BİLGİSAYAR BİLİMLERİ VE TEKNOLOJİLERİ DERGİSİ
JOURNAL OF COMPUTER SCIENCE AND TECHNOLOGIES
http://dergipark.org.tr/tr/pub/bibted

Araştırma Makalesi

Industriel Robot Arm Rigit Dynamics Analysis Using Finite Element Method

Oğuzhan ÇAKAR*[1], Murat ALÇIN[2]

[1] *Afyon Kocatepe University, Graduate School of Natural and Applied Sciences , Mechanical Engineering, Afyon, Turkey*
[2] *Afyon Kocatepe University, Faculty of Technology , Mechatronics Engineering, Afyon, Turkey*

Keywords:
Manipulator
Polylactic Acid
Finite Element Method

ABSTRACT

In today's industry applications, robot and articulated manipulator systems are widely used because of their advantages in continuous operation, minimum cycle time and lowest error rate parameters. In this study, the Rigid Dynamic Analysis of the articulated manipulator, whose 2-D model was drawn, extracted with Polylactic Acid (PLA) raw material in a 3D printer and assembled in accordance with the tolerances, was carried out using the Finite Element Method (FEM) and the Angular Velocity parameter. As a result of the calculated analysis, Angular Velocity ω=0.1 rad/s and Total Deformation applied to the manipulator were obtained as 3.933 N. As a result of the calculated analysis, Angular Velocity applied to the manipulator was obtained as ω=0.1 rad/s, while Total Deformation was obtained as 3.933 N. According to the Rigid Dynamic Analysis results obtained from the study, the design was improved by using different raw materials in Ansys software. For different angular rotation parameters, the endpoint static displacements and the total deformation stresses on the manipulator when the industrial manipulator reaches different points in the three-dimensional working space are numerically calculated. As a result of using different raw materials, 1.5209 N was obtained for Aluminum.

Sonlu Elemanlar Yöntemi Kullanılarak Endüstriyel Robot Kol Rijit Dinamik Analizi

Anahtar Kelimeler:
Manipülatör
Polilaktik asit
Sonlu Elemanlar Yöntemi

ÖZET

Günümüz endüstri uygulamalarında robot ve mafsallı manipülatör sistemleri, sürekli çalışma, minimum çevrim süresi ve en düşük hata oranı parametreleri gibi avantajları nedeniyle yaygın olarak kullanılmaktadır. Bu çalışmada, 2 boyutlu modeli çizilen, Polilaktik Asit (PLA) hammaddesi ile 3 boyutlu yazıcıda ekstrakte edilen ve toleranslara uygun olarak montajı yapılan mafsallı manipülatörün Sonlu Elemanlar Metodu (Finite Element Method, FEM) ile ve açısal hız parametresini kullanarak, Rijit Dinamik Analizi (Rigid Dynamics Analysis) yapılmıştır. Hesaplanan analiz sonucunda manipülatöre uygulanan açısal hız ω=0.1 rad/s iken toplam deformasyon 3.933 N olarak elde edilmiştir. Çalışmadan elde edilen Rijit Dinamik Analiz sonuçlarına göre Ansys yazılımında farklı hammaddeler kullanılarak tasarım iyileştirilmiştir. Farklı açısal dönme parametresi için, endüstriyel manipülatörün üç boyutlu çalışma uzayı içerisindeki farklı noktalara erişmesi durumundaki uç nokta statik yer değiştirmeleri ve manipülatör üzerinde oluşan en büyük toplam deformasyon (Total Deformation) gerilmeleri sayısal olarak hesaplanmıştır. Farklı hammaddeler kullanımın sonucunda, Alüminyumda iken 1.5209 N olarak elde edilmiştir.

*Sorumlu Yazar

*(m.oguzhancakar@gmail.com) 0000-0001-7372-1368
(muratalcin@aku.edu.tr) 0000-0002-2874-7048

e-ISSN: 2717-8579

Geliş Tarihi: 01/05/2022; Kabul Tarihi: 26/07/2022

1. INTRODUCTION

Robot and manipulator systems are systems that are frequently used in today's industrial applications based on automation such as production, assembly, welding, spot welding, packaging, stacking, transport and press feeding. Speed in production, high precision and increase in the quality of the work done are the advantages of robot and manipulator systems. Today's modern engineering tools are used instead of classical methods in the design of these systems with complex geometries. The development of numerical calculations that produce approximate solutions with computer problems provides convenience in solving engineering problems that take a long time to solve analytically. One of these approximate solutions is the Finite Element Method (FEM) (Karagöz, 2010).

