



Improved Hip Joint Design Based on Finite Element Method under Normal Contact Pressure

¹Pooja Bisen, Abhishek Bhandari²

^{1,2} Dept. of Mechanical Engg., NIRT, Bhopal, INDIA

Abstract— *The hip joint is unique anatomically and physiologically, so a lot of problems may appear through its structure lead to damage. In order to analyze the Hip joint it is very important to analyses the different Stress, Strain and deformation on the joint. In this research work analyzing the deformation on the hip joint at different frequency during the normal load condition and calculate the different Stress on the joint. This work also analyzes the three-dimensional ball and socket hip Joint under normal contact pressure and different frequencies and results have been observed and optimize it. It is suggest that occupational therapists and patients with hip resurfacing arthroplasty should be aware of high metal-on-metal friction situations, which could lead to early failure indicated by result of this research. The deleterious effect of resting periods indicated by this research could be alleviated by appropriate re-initiation of synovial lubrication by movement prior to full loading.*

Keywords— *Hip joints, Contact normal Pressure, Hip prosthesis, FEM*

I. INTRODUCTION

In 2000, the market for medical devices including implantable prostheses worldwide was over 300 billion dollars, which take approximately 8% of the global health care Outgoing, which serves to reach more than 20 million patients [21].

Hip Replacement (THR) is a procedure that is performed surgically, and these procedures can be performed as a total replacement or a Hemi Replacement (replacing half of the hip joint) (THR). The hip joint compared to other joints in the body is the most stable joint, but bodyweight may also lead to developing arthritis because of the extra pressure [13]. All medical components such as orthopedic implants and fractures which has no exception against failure, also the most design problems to prevent failures in these cases are complicated because it expected to operate for a long time, so the responsibility lies on the designer, because of relative values of pressures and high strains, that may occur from the components, also from the complex responses of the human body.

The hip joint is one of the human important components that support the body which connects the femur with the pelvis. The hip joint frequently is subjected to high daily pressure from upper body weight. It is known that the hip joint can withstands up to 4 times of human body weight [1]. With increasing age, these pressures can reduce and endanger its function. Osteoarthritis is one of the most common deteriorations in hip function where a condition may cause severe pain due to joint stiffness. For treating this pain, the hip prosthesis has been proposed for an artificial component designed to perform the same function as a natural pelvic joint and which could be implanted surgically. This surgical operation is referred to as Total Hip Arthroplasty. Materials that have been widely used for

hip joint prosthesis can be divided into several coupling materials, namely metal to metal, ceramic to ceramic, polymer to ceramic, and metal to polymer on artificial femoral heads and acetabular inserts. Here the main limitation of the life of artificial hip joints is influenced by tribological aspects.

The ball and socket joint (or spheroidal joint) is a type of synovial joint in which the ball-shaped surface of one rounded bone fits into the cup-like depression of another bone. The distal bone is capable of motion around an indefinite number of axes, which have one common center.

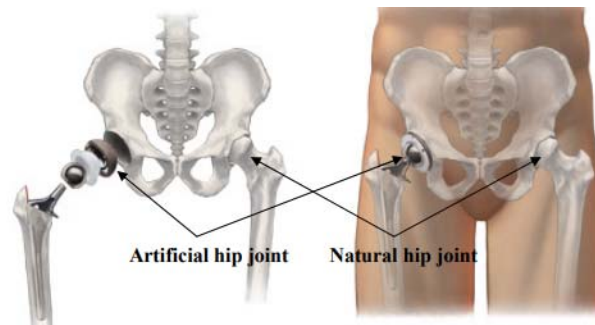


Figure 1.1 Artificial and Natural hip joint

The numerical study using finite element analysis can be used to analyze the hip joint prosthesis stress. ASTM F2996-13 is one of the references to do the analysis. The ASTM F2996-13 is referred to ISO 7206-4 for the hip joint geometry limitation [16]. This analysis can be solved using ANSYS Static Structural software which is already done by the previous studies for the different cross-sectional areas of the hip prosthesis [16]. In this study, the designs of the hip joint prosthesis from different manufacturers were compared for their mass, stress, and safety factor. There are three different designed products from different manufacturers used for static structural approach in ANSYS, which are UNIP, A, and V hip joint prosthesis. The ASTM F2996-13 is selected for the reference to this study with Stainless steel AISI 316L and Titanium alloy Ti6Al4V materials variation.

