REVIEW ARTICLE

A Review on Finite Element Modelling and Simulation for Upper Limb of Human Bone and Implant

Belal Yahya Al-Tam[1], Muhammad Hanif Ramlee[1,2], Asnida Abdul Wahab[3], Gan Hong Seng[4], Abdul Halim Abdullah[5]

[1] Bioinspired Devices and Tissue Engineering (BIOINSPIRA) Research Group, Faculty of Electrical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia
[2] Bone Biomechanics Laboratory (BBL), Department of Biomedical Engineering and Health Sciences, Faculty of Electrical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia
[3] Medical Devices and Technology Centre (MEDiTEC), Institute of Human Centered Engineering (iHumEn), Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia
[4] School of AI and Advanced Computing, XJTLU Entrepreneur College (Taicang), Xi'an Jiaotong -Liverpool University, Suzhou, Jiangsu, P.R. China 215400
[5] School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor, Malaysia

ABSTRACT

Medical implants are normally used in clinical practice to treat most orthopaedics situations involving bone fractures, deformities, dislocation, and lengthening. It should be noted that specific measures regarding biomechanical and biomaterial characteristics are required for a successful post-surgery procedure. Biomechanical evaluations on the medical implants could be performed by utilising computer and engineering technology. One of them is in silico studies using finite element method that could be simulated in high-performance computer. However, various assumptions are required in computer simulation, such as the constraints on data input and computer resources. This review paper discusses current approaches of constructing a finite element model of human bone with specific material properties for upper limb such as the shoulder joint, humerus, elbow joint, radius and wrist joint. Previous related literatures were reviewed from selected keywords and search engines. To narrow the literature search in this study, inclusion and exclusion criteria of the literature searching were applied. We looked at the current level of knowledge in this field and offered recommendations for future study. In conclusion, studies from previous literature have demonstrated several ways for developing mathematical models and simulating medical implants.
Malaysian Journal of Medicine and Health Sciences (2023) 19(1):325-339. doi:10.47836/mjmhs19.1.41

Keywords: Finite Element, Orthopedics, Implant, Biomechanics, Upper limb, Fixator

Corresponding Author:
Muhammad Hanif Ramlee, PhD
Email: muhammad.hanif.ramlee@biomedical.utm.my
Tel: +607-5558480; +6011-56474150

INTRODUCTION

From a previous study, it is reported that upper limb fractures is accounting for up to 51% of all over fractures (1). Surprisingly, half of the total fracture cases have been reported previously, therefore, deep attention is needed due to the fact that the number of cases is huge. The fractures could be the result of car accidents, ergonomic issues, sport injuries, aging and falling from high altitudes (2). It is known that the fractures are simply happen whenever a force or stress in a specific area of bone is higher than the ultimate strength of that particular bone (3). Hence, an operation is needed by medical surgeons to restore the function of the injured bone either by using external or internal fixations. Based on the authors' best knowledge, there is no specific methods, standards and procedures to choose suitable medical implants in treating bone fractures. Medical surgeons are normally performing surgery based on their expertise, knowledge and experience. An interesting topic to highlight, is the way to simulate the condition of human bones when the medical implants are implanted inside. This way could allow surgeons to justify their choices in treating fracture bone for the sake of giving the best treatment to the patients with promising results and low complications. There should be more comprehension studies of the biomechanical and biomaterial properties (strength, stiffness, toughness) of the fixated implants in order to provide a successful post-surgery process. Otherwise, there will be problems pertaining to the biomechanical characteristics of implants after several months of the surgery (4, 5).

Surgical skill, along with technical expertise and biomechanics experimental testing, has historically been used to create and evaluate orthopaedic implants (6). Over the last decade, computational modelling studies, especially finite element analysis (FEA), have greatly shown as an aided biomechanical assessment (7). In the 1970s, the finite-element method (FEM) was initially used in orthopaedic biomechanics to measure stresses and deformations in human bones during functional loadings, as well as in the design and study of implants (8). A number of biomechanical researches have revealed the FEM to be a beneficial technique (9-21) as well as becoming a powerful tool in assisting to solve difficult and complicated questions (22). With the use of FEM in providing qualitative results (stress, strain, displacement, and relative movement of bone fracture) to medical surgeons, this method would be a good indicator for them to select and choose suitable medical implants and configurations for treating bone fractures. Undoubtedly, this method could provide promising results that can potentially help the medical surgeons to plan surgery ahead prior to entering the operation theatre in the hospital.

In the computational FE method, the model will be divided into several bodies (from the bigger rigid bodies into small bodies) that could explain a problem into smaller set of elements with finite dimensions (8, 22). Afterwards, a process of merging these finite components with be taking place with certain geometric form results in the formation of mesh (6). From here, we could develop an equilibrium system between the external forces applied to the element and the corresponding displacements or stresses at the element's nodes by integrating the element's real geometry with its structural and material parameters. Based on authors' knowledge, there is a limited number of literatures regarding the FE modelling and analysis of the upper limb of human body with medical implants consideration. Therefore, we reviewed previous literatures by considering the state-of-the-art in the related area and come out with some recommendations. The methods for constructing a patient-specific computational model using the FE method, as well as area-specific models of the human body's upper limbs, are subjected and limited in this review paper. In this review paper, the biomechanical studies of implants and its conclusion are explained in detail in the accessibility of orthopaedic applications, together with the existing technologies and challenges in FE model creation.

METHODS

Literature search strategy

Searches were conducted by utilising several keywords from various reputable search engines for this review. To examine existing evidence from the previous literatures, the following search techniques were used:

Keywords used: the search terms included "finite element" (all word combinations that start with the term "computational") and specific terms such as "biomechanics", "mechanics", "implant", "stability", "displacement", "stress", "orthopaedics", "bone", "prosthesis", "internal fixation", "external fixation".

Search engines: Google Scholar, ScienceDirect, Web of Science, PubMed/Medline, IEEE Explore and Research Gate.

Inclusion and exclusion criteria

Our study and review were carried out in accordance with a set of inclusion and exclusion rules. Previously reported studies were only considered in this review paper for inclusion if they met the following rules: (1) Articles published in English only; (2) Research published between July 1979 and July 2021; (3) All types of implants were considered; (4) Biomechanical aspects were included in the studies; (5) The studies' observation endpoint was the commencement of implant design.

On the other hand, literatures were excluded if they satisfy the following: (1) Children's biomechanical reports; (2) Geographic or national comparative studies; (3) Duplicate publications; (4) Animal experiments.

RESULTS AND DISCUSSION

Overview of the finite element

Finite Element Method (FEM) has been a popular method in orthopaedics research during the previous decade (23). The model of a complicated structure and geometry such as human bone and other engineering parts may be divided into a finite number of components using the FEM. The models of the human body could be created by combining two-dimensional (2D) and three-dimensional (3D) data as shown in the previous literature (4, 24). The 2D pictures can be acquired by using a Computed Tomography (CT) scanner or a Magnetic Resonance Imaging (MRI) system which are capable in acquiring over 300 slices per scan (25, 26). Some researchers employed patient-specific subjects (27, 28) in the image collecting procedure, whereas others used synthetic bone (29) to get 2D pictures from the scanner. Commercial software such as Mimics and Amira can be used to create the 3D model (30-32). When medical implants are fixed on the bone, the force loading of the human body could be evaluated using this simulated model with partial and simpler structures (33, 34). All of them are based on the case studies procedure and patient-specific data and design.

