0,0
inverse
0,0,0
0,2,-329.6,-31.76,400
3,7,0,-600,-800
-1
```

```
Path 3b
nsim,100
0,0,0
inverse
0,0,0
0,2,-129.69,-31.76,400
3,7,0,-600,-800
-1
```

```
Path 3c
nsim,100
0,0,0
inverse
0,0,0
0,2,70.78,-31.76,400
3,7,0,-600,-800
-1
```

Figure 4.24Databases for the paths in case 3

4.4.1 Simulation results for paths 3a, 3b and 3c

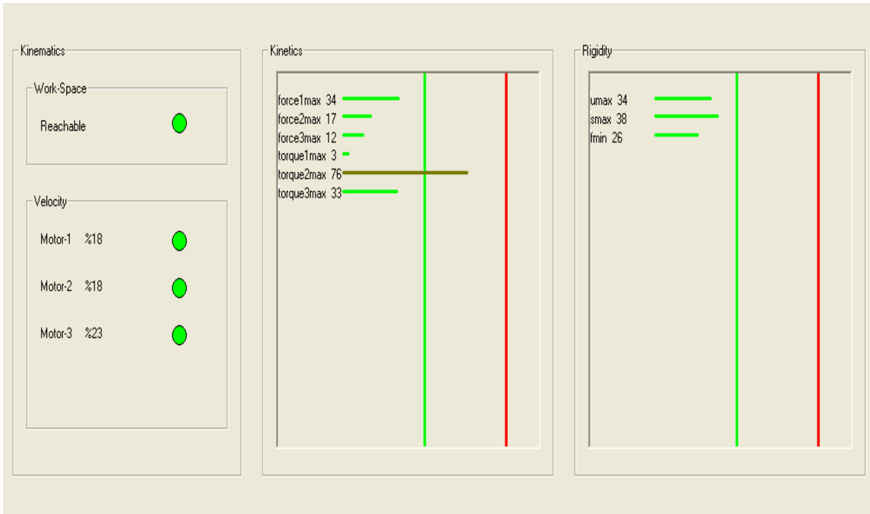


Figure 4.25 BIDAM results for path 3a

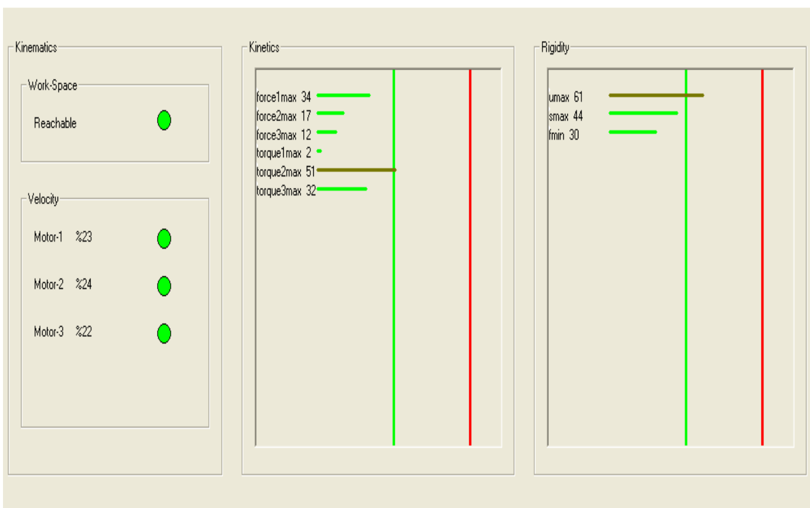


Figure 4.26 BIDAM results for path 3b

Figure 4.27 BIDAM results for path 3c

The torque value for the motor 2 in path 3a and 3b are high. In path 3c motor torques are in green area. The maximum displacement due to flexibility of the robot in path 3c is high and this path is not recommended for welding applications. The velocity limits are in green area and the application can be decreased and evaluate the results again.