The rest of paper is organized as follows: In section II we review the work related Hip joint design. In section III we describe the finding of the study. IV conclude this paper.

II. LITERATURE SURVEY

In recent years, due to unhealthy habit, life styles and other reasons there has been a need for devices/medical support equipments that assist the elderly with and encourage independent walking. The hip joint is a ball and socket synovial joint, formed by an articulation between the pelvic acetabulum and the head of the femur. It forms a connection from the lower limb to the pelvic girdle, and thus is designed for stability and weight-bearing – rather than a large range of movement. Many researchers have worked on different design of hip joint. here reviewed the different design of hip joint.

In this work [14] software modelling design and analysis of artificial hip joint to highlight and study the characteristics of the biomaterials that commonly used to design a hip joint based on the stress, strain, and displacement distribution. In this work, the behaviour of (Ti-6-Al4V, Al₂O₃, and Cr-Co-Mo alloys) have been studied based on the stress, strain, and displacement and the results appeared that the use of Al₂O₃ as a material of femoral head that presents higher stress values compared to use Cr-Co-Mo alloy. However, in the case of strain and displacement Al₂O₃ appears results to better than Cr-Co-Mo alloy. Overall, the final results that conducted through this study and analysis of the artificial hip joint model were acceptable compared to the known followed standards in the hip joint design process.



This work [15] proposed designs of a 2-DOFs and 3-DOFs hip joint designs for an exoskeleton robot. Together with bench marking of 1-DOF hip joint designs for exoskeleton robots that are freely downloadable online. These designs were tested based on the human lower-limb motion of hip abduction and adduction, hip flexion and extension, and circumduction. The alpha and beta designs downloaded online were studied by subjecting them to stress conditions. Both existing designs models with the 1-DOF hip joint showed flaws in terms of full functionality and comfort, as they failed to provide vital degrees of freedom vital in simple gait [15]. The designs were not adaptive to human walking patterns and lacked the comfort that all volunteer users hoped for. Thus, this qualifies these designs as non-eligible.

The main objective of this work [16] is to compare stress analysis results of the previously produced UNDIP hip joint prosthesis with those of the other hip joint prosthesis from the different manufacturers using the computational finite element method (ANSYS Static Structural software). The computational analysis refers to ISO 7206-4 and ASTM F2996-13 to calculate the stress and safety factor from the products. The materials used in the simulation were Stainless Steel AISI 316L and Titanium Alloy Ti6Al4V. The results of this work is showing that the lightest was obtained for the UNDIP prosthesis, followed by A and V prosthesis respectively. For 316L material, the only safe hip joint prosthesis could be A product, which was the only prosthesis with safety factor more than 1. Conversely, for Ti6Al4 material, the UNDIP product might be the best hip prosthesis because of its lightweight with the acceptable safety factor.

In this study [17], author proposed a method to determine the assist timing for wire type assist suit. In this method, since the assist timing is determined based on the hip joint angular acceleration by the IMU sensor, the assist can be performed at the optimal timing for each user. As a result of the experiment that in some trials, the maximum hip extension torque were reduced compared to normal walking can be observed. That is the effectiveness of this method can be expected. [17]

The main purpose of this study [18] was to verify a hypothesis that only the magnitude of sensory noise and stiffness can reproducibly determine trends in the hip or ankle movement strategies. Simulations of postural control of a musculoskeletal model for three noise conditions and three stiffness conditions were performed. Variations in the angles of the hip and ankle suggested that the sensory noise amplitude had no influence on the selection. However, the ankle strategy tended to be selected with the increase of stiffness. Strategy shifts of elderly may be derived from other components; muscle weakness, increase of neurological time delay, or learning based on other evaluation index.

Author [19] found the temporal mean of virtual interference to be significantly different in almost every region between the hip conditions. The significance was most distinct when using the static spherical method of COR. These results are promising to individual clinical assessments of hip pathologies using static radiographs and ultimately work towards preventing premature hip disease.

This work [20] reports the estimation of hip joint visco elasticity during voluntary force control using a novel device that applies leg displacement without constraining the hip joint. The influence of hip angle, applied limb force and perturbation direction on the stiffness and viscosity values was studied in ten subjects. No difference was detected in the hip joint stiffness between the dominant and non-dominant legs, but a small dependency was observed on the perturbation direction. Both hip stiffness and viscosity increased monotonically with the applied force magnitude, with posture being observed to have a slight influence. These results are in line with previous measurements carried out on upper limbs, and can be used as a baseline for lower limb movement simulation and further aeromechanical investigations.