Ansys is one of the finite element software types that has been used frequently in the analysis of complex problems in recent years. Its success, especially in solving static stress and dynamic problems, is at a remarkable level. Besides, Ansys software allows engineers to perform many analyzes such as strength, vibration, fluid mechanics, and heat transfer (Kocabicak, 2001).With all this information, Ansys 2019 software was preferred for the Rigid Dynamics Analysis to be performed in this study.

Components and products produced through 3D printers are increasingly used in the automotive, biomedical, textile and food industries. Understanding the mechanical properties of the materials produced by this method is important for the efficient use of these materials in applications (Özerinç, 2018).

Polylactic Acid (PLA), a thermoplastic polymer made from lactic acid, was chosen as the raw material of the manipulator because it is biocompatible and biodegradable and stands out with its low cost (Çelik, 2017). It is one of the most preferred thermoplastics in the food packaging industry, personal care industry, kitchenware and biomedical industry (Oksman, 2013).

The technical features of the filament are; printing temperature is between 180-230℃. Its melting temperature is between 160-190 ℃. The brittleness temperature is between 60-65℃. It is the value at which the polymeric material loses its glass transition property and starts to show viscous properties in the fragility temperature range value (Aygın, 2019).

The reason for using PLA filaments is that they are the most commonly preferred filament types in three-dimensional printers that create the model with solid melt deposition technology. Another reason why this material is preferred is that it is easy to use because it can melt at low temperatures. The nozzle melting temperature is 180 ℃. There is no need to use a heating plate. Polylactic acid is obtained from products containing starch (Çelik, 2015). Mechanical and thermal properties of PLA material are given in Table 1 (Evlen, 2017).

Table 1. Mechanical and Thermal Properties of Polylactic Acid (Evlen, 2017)

PLA Features	Unit	Value	Method
Intensity	g/cm3	1,24	ISO 1183
Tensile Strength at Yield	MPa	70	ISO 527
Tensile Strength at Break	MPa	55-65	ISO 527
Melting point	°C	180	-
Flex Modulus	MPa	4500-5000	ISO 175

When the analyzes made by the Finite Element Method (FEM) and Analysis System (Ansys) software are examined in the literature, it is suggested that the raw material Polylactic Acid (PLA) be used in the components and products produced by 3D printers in the study by Kaygusuz. In this study, the effects of parameters such as nozzle temperature and filler density on the mechanical properties of structures produced with Polylactic acid, one of the commonly used 3D printer materials, were investigated. It was observed that the upper tensile strength increased as the nozzle temperature increased, while the strength and modulus of elasticity decreased gradually as the filler density decreased. As a result, nozzle temperature and filler density significantly affected the mechanical properties. As the nozzle temperature increased, the voids in the structure decreased and accordingly the tensile strength increased. It has been observed that both the yield strength and the modulus of elasticity decrease significantly when the filling density is reduced (Kaygusuz, 2018).

In his study, Evlen suggested that models modeled or scanned in 3D using computer aided design (CAM) can be produced from a 3D printer. In this study, it has been observed that factors such as fill rate, layer thickness, extruder temperature, ambient conditions have an effect on the part strength in the Melt Storage Modeling method, which is one of the parts printing methods with a 3D printer. As a result, the effect of fill rates on mechanical properties in 3D printers was investigated by tensile test and hardness measurement on the samples obtained in the open and closed conditions of the printer system. It has been observed that the hardness values of the samples printed in the closed system are lower than the samples printed in the open system, their tensile strength and % elongation are higher (Evlen, 2018).

In his study, Şık suggested using Ansys 14.0 software for simulation and fatigue analysis in virtual environment. In this study, it has been observed that surfaces operating under cyclic stresses develop cracks as a result of certain number of repetitions depending on time. As a result, it has been observed that various parameters such as vertical load, material hardness, fatigue strength and material ductility affect the results of the study (Şık, 2015).

In the study, Kaya suggested using the finite element method on Ansys software to examine the continuous contact problem in a constant-height homogeneous layer rigidly supported from the top and loaded with a rigid block. In this study, all surfaces are assumed to be frictionless and the homogeneous layer is loaded by a rigid circular block with external load P. As a result, the contact distances and contact stresses between the rigid block and the elastic layer were determined by the finite element method (Kaya, 2017).

In his study, Kıraç suggested the theoretical investigation of the dynamic behavior of composite linear rods under time-varying loads in Laplace space. In this study, the rod material is assumed to be homogeneous, linear elastic and anisotropic, and ordinary differential equations in scalar form obtained in Laplace space are numerically solved using complementary functions to calculate the dynamic stiffness matrix of the problem. As a result, a general purpose computer program was prepared in FORTRAN language to perform dynamic analysis of straight axis composite rods under time-varying loads, and it was observed that the results were in harmony with the results of the Ansys program (Kıraç, 2008).