Reconstruction of human tissue

The initial step in adopting the FE method to investigate orthopaedics research is to create a 3D model of a human bone. Currently, a 3D model has been developed from a CT or MRI dataset of a subject-specific or synthetic bone utilizing a segmentation process including a large number of 2D pictures (35-37). To be clear, bone and tissue in the human body have different attenuation coefficients, which means they have different grey values. Based on these values, a bone can be created in the segmentation process. Researchers can choose from three programs: Simpleware (38), Scan IP (39) and Mimics (4, 40) all of which offer segmentation techniques and other methods for reducing 2D pictures' noise impacts. Apart from this commercial software, other softwares such as Mechanical Finder, OsiriX as well as 3D Slicer are also available for picture segmentation (41, 42). The segmentation can be accomplished by a variety of approaches, and there is no one procedure is favored than the other.In some literature, a Hounsfield unit was utilised to segment the human bone. For example, HU of 700, for instance, was used to differentiate cortical from cancellous bones (Hu<700 indicates cancellous and Hu>700 indicates cortical) (43). Furthermore, it is worth noting that certain finite element models have been assigned with grey data values in HU for each element (36, 42, 44). Extrusion of soft tissue like cartilage from its bone could be done through Mimics and 3-matic software (45, 46). On the other hand, FE programs were used to simulate the ligament (4, 45).

Modelling and meshing

There are several FE packages available from the market that can help with segmenting the image for 3D model meshing and FE analysis, including Abaqus, Marc Mentat, NASRAN, ANSYS and Catia (47-52). Additionally, mesh creation may be accomplished through the use of image processing tools such as Mimics, U-GRAPH, and 3-Matic software (47, 53). All software applications provide automated mesh generators capable of meshing the human bone correctly. Furthermore, these applications offer manual mesh manipulation, allowing the user to customize the element size and type (54). Nevertheless, a mesh convergence study must be undertaken first to confirm that any additional FE findings are independent of any parameter before manually setting the element size and type (55-57). Both the r-refinement (element type) and the h-refinement (element size) (4) are methods that can be used to conduct the convergence investigation. Implants are made through SolidWorks and Autodesk which are two of the most frequent software when it comes to designing and meshing a model of the body (58, 59). The type of element utilised in the FE model is determined by the type of human tissue being modelled. The majority of researchers employed a triangular tetrahedral element for bone (60, 61) and a hexahedral element for soft tissue like skin and meniscus (32). However, the triangular tetrahedral element was employed for the analyses of a FE model of implants (4). Lastly, it is beneficial to simulate ligaments when doing finite element evaluations of implants particularly in joint locations as it might enhance and imitate the human body's natural movement, allowing for even load distribution.

Assigning materials properties

Young's modulus values are the most frequent approach for assigning material characteristics to bones and implants (46, 49). Finite element program can be used to set values for Young's modulus, which has a relation with both density and elasticity (38, 51). Orthopaedic implants are simple because the material is homogenous and linear isotropic (62-64). Nevertheless, neither of these materials exist in the human tissue and bone. As a result, several researchers reduced the model to be linear isotropic only (55, 59). On the other hand, it was determined that other tissues, like skin and cartilage had hyper-elastic properties (45). Moreover, hyper-elastic and Young's modulus characteristics were discovered through particular studies, whereas others were found through research of prior publications (59). Alternatively, grey scale data from computed tomography may also be used as a way of assigning material characteristics (65).

Boundary and load condition

Relevant boundary and load conditions should be used in finite element analysis in order to simulate the true behavior of the human body. In the physiological condition of healthy individuals, the load is imposed where the reaction force or pressure is applied (66). Joint force, weight-bearing force and muscle force are all examples of what is known as force or pressure. These forces can be calculated and derived from electromyography data, or they can be determined by motion analysis techniques such as inverse dynamics, which more accurately simulate real-world loading circumstances. It should be emphasized that the use of boundary conditions is not the same as the use of finite element models in general; nonetheless, it should imitate a situation that is comparable to that of activities of daily life, pathological situations, or rehabilitation treatments. For instance, in shoulder model, Maurel et al. (67, 68) studied the cemented fixation state in these four places when loaded 500N: anterior scapula, posterior scapula, anterior humerus and posterior humerus, both with physiological loads equal to 0–180o abduction and adduction motions without the use of muscles. Meanwhile, Pomwenger et al. (69) simulated a keeled versus pegged glenoid implant with a force of 650N for a flexion and abduction of 90° and the primary involving muscles of the scapula are m. trapezius, m. rhomboideus, m. deltoideus, m. serratus anterior and inferior at their insertion points on the scapular surface. It is similar happened in other parts such as the radius, where Xu et al. (23) utilised two absorbable screws to fix the fracture and preserve its stability, with a vertical downward force of 100N at the stress point and seven possible angles of the two screws ranging from 0o to 90o. Based on these previous researches, the boundary conditions were chosen based on patient's condition and the values of specific forces that would be used to build the finite element model.

Validation of the finite element model

The validation of the finite element model is a critical step in the computational analysis process, and it should be carried out to determine the accuracy of the predictions (28, 31, 70, 71). Furthermore, it can give evidence and suggestions for the enhancement and adjustment of the model in order to better replicate the real-world characteristics of human tissue. Many researchers employed cadaveric specimens in their study to evaluate their finite element model (70, 72). For example, five fresh-frozen human cadaver radiuses were used by Rogge et al. (73). They were loaded with up to 100N in the axial direction at two points: the scaphoid and the lunate. The scaphoid fossa received 60% of the total stress, whereas the lunate fossa received 40%. It was necessary to completely thaw the specimens, remove any soft tissues, and securely embed the proximal end of the shaft. The projected bone stresses and simulated fracture motion predicted by the model were compared to the test results obtained under 100N loads for the intact and defect situations. For comparison with the FE model predictions, the average variation in strain magnitudes was 7.1%. The average difference in fracture motion, on the other hand, was 3.9%. Other than that, Pistoia et al. (74) simulated the kind of fall that generally results in a Colles'-type fracture in the radius right above the wrist, even though the strain rate in the experiment was considerably lower than in an actual fall. A total of 54 embalmed cadaver arms were used in the study, where the forearms with undamaged soft tissues were compressed. For each arm, epoxy glue was used to embed the proximal portion of the radius, which was then secured to a bottom anvil of an automated testing machine. An experiment simulating compression testing was carried out with a 1000N stress applied to the most proximal surface, whilst the most distal surface was entirely restricted. To validate the computer model, FE models were created, and biomechanical testing was performed. Despite the benefits of utilising human cadaveric specimens, commercially available synthetic models such as Orthobone, Sawbones or Synbone have attracted attention and appear to be an appropriate technique for validating FE models (75, 76).