The objectives of this work are as follow as: Develop the Finite element model of the hip joint. Analyzing the deformation on the hip joint at different frequency during the normal load condition,

calculate the different Stress on the joint. Study and Analysis of Three-Dimensional ball and socket hip Joint under normal contact pressure and different frequencies and results have been observed and optimize it. Optimize the different stress conditions.

III. PROPOSED WORK

The ability to obtain a 3-D model of a hip joint was generated in CATIA V5R16 software as IGES and STP format. IGES is mostly used format unfortunately was not successful for this paper, as a result STP format is used in exchange. Therefore, the models generated from scratch in CATIA were exported to ANSYS workbench for further “Finite element analysis”. The modeling process occurred in four distinct phases. These phases enhance the ANSYS software enabled a smooth transition between each step analysis.

Development of Solid model of Hip Joint

Development of baseline 3-D hip joint implant model in CATIA. This section also enlists the parameters used to design component. A final step to these procedures includes preparation of the developed models to import them into ANSYS 16.0. Finite element modeling of Hip Joint, The Figure 3.1 represent Hip joint model developed using CATIA modeling package. The stem, cap and femur are separately built and assembled using CATIA assembly options. Standardized data based on experimental results as suggested is considered to developed Hip joint model.

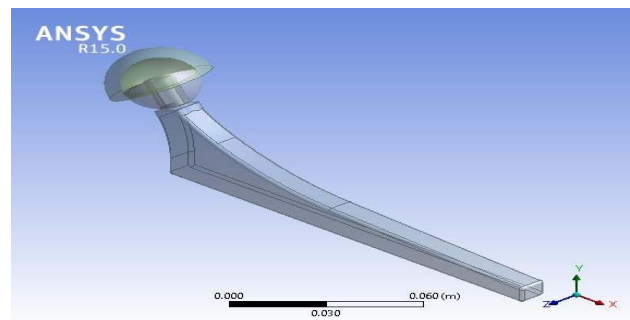


Figure 3.1: Solid Model

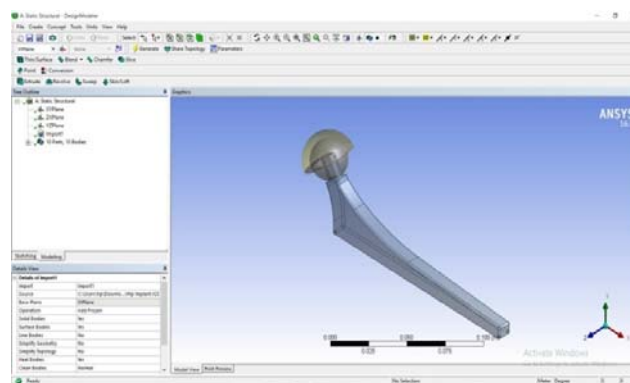


Figure 3.2: Showing the Import Geometry in the ANSYS

Material Properties for the Hip Joint

Two different materials are used in the present finite element simulation: Cobalt Chromium (Co Cr Mo) alloys fall under two main categories: cast alloys (ISO 5832- 4) and wrought alloys (ISO 5832-12). Cast Co-Cr-Mo exhibits elevated mechanical properties and optimal corrosion resistance under

friction condition. Its main drawbacks are related to their poor fatigue resistance and their high cost. Wrought Co-Cr-Mo is even more expensive than cast material, but the higher cost can be justified by the enhanced corrosion and fatigue resistance. In the present study wrought alloy is used to simulate a prosthetic metallic head material due to their high strength and sufficient biocompatibility in clinical conditions.

Table 3.1: Material Property of Co-Cr-Mo and Ti6Al4V

Material Properties	Cobalt-Chromium - Molybdenum Alloy (Co-Cr-Mo)	Titanium Alloy (Ti6Al4V)
Young's Modulus (Gpa)	230	114
Tensile Strength (Mpa)	530	850
Ultimate Tensile Strength (Mpa)	890	960
Density (Kg/m ³)	8300	4420
Expansion (m/m.co)	13.6x10 ⁻⁶	9x10 ⁻⁶
Poisson ratio	0.3	0.35

Meshing

The model is exported to hyper mesh for meshing in step- file format and meshed using solid meshing options. The meshed view of the modeled is shown in Figure 3.3. The structure is Tetra meshed due to complicated geometry with internal Cancellous and cortical bones type geometries. Solid 45 is a 4 noded element with three degree of freedom at each node. Contact Elements TARGE169 is used to represent various 2-D "target" surfaces for the associated contact elements. Conta171, Conta172 and conta175 are used to represent various 3-D solid elements. Contact of elements takes place when surface element penetrates the target segment element. Contact condition of completely bonded type is selected for contact.