In the study presented by Sivri, the non-linear behavior of reinforced concrete frames under horizontal loading with partial reinforced concrete shear additions has been made with Ansys software using the finite element method and STA4CAD program. In this study, nonlinear inelastic analyzes of two-storey and two-span weak frames were made in Ansys package program and STA4CAD software. As a result, the data of some experiments performed for reinforced concrete frames with weak seismic strength, reinforced by adding shear walls, were questioned with the results of nonlinear discrete analyzes (Sivri, 2013).

In his study, Karagöz suggested the use of Ansys software for static, dynamic, rigid dynamic and drop analyzes of the movements of the robot cabin and the stresses caused by the cabin weight by determining the material selection, load and boundary conditions. In this study, a robot operator cabin, which can move in specified angles and directions, is designed in order to increase the operator's field of view from inside the cabin. As a result, the three-dimensional original design of the mobile operator's cabin was made and its analysis has been performed with the Ansys WorkBench package program, which works with the finite element method (Karagöz, 2010).

In his proposed study, Kant performed static and dynamic analyzes for an industrial robot manipulator and completed the analysis with the finite element method. In this study, the endpoint static displacements and the maximum Von-Mises stresses on the manipulator when the robot manipulator reaches different points in the three-dimensional workspace for different workloads are numerically calculated. As a result, the endpoint displacement values for different positions and different working loads of an industrial robot in the workspace and the largest equivalent stress values on the manipulator were numerically obtained with a commercial software using the finite element method and the results were shown in the workspace (Kant, 2009).

In his study, Günal suggested Ansys software for the mechanical analysis of the robot arm mechanism, which includes six servo motors and one gripper. In this study, it is aimed to make analysis before the production of the building elements and to save cost and time by manufacturing according to the results of these analyzes. As a result, it includes the subjects of "design", "manufacturing-assembly" and "control". Robot arm mechanism consisting of 6 servo motors and 1 gripper claw; It was controlled by Atmega 328-P microcontroller card and by applying interface on Microsoft Visual C#. The design was made with SolidWorks and the mechanical analyzes were made using the Ansys program. As a result, the designed system was assembled and 6 servo motors were controlled on a platform (Günal, 2016).

In his study, Cengiz proposed the comparison and performance evaluation of bending, torsion and collision energy absorption capabilities of aluminum profile structures designed in different section geometries with the same cross-sectional area and external dimensions. In this study using Ansys LSDYNA software, geometries with the same contour dimensions have variable interior section designs with the same cross-sectional area amounts. The mechanical behavior of profile structures under different types of loads was investigated using the finite element method. As a result, undesirable situations may be inevitable if collision safety is not considered in designs to be made considering static loads or only bending and torsional loads. For this reason, the designer should refer to the collision analysis data as well as the bending and torsional loads, and even take into account natural frequency analyzes and system harmonics, which are the subject of future studies (Cengiz, 2017).

In this proposed study, Rigid Dynamic Analysis of the manipulator was performed using the Finite Element Method and Ansys WorkBench software. In the second part of the study, information about Finite Element Method and Rigid Dynamic Analysis in Ansys software is given. In the third chapter, the structure of Rigid Dynamic Analysis and the obtained graphs are presented. In the conclusion section of the study, the results obtained are interpreted.

When the analyzes made by the Finite Element Method (FEM) and Analysis System (Ansys) software are examined in the literature, it is suggested that the raw material Polylactic Acid

(PLA) be used in the components and products produced by 3D printers in the study by Kaygusuz. In this study, the effects of parameters such as nozzle temperature and filler density on the mechanical properties of structures produced with Polylactic acid, one of the commonly used 3D printer materials, were investigated. It was observed that the upper tensile strength increased as the nozzle temperature increased, while the strength and modulus of elasticity decreased gradually as the filler density decreased. As a result, nozzle temperature and filler density significantly affected the mechanical properties. As the nozzle temperature increased, the voids in the structure decreased and accordingly the tensile strength increased. It has been observed that both the yield strength and the modulus of elasticity decrease significantly when the filling density is reduced (Kaygusuz, 2018).

In his study, Evlen suggested that models modeled or scanned in 3D using computer aided design (CAM) can be produced from a 3D printer. In this study, it has been observed that factors such as fill rate, layer thickness, extruder temperature, ambient conditions have an effect on the part strength in the Melt Storage Modeling method, which is one of the parts printing methods with a 3D printer. As a result, the effect of fill rates on mechanical properties in 3D printers was investigated by tensile test and hardness measurement on the samples obtained in the open and closed conditions of the printer system. It has been observed that the hardness values of the samples printed in the closed system are lower than the samples printed in the open system, their tensile strength and % elongation are higher (Evlen, 2018).

