Modeling and Analysis of a Novel Smart Knee Joint Prosthesis for Transfemoral Amputation

SH H Kamel¹, M N Hamzah¹, Q A Atiyah¹ and S A Abdulateef²

¹ Mechanical Engineering Dept., University of Technology, Baghdad, Iraq
² College of Medicine, University of Al-Iraqia, Baghdad, Iraq

E-mail: 20299@uotechnology.edu.iq

Abstract. The absence of a limb impacts devastatingly on any person, especially if it is the lower limb as it is paramount to human locomotion. The effect of mobility loss reduces independence, and affects amputees’ lifestyles. A smart prosthetic knee joint is designed and manufactured in which the amputee above the knee can perform various daily and effective movements. It is distinguished by its distinctive and very high efficiency mechanically and electronically. The new design of the artificial smart knee joint has proven to be 100% successful as a passive mechanical movement. Should a power failure occur at the source, the ability to move smoothly is very high, proved highly efficient by walking, sitting, and applying various daily movement activities. The artificial knee joint is able to detect the movement of the residual limb according to the results of the movement study, thus allowing the prosthesis to simulate the biomechanics of the missing limb without any difficulty by using the finite element method (FEM) as a numerical technique. The results showed that 94.303 MPa and 0.1379 mm are the highest stresses and displacements experienced by the knee joints, respectively at 110°. This remains below the yield stresses of 339.15 MPa (depending on the properties of aluminum alloy material 2024-T3). On the other hand, the highest safety of factors was at 0° with a value of 5.5905 and the lowest safety of factors was at 110° with a value of 3.5964. The above results show that von Mises stresses and displacement increase with increasing the angle of flexion of the knee joint while the safety of factors decreases as the angle of flexion of the knee joint increases. The result of finite element analysis shows that the concept is safe enough to use for this specific topic.

Keywords. Smart knee joint, Finite element method, Von Mises stresses, Safety of factors, Topology, Optimization and model.

1. Introduction
The absence of a limb has a devastating effect on any person, especially if it is the lower limb, as this is necessary for human locomotion. Thousands of people around the world lose their lower limbs every year because of circulatory and vascular issues, diabetic complications, cancer, or injuries. The effect of mobility loss decreases autonomy, impacting the high quality of lifestyles of amputees. The exact number of lower limb amputees worldwide is difficult to obtain since many countries do not maintain a list of amputees or the cause of amputations [1]. The only sources available for individuals who lost their lower limbs were walkers, wheelchairs, wooden peg legs, and crutches in the past. However, to take advantage of advances in medical science and technology, people with this type of disability can now use motorized lower limb prosthesis [2]. Xie and Kang [3] studied the design and
control simulation of trans-femoral prosthesis based on a virtual prototype. Karthikeya [4] based his investigation on presenting a lower prosthesis made with the help of a structure that uses a common worm gear. This structure is used by placing a normal worm gear at the knee joint, generally known as prosthetic, to assist in the motion of the leg. This mechanism is carried out using a battery of 12 volts, a 60-rpm motor, a worm gear, and a spur gear. Galbally [5] used a series of elastic actuators with force control. This model focused on developing a prosthesis for above-knee amputees, using one active joint at the knee, and locating a passive joint at the ankle. Furthermore, the socket was designed using compliance mirroring to ensure optimum patient comfort. This study aims to enable the prosthesis without any difficulty to simulate the biomechanics of the lost limb using the finite element method (FEM) and its ability to withstand high loads and pressures to perform successful daily movements without movement failure and to achieve successful movement of the joint mechanically and electronically without the need for power to ensure joint movement.

2. Design novel prosthetic smart knee joint
The design of the knee joint is based on the idea that the single axis of the knee joint acts as an articulated joint and enables flexion and extension through rotation around one axis. Uniaxial knee joints perform a basic rotation around the pivot of the knee axis during flexion and extension. They are clearly built and, according to mechanical principles, are easily synchronized. The dimensions of the artificial active knee joint are constructed in standard dimensions to be similar to the dimensions of the knee joint used to preserve the essential mechanical dimensions of the knee joint on traditional prostheses above the knee. The artificial active knee joint has three parts, the upper part, the lower part, and bearings with a pin. A single screw with needle bearings, as shown in Figure (1), connects the lower and upper sections. Needle bearings aim to reduce a rotating surface friction. Needle bearings have a wider surface area in contact with the race compared to ball bearings and ordinary roller bearings, so they can accommodate a higher load. They are smaller as well because they need less clearance between the axle and the frame around it. In this work, the modeling of the new knee joint model is designed using SolidWorks 2018. The goal of drawing models through SolidWorks is to be used for modeling, meshing, and determining boundary conditions such as applied load in the ANSYS workbench software 2019. The view of the software for Solidworks 2018 can be viewed as a smart assembly drawing of the knee joint in Figure (2).