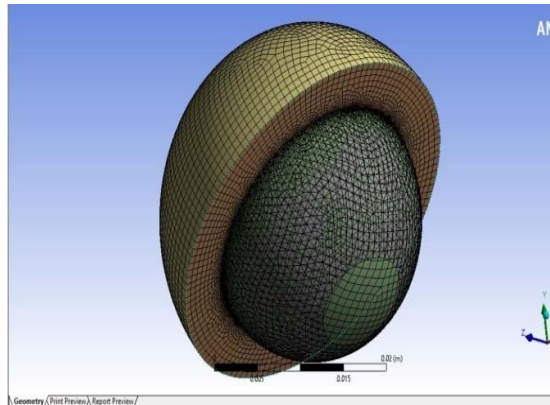


Figure 3.3: Showing the Mesh

Finite Element Analysis

Finite Element Analysis (FEA) or Finite Element Method (FEM) is a relatively new method for solving complex engineering and mathematical problems. Since the 1940s, this method has evolved into the method of choice for computational analysis. In the early years, the finite element method was limited to the manpower available to solve large matrices. However, as technology has evolved FEM has evolved into a computational juggernaut. This process is now only limited by the capabilities of the available hardware to solve matrices that can go out to machine epsilon. FEM is a part of many engineering applications such as structural mechanics, heat transfer, fluid flow, electromagnetic, blade design in orthopedic design for implants and prosthetics. It has become a key part of the design and refinement processes in engineering.

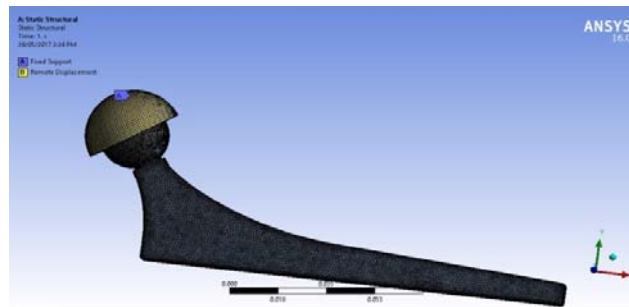


Figure 3.4 Static Force Applications on Head

The load applied at the nodes was automatically transferred from nodes and elements. Nodes at the neck area were coupled using no separation contact with tapered press fit to move at one direction and achieve the similar pattern of motion with a neck inserted into a femur head. The nodes of the outer surface of the neck were fixed with structural fixed support constraint; which resisted the head motion within certain range without allowing femoral component to swing in the space.

Static analysis used in this study was time rate independent; which simply considered time duration as a load step counter and recognized the load steps and load sub steps. The load step is a set of loads applied in the given time duration and load sub steps defines the time steps within a complete load step at which the solutions were calculated for final display of results.

Table: 3.2 Load measurements on Hip Joint

Activity	Load on the hip joint (% of body weight)	Experimental Force (N)
Single-legged stance	300% - 350%	3600
Walking	360% - 400%	3900
Stumbling (peak value)	800%	11,000
Walking upstairs	300%	4200
Walking downstairs	500%	4200
Standing up(supine)	300%	2900
Sitting down	250%	2400

IV. RESULT ANALYSIS

It is well comprehended that contact pressure in the hip joint is closely related to the three dimensional coverage of the socket of the hip bone, and the mechanical stress inside the cartilage increases as contact pressure rises. Finite Element Analysis (FEA) or Finite Element Method (FEM) is a relatively new method for solving complex engineering and mathematical problems. Since the 1940s, this method has evolved into the method of choice for computational analysis. In the early years, the finite element method was limited to the manpower available to solve large matrices. However, as technology has evolved FEM has evolved into a computational juggernaut. This process is now only limited by the capabilities of the available hardware to solve matrices that can go out to machine epsilon. FEM is a part of many engineering applications such as structural mechanics, heat transfer, fluid flow, electromagnetic, blade design in orthopaedic design for implants and prosthetics. It has become a key part of the design and refinement processes in engineering.