Area-specific finite element model

Shoulder joint

Treatment of musculoskeletal injuries of the shoulder is still a complex job, and it is generally performed by shoulder arthroplasty (69). Implant designs continue to be a contentious topic (77). As the failure rates of shoulder arthroplasty are significantly lower than those of hip and knee arthroplasty, indicating that this treatment is even more effective. The most essential joint in the shoulder is the glenohumeral joint (78). It is formed by two bones: the humerus and the scapula. In addition, glenohumeral interaction happens when the humeral head, which is more or less a spherical shape, makes contact with the glenoid, a narrow shallow hollow on the scapula. To analyse the activity of the glenohumeral joint, a numerical model must contain the specific bone geometry and the joint's supporting muscles. Hence, the stress distribution across each bone of the joint was then assessed using models based on the deformable body approach (79, 80). Where the most modern ones integrate the two techniques by utilising the inverse dynamic concept as well as the muscular forces to measure the stress using a finite element (FE) model (78, 81, 82). Shoulder arthroplasty is linked with some risks such as Rotator cuff tears, glenohumeral instability, prosthetic loosening, infection and periprosthetic fracture (69, 83, 84). Where the term instability is generally when the humeral head comes into touch with the glenoid's rim during subluxation. Hence, instability sets in, and the resulting force is just barely projecting into the glenoid cavity. Although experimental investigations have been conducted to investigate the influence of implant constraint and conformity on subluxation loads and intra-articular joint translations but the accompanying materials and labor expenses are usually expensive (85, 86). On the other hand, researchers may save money by using finite element techniques to avoid expensive laboratory testing and they can also better comprehend their findings theoretically (87). It is also feasible to derive estimates of joint pressures, fixation stresses and material deformation using finite element models rather than doing laboratory testing, which is hard or impossible. Additionally, the studies may be readily and rapidly repeated for both little and large adjustments in the design parameters, providing a better comprehension of their respective impacts (88). Anglin et al. (86) and Hopkins et al. (88) carried an experimental analysis to validate the finite element method by testing different glenoid models (figure 1) in order to determine their ability to withstand dislocation using CAD data from Zimmer GmbH. Moreover, they reported that the FE models of the testing equipment for four distinct glenoid designs revealed a high agreement with the experimental

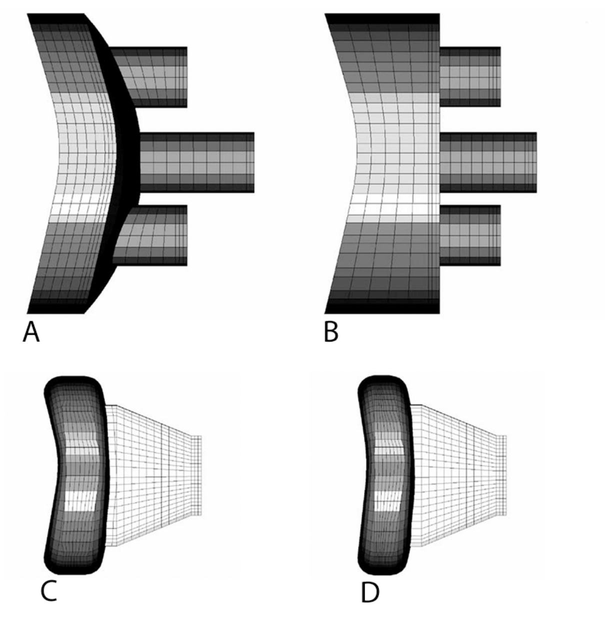

Figure 1: The four modeled glenoid component designs using finite element analysis as follows: pegged curved backed size L (A), pegged flat backed size L (B), keeled conforming size XS (C), and keeled non-conforming size XS (D).

subluxation force ratios. All estimated subluxation forces and translations were within the experimental study's scope. The FEA has indicated that constraint and conformity are two critical implant design features that significantly influence dislocation stress and humeral head movement, respectively.

Humerus bone

1-3% of all fractures are humeral shaft fractures, which account for 20 percent of all humerus injuries (89, 90). The fractures that occur in the lower and upper third of the shaft make more than 80% of all incidences, with 60% of the fractures occurring in adults over 50 years old. Whereas, the central third of the bone was impacted in approximately 60% of the individuals, having fractures of spiral and transverse shape (91, 92). It is more common in older people who have been injured in a fall or torsion trauma than in younger people who have been injured in an accident with a great amount of energy (93). In addition, a lot of fractures can be effectively managed without surgery due to the extensive muscles and soft tissues around the bone which may be utilised to fix the fracture (89). In contrast, there are various forms of fractures necessitate operations and internal implants. Hence, going for plate fixation or intramedullary nails (94) is based on the specific location of the fracture as well as the decision of the surgeons. Moreover, the right mechanical characteristics of the joint components must be included into finite element (FE) models in order to understand their mechanical behavior. This data is extremely critical for the development of accurate and precise fixations (95). Therefore, Masih et al. (96) used CT scan to extract geometrical data of a 17-year-old boy proximal humerus bone in the form of Digital Imaging and Communications in Medicine (DICOM) images. Furthermore, performance of this machine is very accurate, where 1024x1024 are the resolutions and 0.4 mm is the slice thickness. This procedure had undergone five steps as follow:

A) Importing of DICOM into MIMICS 10.01 software: DICOM images are loaded and shown, then the segmentation object is generated by applying a correct threshold value, based on the Hounsfield Unit, which is depicted as a colored mask that contains just those important pixels of the images. MIMICS established a bone (CT) threshold, allowing us to simply pick all bone tissue; the green region represented bone tissue pixels, which we specified as a mask. The humerus was chosen based on the mask, (figure 2a) and it is tweaked until perfection.
B) Using MIMICS 10.01 software to create a FE model and a surface mesh: The bone's 3D model was constructed using the produced region mask. 2D photos are converted into 3D models using 3D grey value interpolation algorithms. It is a true 3D interpolation approach that accounts for the Partial Volume effect, making it more reliable. Surface mesh can be found by a tool called "remeshing" which is utilised to enhance the effectiveness of the triangles so that an FEA package's preprocessor can form tetrahedron meshes out of them (figure 2b).
C) Conversion of surface mesh to volumetric mesh using ABAQUS 6.10: Tetrahedrons are formed from triangles through the process of transferring the mesh to ABAQUS since the results will be precise and real when the mesh is denser.
D) Assigning material properties in the software of MIMICS 10.01: In this stage, Mimics will compute grey values for each element of the volumetric mesh before applying materials to them. Besides, the default material parameters of the CT given dataset are taken into account. Afterwards, Humerus bone was transformed into stereolithography files for FE Analysis using MIMICS STL module (figure 2c).
E) FEA with ANSYS 14: FEA meshes are built in FE modelling process and then they are converted into static structural, in which it is fully assessed under particular boundary and loading conditions (figure 2d). Lastly, the force was placed in tension axially at the distal half of the humerus, while the humeral head remained stationary (figure 2e).

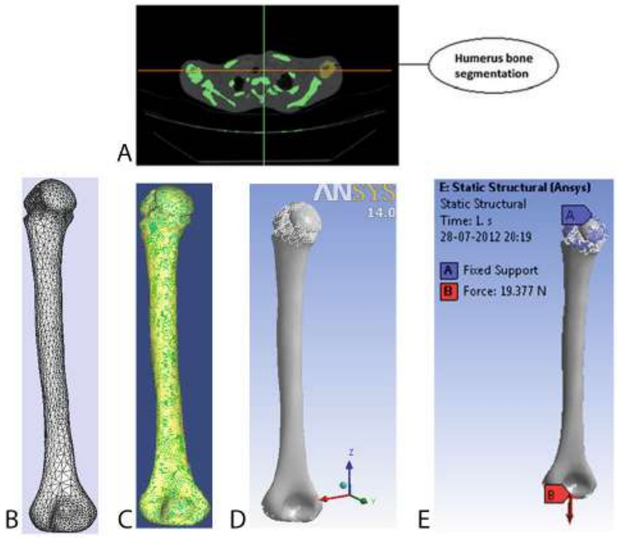

Figure 2: The procedure starts by masking the Humerus bone (A) followed with meshing of the bone surface (B), volumetric mesh with material assignment (C), uploading the 3D Model for analysis (D), and lastly loading and boundary condition are applied using ANSYS (E).