In his study, Şık suggested using Ansys 14.0 software for simulation and fatigue analysis in virtual environment. In this study, it has been observed that surfaces operating under cyclic stresses develop cracks as a result of certain number of repetitions depending on time. As a result, it has been observed that various parameters such as vertical load, material hardness, fatigue strength and material ductility affect the results of the study (Şık, 2015).

In his study, two solid models were created from an industrial robot arm with three degrees of freedom whose solid modeling was designed (the model in which the drives are transmitted with the help of a belt pulley and the drives with the help of a screw shaft) are driven from different regions. These models suggested using rigid dynamic analysis separately for each model with the help of Ansys workbench interface (Şani,2021).

In the study, Kaya suggested using the finite element method on Ansys software to examine the continuous contact problem in a constant-height homogeneous layer rigidly supported from the top and loaded with a rigid block. In this study, all surfaces are assumed to be frictionless and the homogeneous layer is loaded by a rigid circular block with external load P. As a result, the contact distances and contact stresses between the rigid block and the elastic layer were determined by the finite element method (Kaya, 2017).

In his study, Kıraç suggested the theoretical investigation of the dynamic behavior of composite linear rods under time-varying loads in Laplace space. In this study, the rod material is assumed to be homogeneous, linear elastic and anisotropic, and ordinary differential equations in scalar form obtained in Laplace space are numerically solved using complementary functions to calculate the dynamic stiffness matrix of the problem. As a result, a general purpose computer program was prepared in FORTRAN language to perform dynamic analysis of straight axis composite rods under time-varying loads, and it was observed that the results were in harmony with the results of the Ansys program (Kıraç, 2008).

In the study presented by Sivri, the non-linear behavior of reinforced concrete frames under horizontal loading with partial reinforced concrete shear additions has been made with Ansys software using the finite element method and STA4CAD program. In this study, nonlinear inelastic analyzes of two-storey and two-span weak frames were made in Ansys package program and STA4CAD software. As a result, the data of some experiments performed for reinforced concrete frames with weak seismic strength, reinforced by adding shear walls, were questioned with the results of nonlinear discrete analyzes (Sivri, 2013).

In his study, Karagöz suggested the use of Ansys software for static, dynamic, rigid dynamic and drop analyzes of the movements of the robot cabin and the stresses caused by the cabin weight by determining the material selection, load and boundary conditions. In this study, a robot operator cabin, which can move in specified angles and directions, is designed in order to increase the operator's field of view from inside the cabin. As a result, the three-dimensional original design of the mobile operator's cabin was made and its analysis has been performed with the Ansys WorkBench package program, which works with the finite element method (Karagöz, 2010).

In his proposed study, Kant performed static and dynamic analyzes for an industrial robot manipulator and completed the analysis with the finite element method. In this study, the endpoint static displacements and the maximum Von-Mises stresses on the manipulator when the robot manipulator reaches different points in the three-dimensional workspace for different workloads are numerically calculated. As a result, the endpoint displacement values for different positions and different working loads of an industrial robot in the workspace and the largest equivalent stress values on the manipulator were numerically obtained with a commercial software using the finite element method and the results were shown in the workspace (Kant, 2009).

In his study, Günal suggested Ansys software for the mechanical analysis of the robot arm mechanism, which includes six servo motors and one gripper. In this study, it is aimed to make analysis before the production of the building elements and to save cost and time by manufacturing according to the results of these analyzes. As a result, it includes the subjects of "design", "manufacturing-assembly" and "control". Robot arm mechanism consisting of 6 servo motors and 1 gripper claw; It was controlled by Atmega 328-P microcontroller card and by applying interface on Microsoft Visual C#. The design was made with SolidWorks and the mechanical analyzes were made using the Ansys program. As a result, the designed system was assembled and 6 servo motors were controlled on a platform (Günal, 2016).

In his study, Cengiz proposed the comparison and performance evaluation of bending, torsion and collision energy absorption capabilities of aluminum profile structures designed in different section geometries with the same cross-sectional area and external dimensions. In this study using Ansys LSDYNA software, geometries with the same contour dimensions have variable interior section designs with the same cross-sectional area amounts. The mechanical behavior of profile structures under different types of loads was investigated using the finite element method. As a result, undesirable situations may be inevitable if collision safety is not considered in designs to be made considering static loads or only bending and torsional loads. For this reason, the designer should refer to the collision analysis data as well as the bending and torsional loads, and even take into account natural frequency analyzes and system harmonics, which are the subject of future studies (Cengiz, 2017).