![Figure 1. Components of the Knee joint.](image-url)
3. Finite element analysis

Finite Element Analysis (FEA) consists of a computer model of a material or design that is stressed and tested for precise results. It is used to create new products and to improve current products. In other words, FEA is a numerical approach for finding an approximate solution to variables in a problem which is difficult to obtain analytically. Potential changes to the design are measured, such as temperature, buckling, stress, and deflection. Studying finite elements is a theoretical approach that can solve these engineering problems. FEA may be used to help determine design changes in the event of structural model failure to satisfy the new situation [6-8].
4. Analysing knee joint model

4.1. Mesh of model

The important requirement of the FEM is the need to split the solution domain (model geometry) into subdomains that are simply formed, called 'finite elements. This is a process of discretization typically called meshing, and the element is called finite with infinite numbers of degrees of freedom due to its finite, rather than the infinitesimally small size. The meshing process was performed by selecting the geometry and then, as shown in Figure (3), the shape of the element containing the total number of elements was (293024 elements) with a total number of (454821) nodes.

![Image of mesh model](image)

**Figure 3.** Mesh of the knee joint Model.

4.2. Topology optimization

Optimization of topology is a useful feature in FEM software that allows an engineer to recognize overdesigned or redundant parts in an assembly to meet structural requirements. Optimization of topology defines regions of a structure where the material density can be reduced and thus provides space for optimized design. ANSYS Workbench 19.0 FEM is the platform used here. Optimization of topology makes it possible to define where supports and limitations are located on a material number, the optimization region, and the constraints to meet the desired requirements. The structural design problems tend to be constrained by performance constraints identified by specific design objectives. It is an iterative process that typically needs multiple iterations to enter a design solution that meets the performance requirements and meets the stated requirements using finite element analysis. Topology optimization is a modern approach to the computational methods used for a given set of design constraints to determine the optimum shape and material distribution within a given design area. Topology optimization outcomes do not inherently need to be the best solutions, but they provide a fundamental basis for optimal optimization of the design solution. In this case, the optimization target was set to be a mass reduction, with the response limit being the percentage of mass reduction. The proportion of mass reduction varies each time for an iterative process. The original design space was the lower structure of the aluminum-made knee joint prosthesis. On this block, topology optimization was performed to reduce compliance and restriction as a percentage reduction in mass. The Finite Element Topology Optimization is followed by a process of geometric interpretation, as the results of the topology studies are merely rough geometric proposals and some interpretation is needed to construct the final design. In other words, many iterations were treated as a variable (with 90 percent, 80 percent, and 60 percent mass reduction) to define regions where material density is needed and several iterations were conducted to gain a better understanding of the environment of criticality. The results of this optimization are shown in Figure (4). [9], and [10].
4.3. Applying boundary condition

Finite-element analysis (FEA) is carried out to optimize and verify every step of the design. Based on FEA performance, the product consistency, efficiency, and safety are also assured. The relationship between the upper part and the actuator is determined by displacement, protection factor, and stress of the components on top of the knee joint under 5000N loads. The finite-element method was used on the SolidWorks platform and the lower part was fixed which is connected to the pylon, and compression support only on the screw. The FEA research of components of the knee joint is carried out by stress analysis, displacement, and safety factors. The safety of factor (FoS) is usually a design objective for this form of research. To check the functionality of the joint intended for a prosthetic, a finite-element model of the knee joint was created. Several of the components are referred to as body solids. The aluminum alloy 2024-T3 is chosen for all parts except for the needle bearing and bushing pin, for which 10.9-grade alloy steel (SS) and ISO 898 are chosen. The Figure (5) shows the load conditions applied to the manufacture of knee joint.

Figure 4. Result of topology optimization.
Figure 5. Applying Load on the knee joint by the ANSYS Workbench – 19 Program.