Nevertheless, studies by Antoniac et al. (89) showed several issues linked to the treatment of the humerus fracture which might suppress the treatment such as age, poor living habits as well as medical conditions that affect the muscles and bones (97).

Elbow joint

Elbow is one of the rare areas in the body where two bones interact with one another (96). It is one of the most complicated joints in the human body, and it is examined statically as part of a system (98). Total elbow arthroplasty (TEA) has become a more popular and recognised technique (99). Particularly, in patients with rheumatoid arthritis, osteoarthritis, or posttraumatic arthritis (100, 101). The elbow's primary role is to keep the hand in place for bimanual tasks where flexion, extension, pronation, and supination are the main movements (102). In general, the flexion-extension can extend for around 0 to 140 degrees (103-105). However, only 30 to 130 degrees are required for our daily movements. In contrast, the usual range is roughly 90 degrees of supination to 80 degrees of pronation, yet so many tasks of daily life only need 50 degrees in each direction (103, 105).

Previous research by Tarnita et al. (106) has studied a spherical prosthesis-elbow for 4 different positions of the implant such as Flexion-Extension and Supination-Pronation using FEA as follows:

Flexion-extension

The testing of the implant was modelled by Solidworks and analysed by Visual Nastran software and these are the parameters:

-The flexion-extension cycle lasts 1 second in total.

-The applied force is 100N, which refers to lifting and lowering a load by pure flexion-extension movement. In addition, it has a persistent vertical action, and its site of application is at the end of the ulnar stem.

-Other components are flexible or connected to stable or moving pieces, with the exception of the humeral stem, which is fixed.

-Between the two components constituting the spherical joint, a constant angular speed of 160o per second was created.

The maximum stress values of the spherical prosthesis-elbow joint were presented in the diagram (figure 3a) that shows comparison of maximal flexion-extension stresses in a healthy elbow vs a prosthetic elbow. The highest values of the elbow joint in extension flexion motions are recorded for the elbow in the 90o posture, when the bending moment is the highest due to the maximum force of the arm. Whereas, the force of the arm changes to 0, the minimum forces are recorded in the maximum extension condition. Moreover, when the force arm turns very low, but not zero, tiny values will be recorded in the maximum flexion position. In brief, the recorded stress values of the maximum flexion-extension angle in the proposed prosthesis-elbow are nearly 5 times better and higher than a healthy elbow.

Pronation-Supination

As shown in (figure 3b) the maximum stress values for pronation-supination in the situation of healthy and spherical prosthesis elbows are compared in this study. Parts of the bones or prosthesis are only subjected to a vertical load of 100N in the vertical anatomical posture of the arm and the values of the stress in the elbow joint stay nearly constant throughout the analysis. It was observed that spherical prosthesis-elbow has nearly 5 times significant difference in stress levels compared to a healthy elbow. The findings were achieved after some simplifications were made for instance, only the implants and bones components were considered, while the action of the muscles, tendons, and ligaments was ignored so that a finite element analysis on such a very complicated model can be conducted easily. Consequently, Biometrics which are an advanced acquisition system (107, 108) as well as a specialized experimental bench (109) will be used to enhance the study of the kinematics of prosthetic and healthy elbow joints in the coming studies.

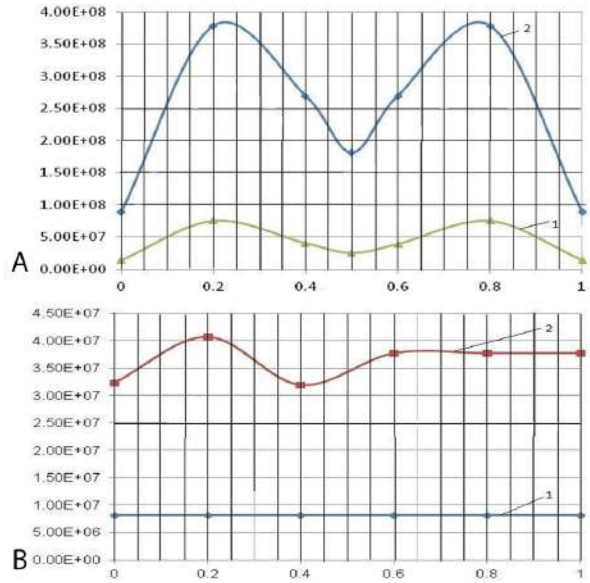

Figure 3: Graphs of Comparing between the maximum flexion-extension (A) and pronation-supination (B) (MPa) for healthy and spherical prosthesis-elbow.

Other researches by Radakisnin et al. (110) have performed a simple FEA using ANSYS WORKBENCH 15.0 software to measure von-Mises Stress and the maximum principal stress on three different elbow materials which are copper, stainless steel and titanium. The yielding of the material will happen as soon as distortion surpasses the yield point as suggested by the von-Mises stress. Whereas, a material will fail when its maximum principal stress approaches its elastic limit in simple tension and exceeds its maximum stress according to the maximum principal stress. The results of the analysis of von-Mises stress are

Titanium: 1.92MPa to 42.37MPa, Stainless Steel: 2.98MPa to 45.05MPa and Copper: 3.60MPa to 45.76MPa. On the other hand, the maximum principal stress are; Titanium: 2.13MPa to 39.30MPa, Stainless Steel: 3.28MPa to 40.42MPa and Copper: 4.00MPa to 41.62MPa. As a result, titanium is the lowest in both stresses then followed by stainless steel and copper. Hence, titanium is the safest and best of all analysed materials for the development of the elbow implant since it can tolerate stresses more than the others.

Radius

The purpose of the radial head is to transmit stress and stabilize the outside section of the elbow joint, which is critical to preserving the elbow joint's stability and functionality (23). Additionally, its treatment has advanced from nonsurgical to internal fixation, artificial prosthesis replacement, and radial head removal. Distal radius fractures are the most common injuries encountered in orthopaedics, which was about 20% before 2000 (111-116) and decreased to 17.5% (117-120) in recent years. It is mostly treated by determining the stability of the fracture form (121, 122) and pattern (114, 123-125). A stable fracture can be handled successfully with cast immobilization; however, an unstable bone fracture may require a fixation surgery (126, 127). External fixation (128, 129), plate fixation (130) and percutaneous pinning (131) are among the surgical procedures used to treat unstable distal radius fractures. Aside from that, adjuncts like bone grafting and bone substitutes are utilized to improve stability and healing (116). A Study by Rogge et al. (73) studied fracture pinning in a simulated distal radius fracture using FE modelling (FEM), with a focus on assessing simulated fracture stability and bone stresses. The main focus of this study is to figure out the best stability technique on the region of greatest changes in stress between intact and pinned fractures which are situated 3mm above to 3mm below the fracture. The proximal end of the model was restricted, and a 100N force was applied, where a scaphoid fossa and lunate fossa each received 60% and 40% of the stress, respectively (132, 133). Therefore, this study was divided into two sets of extra-articular, unstable metaphyseal fractures:

- Set A: A cancellous bone in the volar one-third of the radius was broke but remained in apposition.
- Set B: The cancellous bone contact at the broke area was entirely missing.