F.E results for path 3a

Figure 4.28First mode of the natural frequency 14.327Hz for path 3a

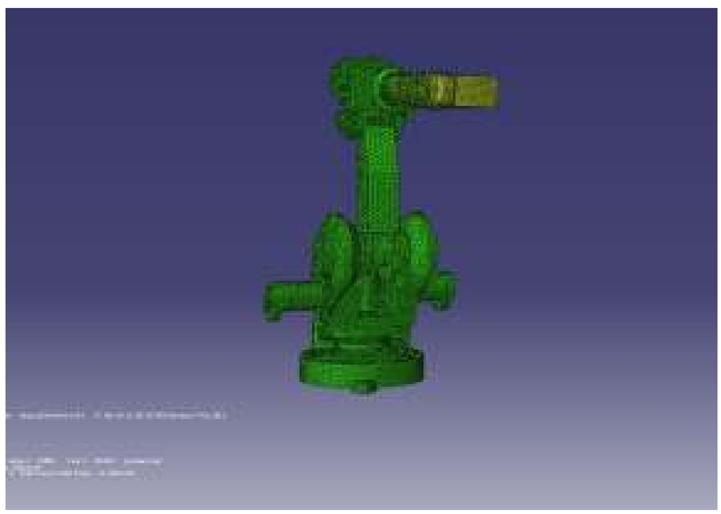


Figure 4.29 Second mode of the natural frequency 20.1Hz for path 3a

Table 4.10 Static and modal analysis results in path 3a

S_{max} (maximum von Mises stress)	11.5 Mpa
U_{max} (the maximum displacement)	84 μ
f_{min} (the lowest natural frequency)	14.0377 Hz

F.E results for path 3b

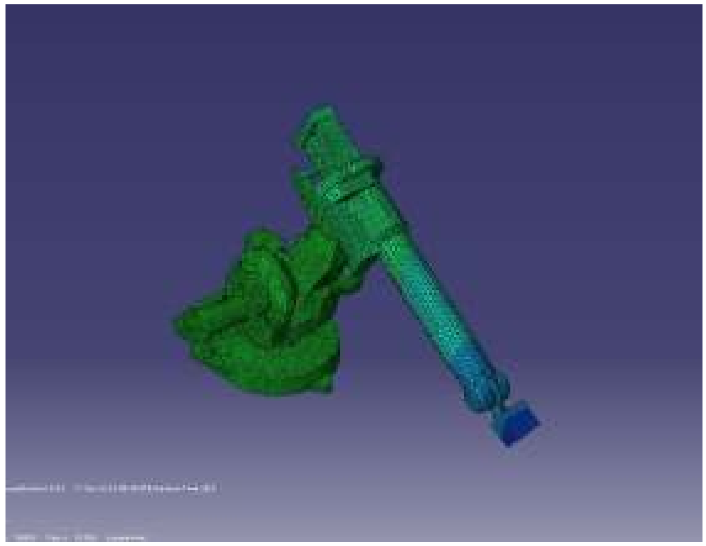


Figure 4.30First mode of the natural frequency 13.919Hz for path 3b

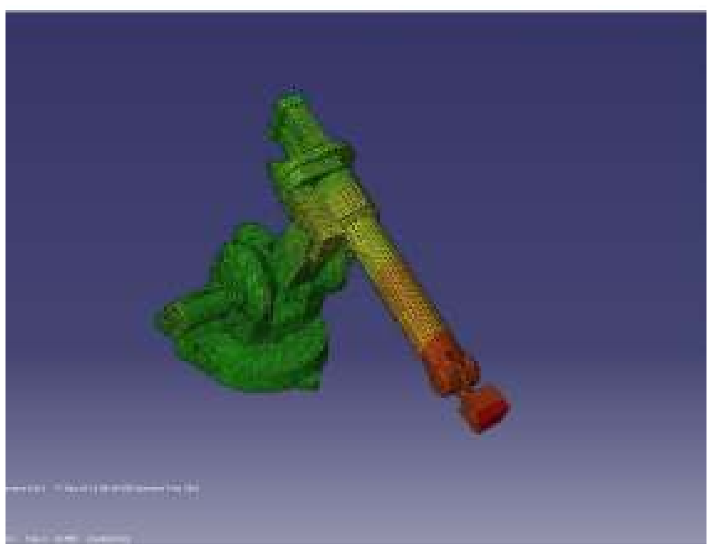


Figure 4.31 Second mode of the natural frequency 14.898Hz for path 3b

Table 4.11 Static and modal analysis results in point 3b

S_{max} (maximum vonMises stress)	13.1 Mpa
U_{max} (the maximum displacement)	152 μ
f_{min} (the lowest natural frequency)	13.919 Hz

F.E results for Path 3c