The properties of the materials used to produce the knee joint are defined by many other essential parameters. To complete the overall results of this model, including Young's module, Poisson ratio, density, bulk module, shear module, yield, and ultimate stress that will be obtained from the mechanical tests of the proposed materials, were used in the production of the knee joint as described in the Table (1).

Table 1. Material of properties.

| Property            | AL 2024 T3  | ISO 898 grade 10.9 |
|---------------------|-------------|---------------------|
| Density             | 2780 kg/m³  | 7800 kg/m³          |
| Young's modulus     | 73000 MPa   | 210000 MPa          |
| Poisson's ratio     | 0.33        | 0.3                 |
| Bulk Modulus        | 71569 MPa   | Bulk Modulus        |
| Shear Modulus       | 27444 MPa   | 80769 MPa           |
| Tensile Yield Strength | 339.15 MPa | Tensile Yield Strength |
| Tensile Ultimate Strength | 429.7 MPa | Tensile Ultimate Strength |

Results analysis was performed using ANSYS Workbench - 19 software for materials and for various knee flexion angles. The stress analysis of Von Mises, the displacement analysis, and the safety factor for different knee flexion angles were conducted in three types of analysis.

5. Results and discussions

5.1. Results and discussions on finite-element analysis
In the finite-element analysis, the knee components are checked for stress, safety of factor, and displacement to verify the viability of the design. The maximum von Mises stress is measured as a means of design safety in most engineering designs. The maximum stress of von Mises is derived from von Mises-Hencky, often referred to as the principle of shear-energy or maximum energy distortion. According to the theory, when the von Mises stress and stress limit become equal, the yield of a ductile material begins. The force of yield is, in general, taken as the limit of stress.

$\sigma_{\text{vonMises}} \geq \sigma_{\text{limit}}$

Figures from Figure (6) to Figure (11) show von Mises stress of various knee components found from the finite-element analysis. The primary focus of the von Mises stress analysis is the state of stability.
The findings of the stress analysis of von Mises indicate that the maximum stress distribution experienced by the prosthetic knee joint for different knee angles ranged from 8.918 E-015 MPa to 60.665 MPa at 0°, from 8.5691E-015 MPa to 63.068 MPa at 30°, from 8.9001E-015 MPa to 67.626 MPa at 45°, from 9.5861E-015 MPa to 74.894 MPa at 60°, from 1.1438E-014 MPa to 94.057 MPa at 90°, and from 0 MPa to 94.303 MPa at 110°. This remains below the maximum material yield strength of 339.15 MPa (depending on the material properties of aluminum alloy 2024-T3). The aforementioned results show that the von Mises stresses increase with increasing the flexion knee joint angle. The importance of the yield strength is the stress value of the material without any deformation.

To calculate the shear stress in the pin:

$$\tau_{pin} = \frac{F}{2A} = \frac{F}{2\times\pi d^2/4} = \frac{5000}{2\times\pi\times10^2/4} = 31.8309\, MPa$$

(1)

The maximum shear stresses:

$$\tau_{max} = \frac{\sigma_{yield}}{2} = \frac{1020}{2} = 510\, MPa$$

(2)

This is correct, which indicates that the design is safe and secure for knee joint movement after applying the maximum load.
The figures and table show the outcomes of static displacement studies. The results of the static displacement analysis indicate that for different knee angles, the maximum static displacement distribution encountered by the prosthetic knee joint differed between different knee angles ranged from 0 mm to 0.0401 mm at 0°, from 0 mm to 0.046107 mm at 30°, from 0 mm to 0.054114 mm at 45°, from 0 mm to 0.066968 mm at 60°, from 0 mm to 0.10999 mm at 90°, and from 0 mm to 0.1379 mm at 110°. The aforementioned results show that the displacement increases with increasing the flexion knee joint angle. The displacement analysis for the model of the knee joint is shown in Figure (12) to Figure (17), where the color refers to the varying displacement value distributed through the knee joint. The region has zero displacements since the bottom of the prosthetic knee joint is set to be fixed. In the colored red/orange regions, the displacement is high, distributed in the top regions for the material Aluminum Alloy 2024-T3. From the observation of the figures and the table that displays the results of calculating the displacement using the finite element method, all the results for different angles of the knee joint showed that the displacement increases with the increase of the angle of the knee joint.
When assessing the outcomes of static stress analysis, allowable stress values can be defined and then the safety of factor contours can be shown where the stresses in the model are below and above those permitted. Viewing the protection of factor contours will help one determine whether a design needs to be changed or is sufficient and ready for production. The safety of factor is the ratio of the allowable stress to the actual stress. To measure the safety of factor at a certain location, the following equation will help:

\[
\text{Factor of safety FoS} = \frac{\sigma_{\text{limit}}}{\sigma_{\text{von Mises}}}
\]