Moreover, three pinning setups engaging the radius styloid and proximal dorsal cortex were used to stabilize the simulated fractures: A single pin, two parallel pins and two crossed pins, one acting as same as the previous ones while the second engaging the distal ulnar corner and proximal volar cortex. The findings show that fixation with a single steel pin resulted in 62% fracture collapse, from a 3mm gap to a 1.14mm gap, for the simulated fractures in set A. The parallel and crossing pin arrangements collapsed at a rate of 35% and 14%, sequentially. Whereas, Set B fractures showed similar patterns: single pin 68%, parallel pins 39%, and crossing pins 18%. In the presence of pin fixation, cancellous bone contact in the volar area of the radius offered somewhat more stability. Fixation with two pins resulted in a more typical stress distribution in the bone at the fracture site, as well as enhanced fracture stability. Two crossing pins implanted through the radius styloid offered significantly more stability than two parallel pins inserted through the radius styloid. Despite crossing pinning was the most stable design and decreased stress in the volar cortex, the cancellous bone was still subjected to greater loads. In short, a crossing arrangement with metal pins proved to be the most effective pinning approach for resisting low to moderate axial stresses. To date, various types of implants have been utilised however, absorbable fixations are very commonly used in orthopaedic operations (134-136). Therefore, Xu et al. (23) have carried out an experiment using FEM on two absorbable screws for the fixation of the radial fracture with a vertical downward 100N load at the stress site as well as, 7 different angles of 0o, 15o, 30o, 45o, 60o, 75o, and 90o. The results of the study show that when the angle was increased from 0o to 45o, the stress between the screws progressively reduced, however when the angle was extended from 45o to 90o, the stress surged. The von-Mises stress and peak displacements attained minimal values of 13.11MPa and 0.15mm when the angle was 45o. On the other hand, they rose dramatically when the screw angle was 90o, up to 24.63MPa and 0.25mm for von-Mises and displacement, respectively; these circumstances potentially weaken the fracture block and enhance stress concentration. The explanation for this behavior is because when the angle between the screws rises, the screws will pass through the radius, lowering the effective fixed length, raising tension, and increasing displacement. Besides, the highest screw displacement was 0.25mm, which had a negligible effect on the patient, demonstrating that the use of absorbable screws to treat radial head fractures was safe. The tension was limited and, as a result, the displacement was small when the angle was 45o, showing that this angle gave the optimum stability and safety. Furthermore, screws positioned at 45o on the bone surface revealed a reduced stress distribution than screws positioned at other angles, which is consistent with Shi et al. (137) findings. In conclusion, in most cases, pin fixation is insufficient in the clinical setting. Higher axial stresses, as well as torque and bending, need the use of a cast or external fixator (73).

Wrist and hand

Rheumatoid arthritis (RA) is among the most frequent bone illnesses related to the wrist joint (138-140). It tremendously affected the human wellbeing all around the globe (141). The basic goals of wrist replacement surgery are to alleviate discomfort and maintain a good function of the wrist (142). Additionally, it can help to keep or regain wrist motion and make it easier to carry out daily tasks. There are three signs of RA which can be observed in the wrist: ligamentous laxity, cartilage degeneration and synovial proliferation (143). Total wrist arthroplasty (TWA) is a prominent surgical option for addressing severe deformities of the joint caused by RA (144). Implant loosening and metacarpal perforation have both been documented as adverse effects of TWA usage (145). Hence, Intercarpal fusion was used to prevent metacarpal perforation, and resurfacing of the distal radius area was used to prevent implant loosening and displacement (146). For both treated and rheumatic wrists, hand grip strength was a prevalent sign of evaluation (145, 147, 148). Using FEM, Gislason et al. (149) computed the loading static gripping force of the wrist. The resultant compression pressure was found as 7.33 MPa and distributed throughout the five metacarpals (figure 4a) using the static gripping force of the wrist. Moreover, to facilitate the convergence of the solution, constraints were placed at the proximal ends of the radius and ulna (figure 4b-d) (150). The insertion of tendons and the carpometacarpal joint were fixed to prohibit mobility in the x and y directions, allowing all bones except the radius and ulna to move in the direction of applied force (151). These constraints were important because they kept the tendons in place at their insertion points which helped in attaining the convergence of the model (152). Bajuri et al. (139) conducted a similar experiment in which they used FEM to simulate three models: a healthy wrist, TWA, and RA models. The contact pressure in the computed RA model is about 10 times higher (3.9 MPa) than in the healthy model (0.40 MPa). The large magnitude of the RA model was decreased to roughly five times its original value after TWA (0.75 MPa), which is practically identical to the healthy model's magnitude (47% bigger). Furthermore, among the three simulated scenarios, the RA model with 11 MPa revealed the trapezium (without the consideration of resected scaphoid) to be the most pressure-sensitive bone.

A

	Thumb	Index	Long	Ring	Little
Cross-sectional Area (mm^2)	126.30	77.00	84.15	57.60	80.20
Load (N)	255.60	120.30	106.40	88.00	77.30
Pressure (MPa)	**2.02**	**1.56**	**1.26**	**1.53**	**0.96**

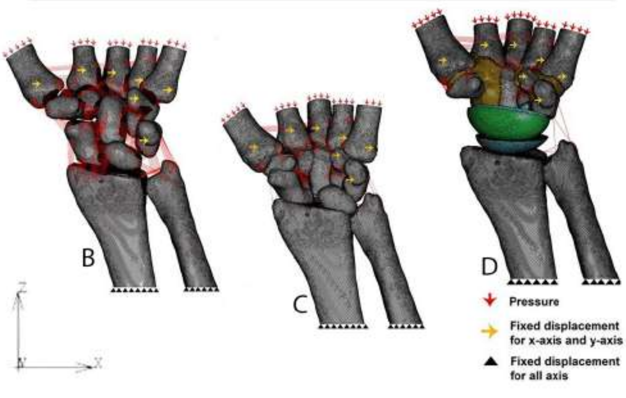

Figure 4: Pressures applied on the metacarpal bones (A), as well as the building of a finite element model of the healthy (B), RA (C), and TWA (D) models.

DIRECTIONS FOR FURTHER STUDY

Despite the fact that present review contains useful data and knowledge, further research is required to offer specific prescriptive recommendations on the use of orthopaedic implants. It is preferable to construct a model capable of simulating real-world conditions in order to conduct a detailed analysis of bone fractures. Additionally, getting the information necessary to design the characteristics of bones and other supporting structures continues to be a challenge. Various characteristics of bone, such as Young's modulus and Poisson's coefficient, can vary significantly between various parts of the bone, which can have significant impact on the results (153, 154, 155, 156, 157). Moreover, while developing material models for patients, their bone density should be considered. For more realistic and accurate result, both elasticity and plasticity should be incorporated in the analysis. As a result, additional research should be conducted to offer more knowledge about the characteristics of simulated materials.

LIMITATIONS OF THIS REVIEW

Despite the fact that the present review contains useful data and knowledge, further research is required to offer specific prescriptive recommendations on the use of orthopaedic implants. This compact paper might serve as one of the helpful resources not only for engineers, but also for medical professionals and scholars in this field. In addition, there are also certain restrictions which have been explored in this assessment. The other constraint is that we only looked at finite element analysis of implants for the human body's upper limbs. Moreover, the review concentrated on existing methodologies for developing a computational model, which were discussed in detail. This review does not cover any more aspects of the FE approach than those mentioned previously.