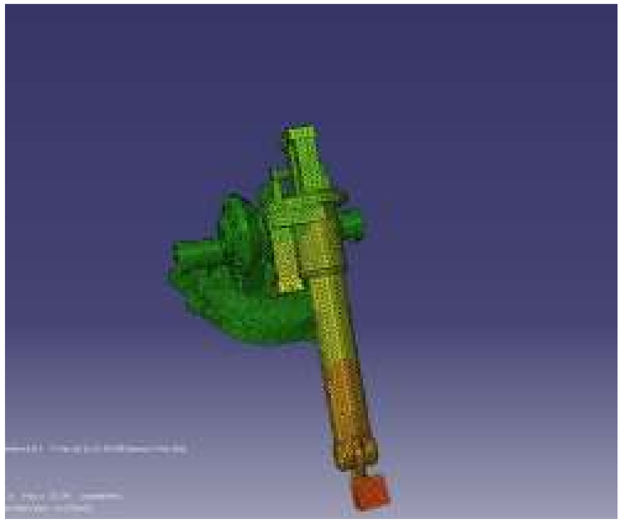


Figure 4.32 First mode of the natural frequency 13.3Hz for path 3c

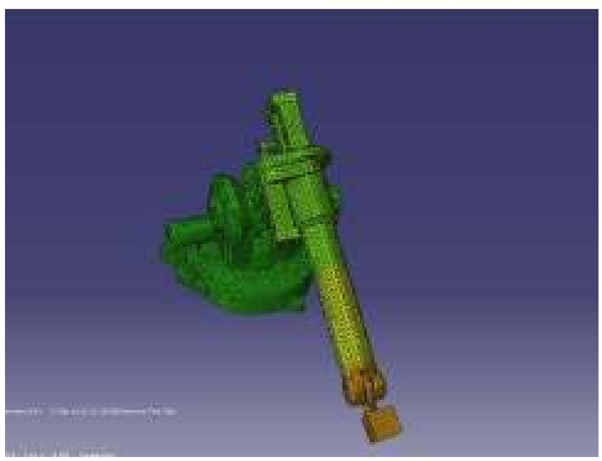


Figure 4.33 Second mode of the natural frequency 14.0Hz for path 3c

Table 4.12 Static and modal analysis results in point 3c

S_{max} (maximum vonMises stress)	13.26 Mpa
U_{max} (the maximum displacement)	171 μ
f_{min} (the lowest natural frequency)	13.374 Hz

4.5Simulation results for dynamic F.E ABAQUS analysis for case 2

Two different points are selected in the robot model figure 6.1. The coordinates of these points are given in table 6.1. The von mises stress in these points during simulations is graphed in ABAQUS. Graphs for case 2 paths are given below. The displacement of the end effector after movement due to residue vibration and von mises stress are considered for evaluation. Each analysis takes about 24 hours with a work station.