In this proposed study, Rigid Dynamic Analysis of the manipulator was performed using the Finite Element Method and Ansys WorkBench software. In the second part of the study, information about Finite Element Method and Rigid Dynamic Analysis in Ansys software is given. In the third chapter, the structure of Rigid Dynamic Analysis and the obtained graphs are presented. In the conclusion section of the study, the results obtained are interpreted.

2. METHOD

2.1. Finite Element Method

Developed and developing countries want high quality, low cost and minimum cycle time parameters for products produced with developing industrial technology (Karataş et al., 2021, Koyuncu et al., 2021). For this reason, the power of robot and manipulator systems is preferred instead of human power for many industrial applications.

The finite element method is used in the analysis of engineering problems that have no solution or are difficult to solve with normal methods in practice. The finite element method was first developed in 1956 for the analysis of airframes, and in the next decade it began to be used in solving applied science and engineering problems. In the following years, this method and solution techniques were developed rapidly and today it has become one of the best methods used for the solution of many engineering problems.

In this study, the manipulator (Çakar, 2020), whose 2D model is designed in the computer aided drawing program, is called to the Ansys software environment and analyzed. First of all, the model made in the computer aided drawing program was saved as .x_t extension suitable for Ansys software. Ansys WorkBench platform was opened and Rigid Dynamic Analysis parameter was dragged from toolbox and left to project schematic to perform Rigid Dynamic Analysis of the designed manipulator.

PLA raw material, which is the raw material of the designed manipulator, was selected from the Engineering Data button. In order for the Rigid Dynamic Analysis results to be obtained correctly, raw materials were selected. According to the analysis results, improvements were made in the design. After the raw material selection, the 2D manipulator design saved as .x_t extension was called from the geometry button to the project diagram section. The view of the imported 3D Manipulator is given in Figure 1.

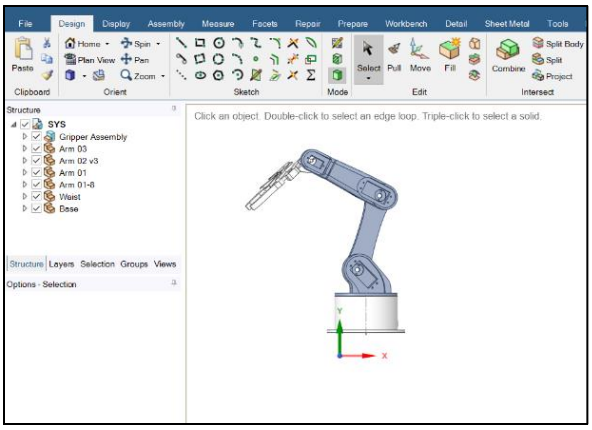

Figure 1. The view of the imported 3D Manipulator

Axis-axis connections are made from the connection parameter of the imported 2D manipulator design. The reason for making the axis-axis connection is to determine the mounting locations in the Ansys software and to enable the analysis to be performed according to the mounting locations. A total of 12 connections were made for the manipulator with a 2D design. One of the 12 connections is Fixed-Ground connection, the rest is Revolute-Axes connection. Since the plinth is fixed to the floor from its lowest point in the experimental study, the ground is fixed from the lowest point in Ansys software. Base fixed floor connection is illustrated in Figure 2.

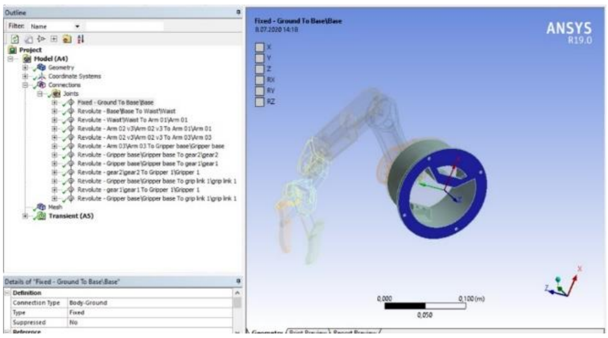

Figure 2. Base fixed floor connection

Rotary axis connection is selected from axis-axis connection parameter in model section. The reference point (Reference-Body) was selected as base and the 1st axis was selected as mobile and its connection was completed by connecting the axes with each other in such a way that the rotation direction was the Z axis.

2.2. Rigid Dynamic Analysis

Rigid Dynamic Analysis is a type of analysis performed by assuming that all the elements of the designed structure are rigid. Therefore, as a result of the analysis, no stress or strain can be obtained in the structure, only the total deformation can be obtained. As a result of parameters such as angular rotation and force applied to the manipulator, it was carried out to determine the angular rotation analysis values that are planned to come to the connection points of the manipulator.