At 0 deg., the safety of factor was 339.15/60.665 = 5.5905. For various knee angles from 0 deg to 110 deg, it is measured with the same rule, which is good enough to ensure the design safety. From the observation of the shapes and the table that displays the results of calculating the safety factor using the finite element method, all the results are shown for different angles of the knee joint that it is greater than one and this is what achieves the design success. From the observation of the Figure (18) to Figure (23) and the Table (2) that displays the results of calculating the safety factor using the finite element method.
Figure 18. The Safety factor analysis of Aluminum Alloy 2024-T3 (0 Deg).

Figure 19. The Safety factor analysis of Aluminum Alloy 2024-T3 (30 Deg).

Figure 20. The Safety factor analysis of Aluminum Alloy 2024-T3 (45 Deg).

Figure 21. The Safety factor analysis of Aluminum Alloy 2024-T3 (60 Deg).

Figure 22. The Safety factor analysis of Aluminum Alloy 2024-T3 (90 Deg).

Figure 23. The Safety factor analysis of Aluminum Alloy 2024-T3 (110 Deg).
Table 2. The Von Mises stress analysis summary, deformation analysis, and safety factor analysis

| Material | Al 2024-T3 | Von Mises stress analysis (MPa) | Deformation analysis (mm) | Safety factor |
|----------|------------|--------------------------------|---------------------------|---------------|
|          | Min       | Max                            | Yield Stress              | Min           | Max           | Min | Max |
| 0°       | 8.918E-015| 60.665                         | 339.15                    | 0             | 0.0401        | 5.5905 | 15 |
| 30°      | 8.5691E-015| 63.068                         | 339.15                    | 0             | 0.046107      | 5.3775 | 15 |
| 45°      | 8.9001E-015| 67.626                         | 339.15                    | 0             | 0.054114      | 5.0151 | 15 |
| 60°      | 9.5861E-015| 74.894                         | 339.15                    | 0             | 0.066986      | 4.5284 | 15 |
| 90°      | 1.1438E-014| 94.057                         | 339.15                    | 0             | 0.10999       | 3.6058 | 15 |
| 110°     | 0         | 94.303                         | 339.15                    | 0             | 0.1379        | 3.5964 | 15 |

From the finite element analysis, the maximum stress from von Mises stresses, the safety of factor, and the part displacement that occurred during various movements of the smart artificial knee joint movement is found to remain far below the strength of material yield, safety factor, and displacement limits. Consequently, the suggested design is suitable for the specific application and no unexpected failure can occur. The smart knee joint designed for a transfemoral amputee can simulate the various activities of a healthy biological limb on a daily basis and can be used with sufficient protection. According to the findings of the motion analysis, the prosthetic knee joint is able to sense residual limb movement and thus helps the prosthesis to imitate the biomechanics of the missing limb without any difficulty. 94.303 MPa and 0.1379 mm are the highest stresses and displacements experienced by the knee joints, respectively at 110°. This remains below the maximum material production resistance of 339.15 MPa (depending on the properties of aluminum alloy material 2024-T3). While the highest safety of factors was at 0° with a value of 5.5905 and the lowest safety of factors was at 110° with a value of 3.5964. The above results show that von Mises stresses and displacement increase with increasing the angle of flexion of the knee joint while the safety of factors decreases as the angle of flexion of the knee joint increases. Limits well below the acceptable material limits were used to make components. The outcome of the study of finite elements shows that the concept is secure enough to use for this particular subject. The results obtained from the F.E study were then compared with the actual data for a similar topic obtained from the literature and with the real production of smart knee joints. The pattern of motion analysis results showed great similarity to a stable biological limb's different everyday activities.