CONCLUSION

This review paper concluded that the process of developing a finite element model from previously published studies was dependent on the specific pathological disorders, homogeneous or inhomogeneous as properties of the material, type of data either Magnetic Resonance Imaging or Computed Tomography, as well as the boundary conditions. Other than that, the validation of FE models is based on researcher's preferences, since there is no specific method and standards that should be followed.

ACKNOWLEDGEMENTS

The work has been carried out with the support of research funding from the Ministry of Higher Education Malaysia, under Fundamental Research Grant Scheme (FRGS) (Grant no.: 5F135 and FRGS/1/2019/TK05/UTM/02/3), Universiti Teknologi Malaysia (UTM) under UTMSPACE (Grant no.: 4J547), Prototype Development Fund from Innovation and Commercialisation Centre (ICC) (Grant no.: 4J583), Matching Grant (Grant no.: 04M13) and Universiti Kuala Lumpur under Collaboration Research Grant Scheme (Grant no.: 4B618).

REFERENCES

1. Curtis EM, van der Velde R, Moon RJ, van den Bergh JP, Geusens P, de Vries F, et al. Epidemiology of fractures in the United Kingdom 1988-2012: Variation with age, sex, geography, ethnicity and socioeconomic status. Bone. 2016;87:19-26. doi: 10.1016/j.bone.2016.03.006
2. Milenkovic S, Mitkovic M, Bumbasirevic M. External fixation of open subtalar dislocation. Injury. 2006;37(9):909-13. doi: 10.1016/j.injury.2006.02.051.
3. Sherekar R, Ganjare A, Pawar A. Finite Element Analysis of Human Clavicle Bone: A Methodology Review. American Journal of Mechanical Engineering and Automation. 2014;1(5):54-9. Retrieved from http://www.openscienceonline.com/journal/ajmea
4. Ramlee MH, Abdul Kadir MR, Murali MR, Kamarul T. Biomechanical evaluation of two commonly used external fixators in the treatment of open subtalar dislocation—A finite element analysis. Medical Engineering & Physics. 2014;36(10):1358-66. doi: 10.1016/j.medengphy.2014.07.001.
5. Kapukaya A, Subasi M, Arslan H, Tuzuner T. Non-reducible, open tibial plafond fractures treated with a circular external fixator (is the current classification sufficient for identifying fractures in this area?). Injury. 2005;36(12):1480-7. doi: 10.1016/j.injury.2005.05.005
6. Inzana JA, Varga P, Windolf M. Implicit modeling of screw threads for efficient finite element analysis of complex bone-implant systems. Journal of Biomechanics. 2016;49(9):1836-44. doi: 10.1016/j.jbiomech.2016.04.021
7. Driscoll M. The Impact of the Finite Element Method on Medical Device Design. Journal of Medical and Biological Engineering. 2019;39(2):171-2. doi: 10.1007/s40846-018-0428-4
8. Choi AH, Conway RC, Ben-Nissan B. Finite-element modeling and analysis in nanomedicine and dentistry. Nanomedicine. 2014;9(11):1681-95. doi: 10.2217/nnm.14.75
9. Dawson JM, Khmelniker BV, McAndrew MP. Analysis of the structural behavior of the pelvis during lateral impact using the finite element method. Accident Analysis & Prevention. 1999;31(1):109-19. doi: 10.1016/S0001-4575(98)00052-9
10. Kaku N, Tsumura H, Taira H, Sawatari T, Torisu T. Biomechanical study of load transfer of the pubic ramus due to pelvic inclination after hip joint surgery using a three-dimensional finite element model. Journal of Orthopaedic Science. 2004;9(3):264-9. doi: 10.1007/s00776-004-0772-9.
11. Dalstra M, Huiskes R, van Erning L. Development and Validation of a Three-Dimensional Finite Element Model of the Pelvic Bone. Journal of Biomechanical Engineering. 1995;117(3):272-8. doi: 10.1115/1.2794181.
12. Spears IR, Pfleiderer M, Schneider E, Hille E, Morlock MM. The effect of interfacial parameters on cup–bone relative micromotions: A finite element investigation. Journal of Biomechanics. 2001;34(1):113-20. doi: 10.1016/s0021-9290(00)00112-3.
13. Thompson MS, Northmore-Ball MD, Tanner KE. Effects of acetabular resurfacing component material and fixation on the strain distribution in the pelvis. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine. 2002;216(4):237-45. doi: 10.1243/09544110260138727.
14. Schultze C, Klьss D, Martin H, Hingst V, Mittelmeier W, Schmitz K-P, et al. [Finite element analysis of a cemented ceramic femoral component for the assembly situation in total knee arthroplasty]. Biomed Tech (Berl). 2007;52(4):301-7. doi: 10.1515/BMT.2007.051.
15. Bachtar F, Chen X, Hisada T. Finite element contact analysis of the hip joint. Medical and Biological Engineering and Computing. 2006;44(8):643-51. doi: 10.1007/s11517-006-0074-9
16. Kluess D, Martin H, Mittelmeier W, Schmitz K-P, Bader R. Influence of femoral head size on impingement, dislocation and stress distribution in total hip replacement. Medical Engineering & Physics. 2007;29(4):465-71. doi: 10.1016/j.medengphy.2006.07.001
17. Dalstra M, Huiskes R. Load transfer across the pelvic bone. Journal of Biomechanics. 1995;28(6):715-24. doi: 10.1016/0021-9290(94)00125-n.
18. Manley MT, Ong KL, Kurtz SM. The Potential for Bone Loss in Acetabular Structures Following THA. Clinical Orthopaedics and Related Research®. 2006;453. doi: 10.1097/01.blo.0000238855.54239.fd.
19. Oki H, Ando M, Omori H, Okumura Y, Negoro K, Uchida K, et al. Relation Between Vertical Orientation and Stability of Acetabular Component in the Dysplastic Hip Simulated by Nonlinear Three-dimensional Finite Element Method. Artificial Organs. 2004;28(11):1050-4. doi: 10.1111/j.1525-1594.2004.00017.x.
20. Anderson AE, Peters CL, Tuttle BD, Weiss JA. Subject-Specific Finite Element Model of the Pelvis: Development, Validation and Sensitivity Studies. Journal of Biomechanical Engineering. 2005;127(3):364-73. doi: 10.1115/1.1894148.
21. Garcı´a JM, Doblare´ M, Seral B, Seral F, Palanca D, Gracia L. Three-Dimensional Finite Element Analysis of Several Internal and External Pelvis Fixations. Journal of Biomechanical Engineering. 2000;122(5):516-22. doi: 10.1115/1.1289995.
22. Kluess D, Wieding J, Scouffrant R, Mittelmeier W, Bader R. Finite Element Analysis in Orthopaedic Biomechanics. InTech. 2010:151-70. doi: 10.5772/