In this study, the rigid dynamic analysis of the manipulator was carried out with the angular rotation parameter. The manipulator, produced with traditional methods, was modeled in the SolidWorks 2018 program. The prototypical manipulator has been jointed and has 5 axes. The designed manipulator was printed using PLA filament, with the 3D printer closed. The analyzes were made after the printed manipulator parts were assembled in accordance with their tolerances and meticulously.

An angular velocity of ω=0.1 rad/s was given to all connection points of the manipulator in the counterclockwise direction, and the analysis was carried out for 60 s and a full rotation was achieved. An analysis time of 60 s is required for the manipulator, which has an angular velocity of ω=0.1 rad/s, to provide one full turn (2π radian). Equations proving that the time required for the manipulator to produce 2π radian rotations is 60 s, are shown in Eq. (1), Eq. (2), Eq. (3) and Eq. (4).

$$\theta = \omega \cdot t \tag{1}$$

$$2\pi = \omega \cdot t \tag{2}$$

$$\omega = 2\pi / t \tag{3}$$

$$\omega = 2\pi f \tag{4}$$

The forces acting on the 1st axis of the manipulator whose Rigid Dynamic Analysis is performed are shown in Figure 3.

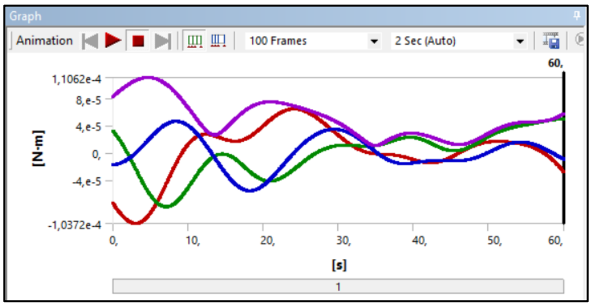

Figure 3. Total forces acting on the 1st axis of the manipulator

Red represents the force applied on the X axis, green represents the force applied on the Y axis, blue represents the force applied on the Z axis and purple represents the total force applied on all axes. According to Figure 3, the highest force value is the total force shown in purple, with a value of 1.1062 N.

Figure 4. presents the forces acting on the 2nd axis of the manipulator whose Rigid Dynamic Analysis is performed. According to Figure 4, the highest force value is the total force shown in purple, with a value of 9,3776 N.

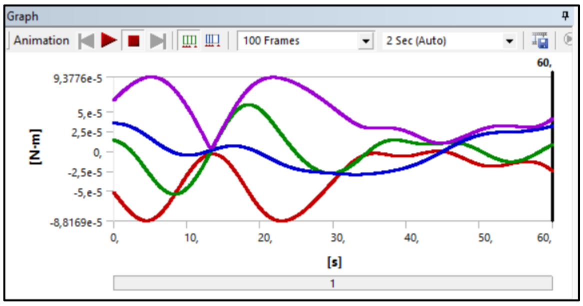

Figure 4. Total forces acting on the 2nd axis of the manipulator

Figure 5. gives the forces acting on the 3rd axis of the manipulator whose Rigid Dynamic Analysis is performed. According to Figure 5, the highest force value is the total force shown in purple, with a value of 4,6699 N.

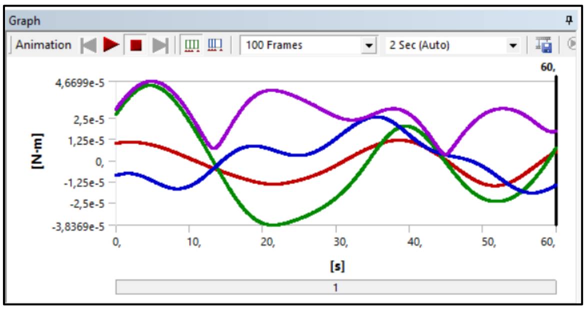

Figure 5. Total forces acting on the 3rd axis of the manipulator

Figure 6. shows the forces acting on the 3rd axis of the manipulator whose Rigid Dynamic Analysis is performed. According to Figure 6, the highest force value is the total force shown in purple, with a value of 1,4661 N.

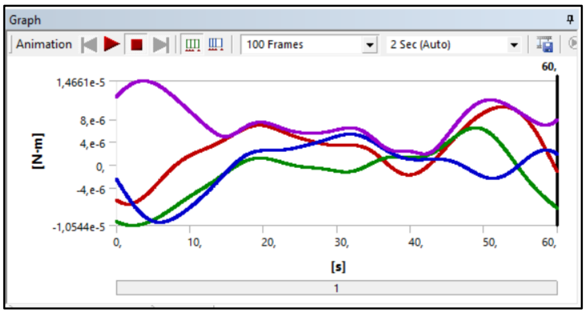

Figure 6. Total forces acting on the 4th axis of the manipulator

Figure 7. shows the forces acting on the 5th axis of the manipulator whose Rigid Dynamic Analysis is performed. According to Figure 7, the highest force value is the total force shown in purple, with a value of 1,3521 N.