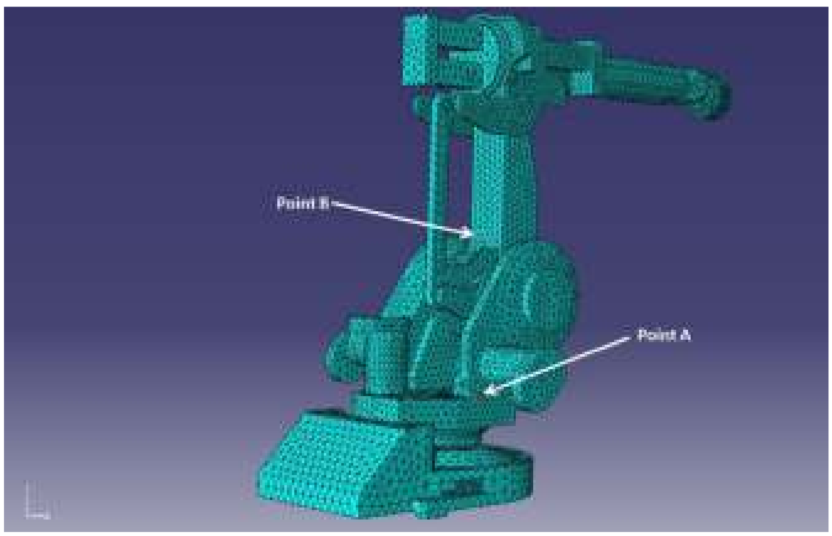


Figure 4.34 Points A and B in robot model

Table 4.13Coordinates of the points in model

Point A	**(-16.488,233.904,144)**
Point B	**(129,594.55,-1)**

Figure 4.35 Von mises stress in point A for path 2a

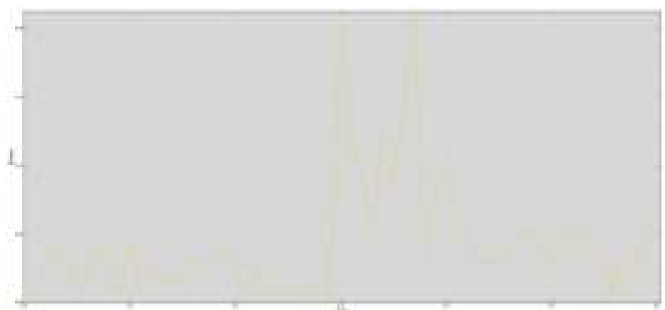


Figure 4.36 Von mises stress in point B for path 2a

Maximum stress during simulation is in support of the robot which is 130 Mpa.Maximum displacement due to residual vibration is 0.3 mm.

Figure 4.37 Von mises stress in point A for path 2b

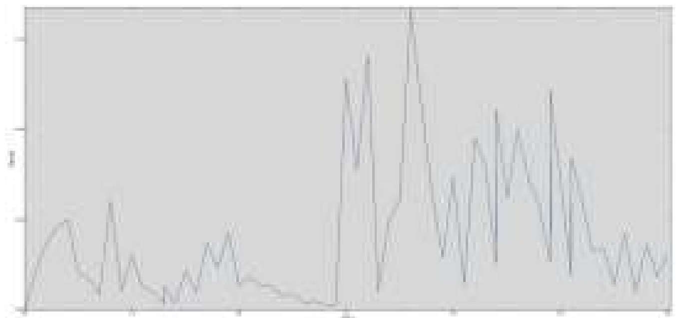


Figure 4.38 Von mises stress in piont B for path 2b

Maximum stress during simulation is in support of the robot which is 88 Mpa.Maximum displacement due to residual vibration is 0.23 mm.

Figure 4.39Von mises stress in point A for path 2c

Figure 4.40 Von mises stress in point B for path 2c

Maximum stress during simulation is in support of the robot which is 167 Mpa.Maximum displacement due to residual vibration is 0.429 mm. Results show that the vonmises stress in path 2c is the highest the application can be improved by choosing 2b path.

CHAPTER FIVE
EXPERIMENTAL RESULTS FOR MODAL ANALYSIS

Experimental modal analysis provides a dynamic characterization of structures under real mechanical conditions (with cabling, supports, etc…). Each structure has its own natural frequencies (where the amplitude of the system's response is much greater than the amplitude of the excitation) and natural modes of vibration. The natural frequencies calculated with this experiment are compared to natural frequencies of the model.

This type of analysis is essential to verify that the finite element model operates according to real robot.