intechopen.83980
23. Xu G-m, Liang Z-y, Li W, Yang Z-z, Chen Z-b, Zhang J. Finite Element Analysis of Insertion Angle of Absorbable Screws for the Fixation of Radial Head Fractures. Orthopaedic Surgery. 2020;12(6):1710-7. doi: 10.1111/os.12797.
24. Mehboob H, Chang S-H. Evaluation of healing performance of biodegradable composite bone plates for a simulated fractured tibia model by finite element analysis. Composite Structures. 2014;111:193-204. doi: 10.1016/j.compstruct.2013.12.013
25. Halloran JP, Petrella AJ, Rullkoetter PJ. Explicit finite element modeling of total knee replacement mechanics. Journal of Biomechanics. 2005;38(2):323-31. doi: 10.1016/j.jbiomech.2004.02.046.
26. Cody DD, Hou FJ, Divine GW, Fyhrie DP. Short Term In Vivo Precision of Proximal Femoral Finite Element Modeling. Annals of Biomedical Engineering. 2000;28(4):408-14. doi: 10.1114/1.278.
27. Raja Izaham RMA, Abdul Kadir MR, Abdul Rashid AH, Hossain MG, Kamarul T. Finite element analysis of Puddu and Tomofix plate fixation for open wedge high tibial osteotomy. Injury. 2012;43(6):898-902. doi: 10.1016/j.injury.2011.12.006.
28. Tuncer M, Cobb JP, Hansen UN, Amis AA. Validation of multiple subject-specific finite element models of unicompartmental knee replacement. Medical Engineering & Physics. 2013;35(10):1457-64. doi: 10.1016/j.medengphy.2013.03.020
29. Zhao X, Chosa E, Yamako G, Watanabe S, Deng G, Totoribe K. Effect of Acetabular Reinforcement Ring With Hook for Acetabular Dysplasia Clarified by Three-Dimensional Finite Element Analysis. The Journal of Arthroplasty. 2013;28(10):1765-9. doi: 10.1016/j.arth.2013.07.026
30. MR AK. Computational biomechanics of the hip joint. Berlin: Springer. 2014. doi: 10.1007/978-3-642-38777-7
31. Terrier A, Larrea X, Guerdat J, Crevoisier X. Development and experimental validation of a finite element model of total ankle replacement. Journal of Biomechanics. 2014;47(3):742-5. doi: 10.1016/j.jbiomech.2013.12.022.
32. Hopkins AR, New AM, Rodriguez-y-Baena F, Taylor M. Finite element analysis of unicompartmental knee arthroplasty. Medical Engineering & Physics. 2010;32(1):14-21. doi: 10.1016/j.medengphy.2009.10.002
33. Wang Y, Wong DW, Zhang M. Computational Models of the Foot and Ankle for Pathomechanics and Clinical Applications: A Review. Ann Biomed Eng. 2016;44(1):213-21. doi: 10.1007/s10439-015-1359-7.
34. Ramlee MH, Kadir MRA, Harun H. Three-Dimensional Modelling and Finite Element Analysis of an Ankle External Fixator. Advanced Materials Research. 2014;845:183-8. doi: 10.4028/www.scientific.net/AMR.845.183
35. Roland M, Tjardes T, Otchwemah R, Bouillon B, Diebels S. An optimization algorithm for individualized biomechanical analysis and simulation of tibia fractures. Journal of Biomechanics. 2015;48(6):1119-24. doi: 10.1016/j.jbiomech.2015.01.015
36. Abdullah AH, Todo MN, Nakashima Y. Stress and damage formation analysis in hip arthroplasties using CT-based finite element method. Journal of Engineering and Applied Sciences, 12(10), 2715-2719. 2017. doi: 10.3923/jeasci.2017.2715.2719
37. Ramlee MH, Kadir MRA, Harun H. Three-dimensional modeling and analysis of a human ankle joint. 2013 IEEE Student Conference on Research and Developement; 2013 16-17 Dec. 2013. doi: 10.1109/SCOReD.2013.7002545
38. MUNCKHOF SVD, NIKOOYAN AA, ZADPOOR AA. ASSESSMENT OF OSTEOPOROTIC FEMORAL FRACTURE RISK: FINITE ELEMENT METHOD AS A POTENTIAL REPLACEMENT FOR CURRENT CLINICAL TECHNIQUES. Journal of Mechanics in Medicine and Biology. 2015;15(03):1530003. doi: 10.1142/S0219519415300033
39. Qiu T-X, Teo E-C, Yan Y-B, Lei W. Finite element modeling of a 3D coupled foot–boot model. Medical Engineering & Physics. 2011;33(10):1228-33. doi: 10.1016/j.medengphy.2011.05.012
40. Radcliffe IAJ, Taylor M. Investigation into the affect of cementing techniques on load transfer in the resurfaced femoral head: A multi-femur finite element analysis. Clinical Biomechanics. 2007;22(4):422-30. doi: 10.1016/j.clinbiomech.2006.12.001
41. Wittek A, Grosland NM, Joldes GR, Magnotta V, Miller K. From Finite Element Meshes to Clouds of Points: A Review of Methods for Generation of Computational Biomechanics Models for Patient-Specific Applications. Ann Biomed Eng. 2016;44(1):3-15. doi: 10.1007/s10439-015-1469-2.
42. Izmin NAN, Todo M, AH A. Prediction of bone damage formation in resurfacing hip arthroplasty. International Journal of Engineering and Advanced Technology. 2019. doi: 10.1016/j.medengphy.2017.03.006
43. Yosibash Z, Trabelsi N, Milgrom C. Reliable simulations of the human proximal femur by high-order finite element analysis validated by experimental observations. Journal of Biomechanics. 2007;40(16):3688-99. doi: 10.1016/j.jbiomech.2007.06.017.
44. Abdullah AH, Todo M, Nakashima Y. Prediction of damage formation in hip arthroplasties by finite element analysis using computed tomography images. Medical Engineering & Physics. 2017;44:8-15. doi: 10.1016/j.medengphy.2017.03.006
45. Wong DW, Niu W, Wang Y, Zhang M. Finite