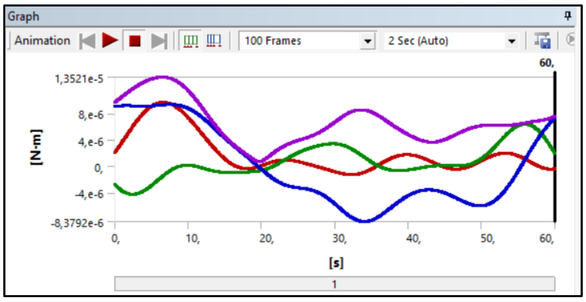

Figure 7. Total forces acting on the 5th axis of the manipulator

The forces acting on the 1st gear group of the holder are shown in Figure 8. The highest force value is the total force shown in purple, with a value of 3.7565 N.

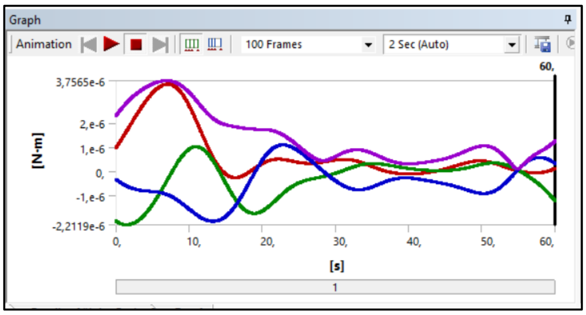

Figure 8. Total forces acting on the 1st gear group of the gripper (gearbox)

The forces acting on the 2nd gear group of the holder are shown in Figure 9. The highest force value is the total force shown in purple, with a value of 3.1535 N.

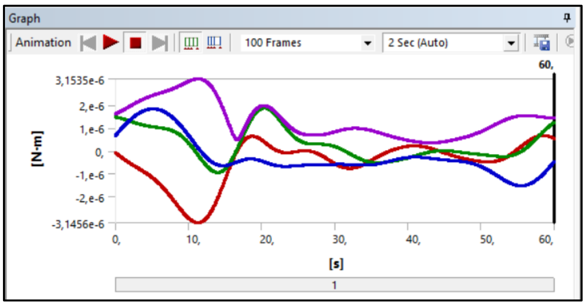

Figure 9. Total forces acting on the 2nd gear group of the gripper (gearbox)

3. FINDINGS

In the analysis made within the scope of the study, the Rigid Dynamic Analysis differences made using polylactic acid and aluminum raw materials are shown in Table 2.

Table 2. Acquired Rigid Dynamic Analysis Results

Axes	Unit	PLA Value	Aluminum Value
1	N	1,106	3,131
2	N	9,377	2,654
3	N	4,669	1,321
4	N	1,466	4,149
5	N	1,352	3,827
1st gear group	N	3,756	1,062
2nd gear group	N	3,153	1,251

As a result of the analysis, when aluminum is used in the engineering data, it has been observed that the 2nd and 3rd axis total moment values, where the most deformation is experienced, decrease.

4. CONCLUSION

One of the most important features of package programs that analyze using the finite element method is that they can be displayed with curves and colors showing the analysis values obtained.

Basic elements such as high speed, precision, repeatability and low energy consumption required in today's industrial applications are the basic elements that should be considered and provided in the design of robot and manipulator systems. Providing the desired performance values of robot and manipulator systems with complex geometries and high operating speeds is a goal that is difficult to reach with classical design methods. In order to eliminate this difficulty and to obtain a suitable manipulator that will meet the expectations of the industry, it has been designed with the help of Computer Aided Design (CAD) programs, and Rigid Dynamic Analysis has been performed in Ansys software by printing it in a 3D printer, which is one of the modern production methods.

In this study, different angular rotation values of an industrial manipulator in the working space and the maximum equivalent stress values on the manipulator were obtained numerically in the finite element method and the results were shown in the working space. While Ansys was performing the analyses, the designed manipulator was accepted as rigid and no flexibility definition was made. It has been shown that the results found are in agreement with both the literature and the Ansys results. In the rigid dynamic analysis performed, the forces on the motors and axes during the motion of the manipulator were obtained in Newton meters.

The maximum force value occurring in the X, Y and Z axes from the rotational forces realized in the manipulator joints is given in the Table 2 in the findings section of the performed rigid dynamic analysis. Thus, a rigid dynamic analysis was obtained with two different force values for both raw materials.