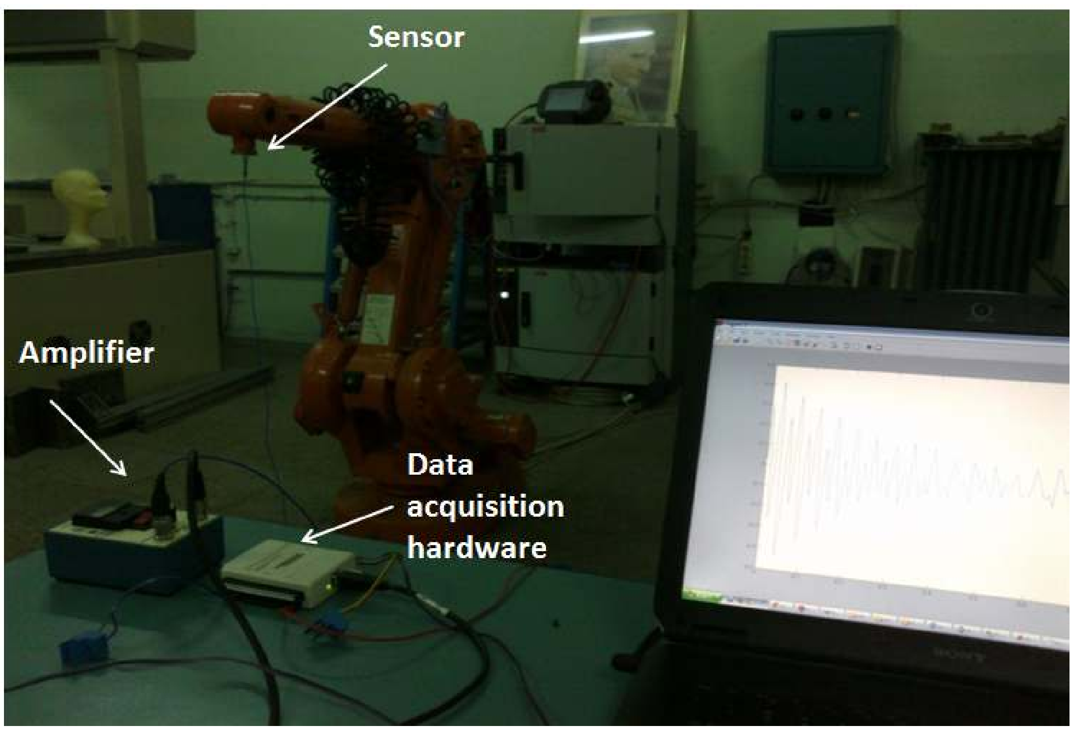


Figure 5.1 Accelerometer connected to end-effector of the robot

In this method response of the robot to excitation force (impacted hammer) is measured in onedirection with motion sensors (accelerometers).

Figure 5.2 PCB accelerometer and impact hammer

An amplifier has been used to amplify the signals. Specializeddata acquisition hardware is needed to properly acquire these vibration signals. National instrument hardware is used to transfer the data to pc. A program which is developed in VisualBasic saves analog signals time history in a text document. The signal can be plotted in MATLAB figure 5.3.

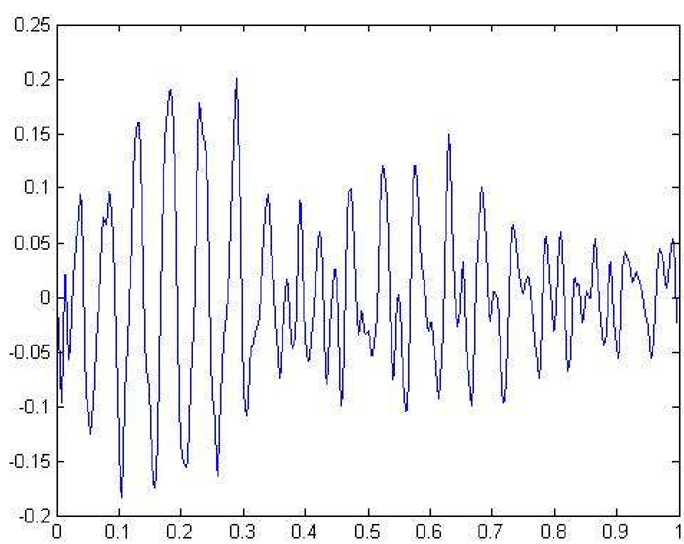


Figure 5.3 Amplitude of acceleration signal versus time

The measured time data is transformed from the time domainto the frequency domain using a Fast Fourier Transform algorithm which is written in MATLAB figure 5.4. Thepeaks in frequency response spectrum show the natural frequencies of the model. The results of the experiment are compared with the results of the simulation table 5.1.