Table 4.1: Showing the Deformation at the joint under different frequency at normal pressure

S.N	Frequency (Hz)	Minimum Deformation at the joint (m)	Maximum deformation at the joint (m)	Mean deformation at the joint (m)
-----	----------------	--------------------------------------	--------------------------------------	-----------------------------------

1	18790	0.59772	5.3796	4.185
2	22158	0.5619	5.3265	4.1427
3	18449	0.59135	5.3223	4.1393
4	50648	0.67398	6.0659	3.3702

After analyzing the hip joint at different frequencies under normal pressure condition it is shown that at 50647 Hz the deformation over the hip joint is mean while less as compared to the other frequencies during normal pressure condition. Therefore 50647 Hz was the best frequency during the normal pressure condition in order to increase the life of the joint.

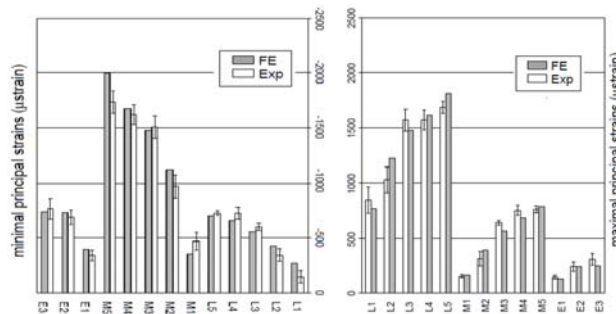


Figure 4.1: Showing the values of strain energy at normal pressure

V. CONCLUSION

In this work analyze the hip joint at different frequencies under normal pressure condition it is shown that at 50648 Hz the deformation over the hip joint is mean while less as compared to the other frequencies during normal pressure condition. Therefore 50648 Hz was the best frequency during the normal pressure condition in order to increase the life of the joint. A reliable methodology to assess hip fracture risk in individuals is crucially important for preventing hip fracture and initiating a repair work. The purpose of this study is to propose a more effective hip failure risk index that is based on the strain energy failure criterion, and it is able to better describe joint failure mechanism. The proposed failure risk index can predict not only the failure risk level, but also the potential failure location. The results of this study showed that there is a very low hip failure risk at optimum frequency and at optimum stress, while, during the, there is a high failure risk at the femoral neck and the intertrochanteric region, compared to the sub trochanteric region. In the future it can also analyze the effect of different pressure on the joint at same frequency and residual stress remain in the joint can also be find out with the help of FEM.

REFERENCES

- [1]. Robert Karpinski, Lukasz Jaworski, JaroslawZubrzycki, "Structural Analysis of Articular Cartilage of The Hip Joint Using Finite Element Method", Advances in Science and Technology Research Journal 2016.
- [2]. Ehsan Askari, Paulo Flores, "A review of squeaking in ceramic total hip prostheses", Tribology International 2016.
- [3]. Mohammad Rabbani and Hossein Saidpour, "Stress Analysis of a Total Hip Replacement Subjected to Realistic Loading Conditions", Verizona Publisher 2015.