REFERENCES

Aygın, M., Yıldırım, F., Çantı, E., "Farklı Yazdırma Parametrelerinde Pla Filamentin İşlem Performansının İncelenmesi", Internatıonal Journal Of 3d Prıntıng Technologıes And Dıgıtal Industry, 3:2, 102-115, 2019.

Cengiz A., ''Statik ve Dinamik Yük Altındaki Çekme Alüminyum Profillerde Kesit Alan Tasarımının Mekanik Davranışa Etkisi'', Afyon Kocatepe Üniversitesi Fen ve Mühendislik Bilimleri Dergisi, 264-273, 2017.

Çakar O., Alçın M., Koyuncu İ., Tuna M., ''Endüstriye Tabanlı Yeni Bir Robot Kol Tasarımı'', 4. Uluslararası Asya Çağdaş Bilimler Kongresi, 578-589, 2020.

Evlen H., Erel G., ve Yılmaz E., ''Açık ve kapalı sistemlerde doluluk oranının parça mukavemetine etkisinin incelenmesi'', Politeknik Dergisi, 21(3): 615-662, 2018.

Günal A., ''6 Eksenli Robot Kol Tasarimi ve Kontrolü'', Lisans Tezi, Niğde Üniversitesi Mühendislik Fakültesi Mekatronik Mühendisliği, 2016.

Kant Y., ''Bir Robot Manipülatörün Bilgisayar Destekli Mühendislik Araçları ile Çalışma Uzayı Analizi'', Yüksek Lisans Tezi, Dokuz Eylül Üniversitesi Fen Bilimleri Enstitüsü, 2009.

Karagöz M., ''Mobil Vinç Robot Kabinin Tasarımı ve Analizi'', Yüksek Lisans Tezi, Selçuk Üniversitesi Fen Bilimleri Enstitüsü, 2010.

Karataş F., Koyuncu İ., Tuna M., Alçın M., Avcioglu E., Akgul A. (2021), ''Design and implementation of arrhythmic ECG signals for biomedical engineering applications on FPGA, Eur. Phys. J. Spec. Top., 181 1-16, https://doi.org/10.1140/epjs/s11734-021-00334-3.

Kaya Y., ''Rijit Olarak Mesnetlenmiş Homojen Tabakada Sürekli Temas Probleminin Sonlu Elemanlar Yöntemi ile Analizi'', International Conference on Advanced Engineering Technologies, 2017.

Kaygusuz, B., ''3 Boyutlu Yazıcı ile Üretilen PLA Bazlı Yapıların Mekanik Özelliklerinin İncelenmesi'', Makina Tasarım ve İmalat Dergisi, 2018.

Kıraç M., Çalım F. F., ''Doğru Eksenli Kompozit Çubukların Dinamik Analizi'', Çukurova Üniversitesi Mühendislik-Mimarlık Dergisi, 23(1), 2008.

Koyuncu İ., Rajagopal, K., Alcin M., Karthikeyan A., Tuna M., Varan M., Karagöz M. (2021), ''Control, synchronization with linear quadratic regulator method and FFANN-based PRNG application on FPGA of a novel chaotic system'', Eur. Phys. J. Spec. Top., 230 (7), 1915-1931. https://doi.org/10.1140/epjs/s11734-021-00178-x.

Koyuncu İ., Rajagopal K., Alcin M., Karthikeyan A., Tuna M., Varan M., "Control, synchronization with linear quadratic regulator method and FFANN-based PRNG application on FPGA of a novel chaotic system", The European Physical Journal Special Topics, 230-7, 1915-1931.

Oksman K., Skrifvars M., Selin J.-F., "Natural fibres as reinforcement in polylactic acid (PLA) composites", Composites Science and Technology Issue 63, 1317–1324, 2003.

Sivri M., ''Perde Duvar ile Güçlendirilen Betonarme Çerçevenin Ansys ve STA4CAD Analiz sonuçlarının Karşılaştırılması'', Süleyman Demirel Üniversitesi Teknik Bilimler Dergisi, 3(2), 26-34, 2013

Şani İ., "Endüstriyel Bir Robotun Farklı Tahriklerde Oluşan Titreşimlerin Ölçülmesi ve Karşılaştırılması", Batman Üniversitesi Lisansüstü Eğitim Enstitüsü, 2021.

Şık A., Önder M., Korkmaz S., ''Taşıt Jantlarının Yapısal Analiz ile Yorulma Dayanımının Belirlenmesi'', Gazi Üniversitesi Fen Bilimleri Dergisi, 3:3, 565-574(2015).