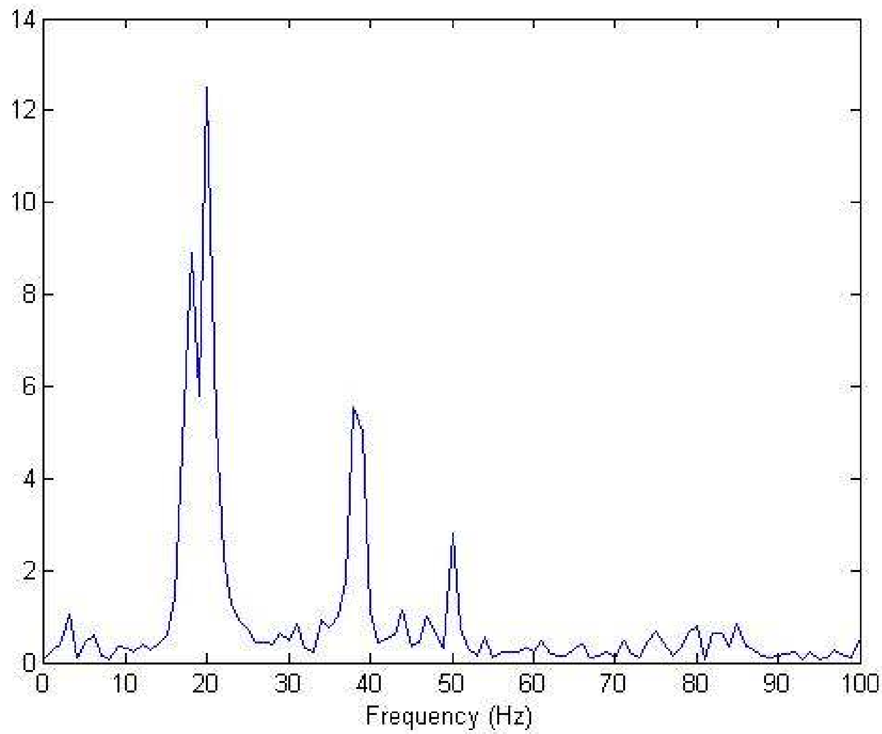


Figure 5.5 Frequency spectrum of signal

Table 5.1 Simulation and experiment results of modal analysis

	Experiment	Simulation
Fisrt natural frequency	18	18.625
Second natural frequency	20	22.526
Third natural frequency	38	45.563

CHAPTER SIX
CONCLUSIONS

Three axis serial robots are used in a wide range of industrial applications. In industry, these robots are applied to various jobs, which consist of different tasks. Evaluation of robot strength in every task is a very complex procedure that plays an important role in optimizing of application of robots. Several simulation programs are available for programming robots. These programs are restricted in only kinematic analysis. The research objective of this thesis is to evaluate the kinetic parameters and rigidity workspace of the robot and to determine the optimum trajectories for given tasks in view of life span of robot and less energy consumption.

In this study strength of a 3 axis serial robot for a given task is evaluated by integrated dynamic analysis.The process uses SolidWorks+ CosmosMotion+Abaqus program packages.A program is developed in VisualBASIC to analyze the strength of the robot for a given path. The programming applicationinterface (API) capabilities of the package are used. The program uses the package for the rigid body dynamic analyses necessary to test kinematic and kinetic limits and F.E analyses for rigidity work space.

The following parameters are considered in the analyses:

1. Robot workspace
2. Motor angular velocity limits
3. Reaction forces in bearings
4. Motor torques
5. Maximum displacement during the path
6. Von mises stress
7. Lowest natural frequency

3 different cases considered for analysis which are Vertical, horizontal and diagonal paths. Each case has three different paths. Every path has different distance from robot. The simulation results show that the motor torques in case 2 are the highest between cases. Robot is more rigid in case 1 and the motor torques are in acceptable ranges. The third path in case 3 has the lowest motor torques (Path 3c). This path can be selected if the application is peak and place but for welding the displacement due to flexibility of robot is important and second path in case 1 (Path 1b) can be considered for this kind application.

The proposed approach uses the strength of the robot as the optimality, which is very important in view of the life span of a robot. The process can be applied to any available path for a given task by user. Theresults for the paths are compared with each other and the desired path is selected by user. The process described in chapter 3 can be used for easy and effective strength evaluating and optimal usage of robots in general. Defining an optimum path automatically for a given task with an integrated program can be considered for a future work.