- [4]. Tushar V Kavatkar, Milind S Kirkire, Harshal Salvi, DipakPatil, "Wear Analysis of Hip Joint Prosthesis", IJRASET 2016.
- [5]. C. Desai, H. Hirani, A. Chawla, "Life Estimation of Hip Joint Prosthesis", J. Inst. Eng. India Ser. C 2014.
- [6]. Whig, P., Gupta, K., Jiwani, N., Kouser, S., & Anand, M. (2022). Adaptive Clinical Treatments and Reinforcement Learning for Automatic Disease diagnosis. In S. Kautish, & G. Dhiman (Ed.), AI-Enabled Multiple-Criteria Decision-Making Approaches for Healthcare Management (pp. 204-221). IGI Global. <https://doi.org/10.4018/978-1-6684-4405-4.ch011>
- [7]. EkoSaputra, IwanBudiwan Anwar, J. Jamari, Emile van der Heide, "Finite Element Analysis of Artificial Hip Joint Movement during Human Activities", Science Direct 2013.
- [8]. RajiNareliya et al. Biomechanical analysis of Human femur Bone: International Journal of Engineering Science and Technology (IJEST), (2011).
- [9]. Eng. Radu RACA'AN, Contributions Regarding the use of Modern Techniques for Measuring and Modeling Complex Surfaces in the Wear Evaluation of Total Hip Replacements, (2011).
- [10]. JIANG Hai-bo, LIU Hong-tao, HAN Shuyang, LIU Fen, Biomechanics Characteristics of New Type Artificial Hip Joint, Advances in Natural Science Vol. 3, No. 2, 2010, pp. 258-262.
- [11]. K. Gupta, N. Jiwani and N. Afreen, "Blood Pressure Detection Using CNN-LSTM Model," 2022 IEEE 11th International Conference on Communication Systems and Network Technologies (CSNT), 2022, pp. 262-366, doi: 10.1109/CSNT54456.2022.9787648.
- [12]. J. Lord, T. Joyce, Analysis of failed metalon-metal hip prostheses,
- [13]. Dianne-Anand, Wear Analysis of Acetabular Components in a Total Hip Replacement, pp:1.
- [14]. D. Dowson, New joints for the Millennium:Wear control in total replacement hip joints, Journal of Engineering in Medicine 215:335.
- [15]. Satish Prajapati, Dr. S. S. Chouhan, Ranjeet Kumar "Finite Element Analysis of Hip Joint under Normal Contact Pressure" International Journal of Engineering Technology and Applied Science ISSN: 2395 3853, Vol. 3 Issue 5 May 2017
- [16]. N. Jiwani, K. Gupta, M. H. U. Sharif, N. Adhikari and N. Afreen, "A LSTM-CNN Model for Epileptic Seizures Detection using EEG Signal," 2022 2nd International Conference on Emerging Smart Technologies and Applications (eSmarTA), 2022, pp. 1-5, doi: 10.1109/eSmarTA56775.2022.9935403.
- [17]. N. Jiwani, K. Gupta and N. Afreen, "Automated Seizure Detection using Theta Band," 2022 International Conference on Emerging Smart Computing and Informatics (ESCI), 2022, pp. 1-4, doi: 10.1109/ESCI53509.2022.9758331.
- [18]. Mohammed Abdulrahman Abdullah, Loay Salah Al-dein, Nitturi Naresh Kumar," Simulation and Study of Hip Prosthesis Based on Different Common Materials Using Software Modeling" 978-1-7281-9111-9/20/\$31.00 ©2020 IEEE
- [19]. Zimele Gwebu, Kabo Moruti and Rodrigo S. Jamisola Jr. Simulation, "Design and Analysis of Hip Joint DOFs for Lower Limb Robotic Exoskeleton" 978-1-7281-3044-6/19/\$31.00 ©2019 IEEE.
- [20]. Rilo Berdin Taqriban, Rifky Ismail, Jamari J., Athanasius Priharyoto Bayuseno "Computational Analysis of Different Designed Hip Joint Prostheses Using Finite Element Method" Proc. of 2020 7th Int. Conf. on Information Tech., Computer, and Electrical Engineering (ICITACEE) IEEE
- [21]. Junyuan Zhang, Hiroumi Murai, Akihito Ito "Assist Timing Decision Method for Wire Type Walking Assist Suit by Hip Joint Angular Acceleration" 2020 IEEE 9th Global Conference on Consumer Electronics (GCCE) IEEE
- [22]. K. Kaminishi, P. Jiang, R. Chiba, K. Takakusaki, and J. Ota, "Musculoskeletal Simulation for Determining Influences of the Magnitude of Sensory Noise and Stiffness on the Selection of Hip or Ankle Movement Strategies" 978-1-5386-3646-6/18/\$31.00 ©2018 IEEE
- [23]. Adewale Adewuyi, Emily T. Levy, Joel Wells, Avneesh Chhabra, and Nicholas P. Fey "Kinematic simulations of static radiographs provides discriminating features of multiple hip pathologies" 978-1-7281-1990-8/20/\$31.00 ©2020 IEEE



- [24]. Hsien Yung Huang , Arash Arami , Ildar Farkhatdinov , Domenico Formica , and Etienne Burdet “The Influence of Posture, Applied Force and Perturbation Direction on Hip Joint Viscoelasticity” IEEE TRANSACTIONS ON NEURAL SYSTEMS AND REHABILITATION ENGINEERING, VOL. 28, NO. 5, MAY 2020
- [25]. Tram T. Dang and Ali Khademhossein, " polymeric Biomaterials for implantable prosthesis," Boston Massachusetts, p309-331, 2014.