5.2. The test is performed on the above-knee amputee wearing a smart artificial knee joint and alignment adjustment

After designing and manufacturing the smart artificial knee joint and linking the parts, the knee joint was ready to be worn by the amputated patient over the knee and before starting the movement through the controls where the alignment and mechanical movement of the amputee above the knee was 185 cm long, 34 years old, weighed 85 kg, and the level of amputation was short and considered the most difficult case of amputation. The level of amputation should give the amputee the maximum function and weight-bearing area possible. A general rule is that the residual limb should be as long as possible, but there should always be approximately 12 cm from the end of the stump to the knee to allow space to fit a prosthetic knee joint. The level of the amputation is above the knee about 15 cm from the end of the amputation to the middle of the knee as shown in the following Figure (24):

Figure 24. Level of above knee amputation.
However, it proved highly efficient by walking, sitting, and applying various daily movements activities. Test of the movement is a mechanical movement only. The new design of the artificial smart knee joint has proven to be 100% successful as a passive mechanical movement. Should a power failure occur at the source, the ability to move smoothly is very high, as in the following Figures (25):

Figure 25. Testing on an amputee above the knee wears a mechanically smart prosthetic knee joint and alignment adjustment.

6. Conclusions
The new design of the artificial smart knee joint has proven to be 100% successful as a passive mechanical movement. Should a power failure occur at the source, the ability to move smoothly is very high, proved highly efficient by walking, sitting, and applying various daily movements and activities. According to the results of the movement study, the artificial smart knee joint can detect the movement of the residual limb, allowing the prosthesis to simulate the biomechanics of the missing limb without any difficulty. 94.303 MPa and 0.1379 mm are the highest stresses and displacements experienced by the knee joints, respectively at 110°. This remains below the maximum material production resistance of 339.15 MPa (depending on the properties of aluminum alloy material 2024-T3). While the highest safety of factors was at 0° with a value of 5.5905 and the lowest safety of factors was at 110° with a value of 3.5964. The above results show that von Mises stresses and displacement increase with increasing the angle of flexion of the knee joint while the safety of factors decreases as the angle of flexion of the knee joint increases. Limits well below the acceptable material limits were used to make components. The outcome of the study of finite elements shows that the definition is secure enough to use for this particular subject. The results of the F.E study were then compared with the existing data for a related topic from the literature and with the existing development of smart knee joints. The pattern of the effects of motion analysis displayed a great resemblance to the various daily activities of a healthy biological limb.

7. References
[1] Z W Lui, M J Awad, A Abouhossein, A A Dehghani-Sanj and N Messenger 2015 Virtual Prototyping of a Semi-Active Transfemoral Prosthetic Leg (Proc. Inst. Mech. Eng. Part H J. Eng. Med.) vol 229 no 5 pp 350–361
[2] R Borjian 2008 Design, Modeling and Control of an Active Prosthetic Knee (MSc thesis, Mechanical Engineering, University of Waterloo)
[3] Hualong Xie, Guocun Kang and Fei Li 2013 The Design and Control Simulation of Transfemoral Prosthesis Based on Virtual Prototype (International Journal of Hybrid Information Technology) vol 6 no 6 pp 91–100
[4] K Karthikeya 2017 Simulation and Analysis Prosthetic Leg (International Journal of Mechanical Engineering and Technology (IJMET)) vol 8 pp 102–106
[5] Elena Galbally, Frank Small and Ivan Zanco 2015 Retractable Prosthesis for Transfemoral
Amputees using Series Elastic Actuators and Force Control

[6] saeed moaveni 2015 Finite Element Analysis Theory and Application with ANSYS (Fourth Edition Global Edition, Pearson Education Limited) p 929

[7] D L Logan 2016 Daryl L Logan - A First Course in the Finite Element Method-CL Engineering (sixth edition, Cengage Learning) p 400

[8] H-H Lee 2018 Finite Element Simulations with ANSYS Workbench 18: Theory,(Applications, Case Studies) p 612

[9] Cities, Twin Meeting, ANSYS® User 2019 Topology Optimization (https://www.epsilonfea.com/wp-content/uploads/2019/05/ANSYS-Topology_Optimization_User_Meeting_020619.Pdf.)

[10] S Ramesh, R Handal, M J Jensen, and R Rusovici 2020 Topology Optimization and Finite Element Analysis of a Jet Dragster Engine Mount (Cogent Eng.) vol 7 no 1 p 1723821