Arobotic system is needed to achieve certain tasks. Efforts are needed to achieve integrated analysis after defining tasks, although there are many computer programs.The next goal for computer programs may be to achieve all the efforts for the integrated analysis more user-friendly. Engineers should only define tasks and choose the type of themechanical structure. Then, integrated computer programs should offer an optimum path and show simulation results.

REFERENCES

Borrow, J.E. (1988). Optimal robot path planning using the minimum-time criterion.*IEEE Journal of Robotics and Automation*.*Vol 4*. 443-450

Karagülle, H. &Akdağ, M. &Malgaca, L. (2011). An integrated approach forsimulation of mechatronic systems applied to a hexapod robot. *Mathematics andcomputer in simulation.*

Korayem, M.H. &Ghariblu, H.(2003).Maximum allowable load on wheeled mobilemanipulators imposing redundancy constraints. *Robotics and autonomoussystems*. *Vol 44.*151-159

Korayem, M.H. &Shokri, M.(2008).Maximum dynamic load carrying capacity of a6 ups-stewartplatform manipulator.*ScientiaIranica*. *Vol 15*. 131-143

Luh, J.Y., &Gu, Y.L.(1985). Industrial Robots with seven Joints. *IEEEInternationalConference on Robotics and Automation*

Wang, L.T. & Ravani, B.(1988)Dynamic load carrying capacity of mechanicalmanipulators.*Dynamic Systems Measurement and Control.Vol 100*

Witney, D. (1969). Resolved motion rate control of manipulators and humanprostheses.*Man and cybernetics*. *Vol MMS-10*. 47-53

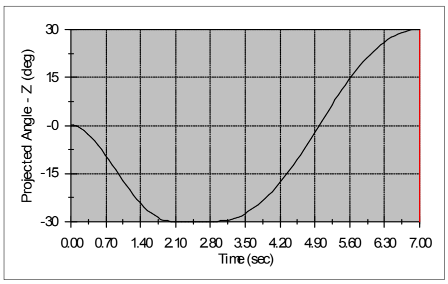
Yun, W. M., & Xi, Y. G.(1991). Optimum motion planning in joint space for robotsusing genetic algorithm.*IEEE on Robotics and Automation Systems.Vol 18*.373-393

Zhaocai,D. (2006). Analysis of dynamic stress and fatigue property of flexible robot.*International Conference on BiomedicalRoboticsandBiomechatronics*.1351-1355

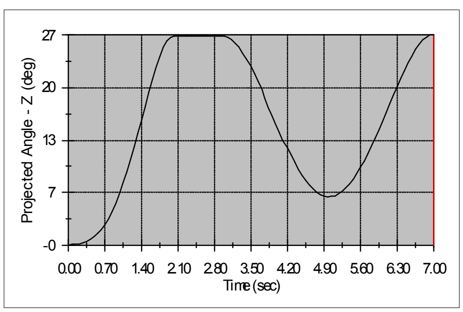
APPENDIX A

KINETIC AND KINEMATIC PARAMETERS GRAPHS FOR PATH 1a

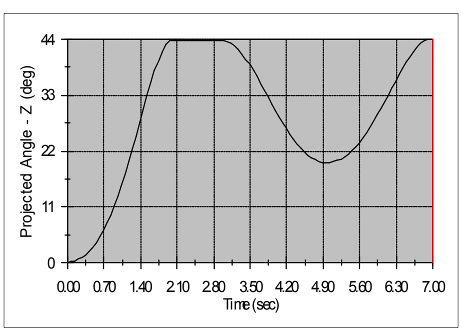
Projected angle-z



Motor 1

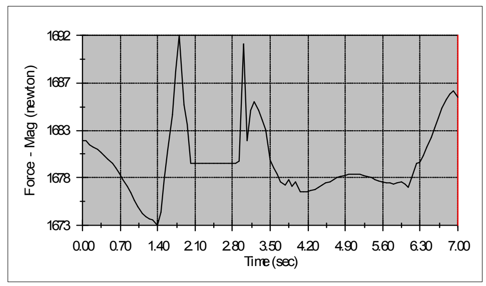


Motor 2

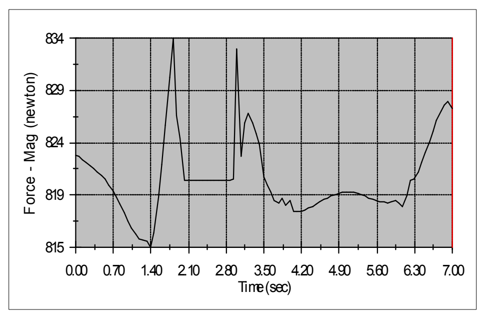


Motor 3

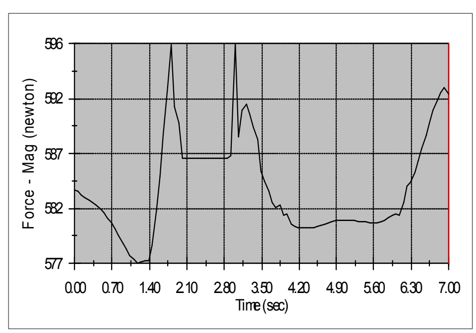
Reaction forces



Motor 1

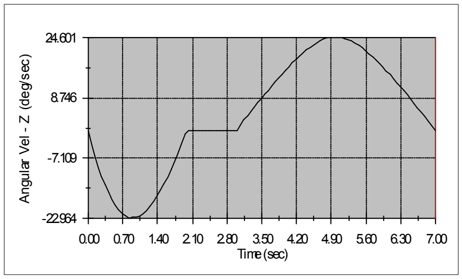


Motor 2

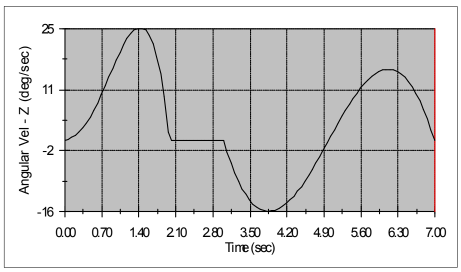


Motor 3

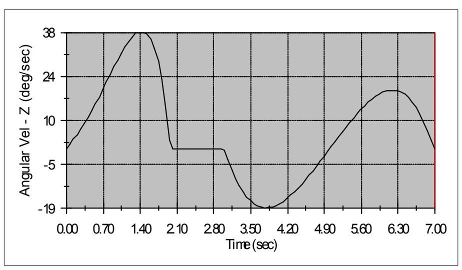
Motors angular velocities



Motor 1

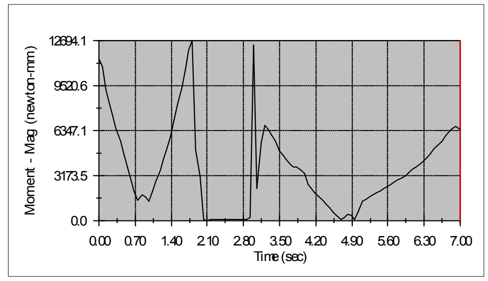


Motor 2

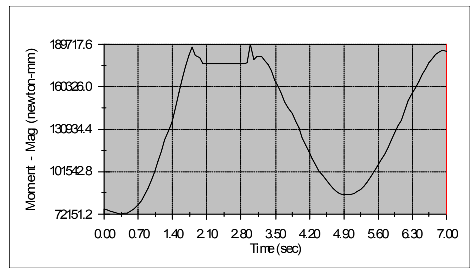


Motor 3

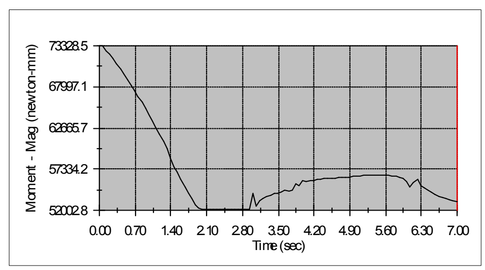
Motor torques



Motor 1



Motor 2



Motor 3