Element Analysis of Foot and Ankle Impact Injury: Risk Evaluation of Calcaneus and Talus Fracture. PLoS One. 2016;11(4):e0154435. doi: 10.1371/journal.pone.0154435
46. Ramlee MH, Abdul Kadir MR, Murali MR, Kamarul T. Finite element analysis of three commonly used external fixation devices for treating Type III pilon fractures. Medical Engineering & Physics. 2014;36(10):1322-30. doi: 10.1016/j.medengphy.2014.05.015.
47. Miyoshi S, Takahashi T, Ohtani M, Yamamoto H, Kameyama K. Analysis of the shape of the tibial tray in total knee arthroplasty using a three dimension finite element model. Clinical Biomechanics. 2002;17(7):521-5. doi: 10.1016/s0268-0033(02)00064-5.
48. Completo A, Rego A, Fonseca F, Ramos A, Relvas C, Simxes JA. Biomechanical evaluation of proximal tibia behaviour with the use of femoral stems in revision TKA: An in vitro and finite element analysis. Clinical Biomechanics. 2010;25(2):159-65. doi: 10.1016/j.clinbiomech.2009.10.011.
49. Dai X-Q, Li Y, Zhang M, Cheung JT-M. Effect of sock on biomechanical responses of foot during walking. Clinical Biomechanics. 2006;21(3):314-21. doi: 10.1016/j.clinbiomech.2005.10.002.
50. Wang CJ, Yettram AL, Yao MS, Procter P. Finite element analysis of a Gamma nail within a fractured femur. Medical Engineering & Physics. 1998;20(9):677-83. doi: 10.1016/s1350-4533(98)00079-4.
51. Jay Elliot B, Gundapaneni D, Goswami T. Finite element analysis of stress and wear characterization in total ankle replacements. Journal of the Mechanical Behavior of Biomedical Materials. 2014;34:134-45. doi: 10.1016/j.jmbbm.2014.01.020.
52. Dopico-González C, New AM, Browne M. Probabilistic finite element analysis of the uncemented hip replacement—effect of femur characteristics and implant design geometry. Journal of Biomechanics. 2010;43(3):512-20. doi: 10.1016/j.jbiomech.2009.09.039
53. Liu X, Zhang M. Redistribution of knee stress using laterally wedged insole intervention: Finite element analysis of knee–ankle–foot complex. Clinical Biomechanics. 2013;28(1):61-7. doi: 10.1016/j.clinbiomech.2012.10.004.
54. Viceconti M, Bellingeri L, Cristofolini L, Toni A. A comparative study on different methods of automatic mesh generation of human femurs. Medical Engineering & Physics. 1998;20(1):1-10. doi: 10.1016/s1350-4533(97)00049-0.
55. Innocenti B, Pianigiani S, Ramundo G, Thienpont E. Biomechanical Effects of Different Varus and Valgus Alignments in Medial Unicompartmental Knee Arthroplasty. The Journal of Arthroplasty. 2016;31(12):2685-91. doi: 10.1016/j.arth.2016.07.006.
56. Bah MT, Nair PB, Browne M. Mesh morphing for finite element analysis of implant positioning in cementless total hip replacements. Medical Engineering & Physics. 2009;31(10):1235-43. doi: 10.1016/j.medengphy.2009.08.001.
57. Hulzer A, Schruder C, Woiczinski M, Sadoghi P, Scharpf A, Heimkes B, et al. Subject-specific finite element simulation of the human femur considering inhomogeneous material properties: A straightforward method and convergence study. Computer Methods and Programs in Biomedicine. 2013;110(1):82-8. doi: 10.1016/j.cmpb.2012.09.010.
58. Conlisk N, Howie CR, Pankaj P. An efficient method to capture the impact of total knee replacement on a variety of simulated patient types: A finite element study. Medical Engineering & Physics. 2016;38(9):959-68. doi: 10.1016/j.medengphy.2016.06.014
59. Ma C-H, Wu C-H, Tu Y-K, Lin T-S. Metaphyseal locking plate as a definitive external fixator for treating open tibial fractures—Clinical outcome and a finite element study. Injury. 2013;44(8):1097-101. doi: 10.1016/j.injury.2013.04.023.
60. Wang Y, Wong DW-C, Zhang M. Computational Models of the Foot and Ankle for Pathomechanics and Clinical Applications: A Review. Annals of Biomedical Engineering. 2016;44(1):213-21. doi: 10.1007/s10439-015-1359-7.
61. Goebel P, Kluess D, Wieding J, Souffrant R, Heyer H, Sander M, et al. The influence of head diameter and wall thickness on deformations of metallic acetabular press-fit cups and UHMWPE liners: a finite element analysis. J Orthop Sci. 2013;18(2):264-70. doi: 10.1007/s00776-012-0340-7
62. Pérez MA, Palacios J. Comparative Finite Element Analysis of the Debonding Process in Different Concepts of Cemented Hip Implants. Annals of Biomedical Engineering. 2010;38(6):2093-106. doi: 10.1007/s10439-010-9996-3
63. Clary CW, Fitzpatrick CK, Maletsky LP, Rullkoetter PJ. The influence of total knee arthroplasty geometry on mid-flexion stability: An experimental and finite element study. Journal of Biomechanics. 2013;46(7):1351-7. doi: 10.1016/j.jbiomech.2013.01.025
64. Kaman MO, Celik N, Karakuzu S. Numerical Stress Analysis of the Plates Used to Treat the Tibia Bone Fracture. Journal of Applied Mathematics and Physics. 2014;Vol.02No.06:6. doi: 10.4236/jamp.2014.26036
65. Sawatari T, Tsumura H, Iesaka K, Furushiro Y, Torisu T. Three-dimensional finite element analysis of unicompartmental knee arthroplasty–the influence of tibial component inclination. Journal of Orthopaedic Research. 2005;23(3):549-54. doi: 10.1016/j.orthres.2004.06.007
66. Abdul-Kadir MR, Hansen U, Klabunde R, Lucas

D, Amis A. Finite element modelling of primary hip stem stability: The effect of interference fit. Journal of Biomechanics. 2008;41(3):587-94. doi: 10.1016/j.jbiomech.2007.10.009
67. Maurel N, Diop A, Grimberg J, Elise S. In vitro biomechanical analysis of glenoïds before and after implantation of prosthetic components. Journal of Biomechanics. 2002;35(8):1071-80. doi: 10.1016/s0021-9290(02)00065-9.
68. Diop A, Maurel N, Grimberg J, Gagey O. Influence of glenohumeral mismatch on bone strains and implant displacements in implanted glenoпds. An in vitro experimental study on cadaveric scapulae. Journal of Biomechanics. 2006;39(16):3026-35. doi: 10.1016/j.jbiomech.2005.10.015
69. Pomwenger W, Entacher K, Resch H, Schuller-Gцtzburg P. Multi-patient finite element simulation of keeled versus pegged glenoid implant designs in shoulder arthroplasty. Medical & Biological Engineering & Computing. 2015;53(9):781-90. doi: 10.1007/s11517-015-1286-7.
70. Cheung JT-M, Zhang M, An K-N. Effect of Achilles tendon loading on plantar fascia tension in the standing foot. Clinical Biomechanics. 2006;21(2):194-203. doi: 10.1016/j.clinbiomech.2005.09.016.
71. Gray HA, Taddei F, Zavatsky AB, Cristofolini L, Gill HS. Experimental Validation of a Finite Element Model of a Human Cadaveric Tibia. Journal of Biomechanical Engineering. 2008;130(3). doi: 10.1115/1.2913335.
72. Taddei F, Cristofolini L, Martelli S, Gill HS, Viceconti M. Subject-specific finite element models of long bones: An in vitro evaluation of the overall accuracy. Journal of Biomechanics. 2006;39(13):2457-67. doi: 10.1016/j.jbiomech.2005.07.018
73. Rogge RD, Adams BD, Goel VK. An analysis of bone stresses and fixation stability using a finite element model of simulated distal radius fractures. The Journal of Hand Surgery. 2002;27(1):86-92. doi: 10.1053/jhsu.2002.29485.
74. Pistoia W, van Rietbergen B, Lochmüller EM, Lill CA, Eckstein F, Rьegsegger P. Image-Based Micro-Finite-Element Modeling for Improved Distal Radius Strength Diagnosis: Moving From “Bench” to “Bedside”. Journal of Clinical Densitometry. 2004;7(2):153-60. doi: 10.1385/jcd:7:2:153.
75. Iesaka K, Kummer FJ, Di Cesare PE. Stress Risers Between Two Ipsilateral Intramedullary Stems: A Finite-Element and Biomechanical Analysis. The Journal of Arthroplasty. 2005;20(3):386-91. doi: 10.1016/j.arth.2004.05.002.
76. Chen Y-N, Lee P-Y, Chang C-H, Chang C-W, Ho Y-H, Li C-T, et al. Computational comparison of tibial diaphyseal fractures fixed with various degrees of prebending of titanium elastic nails and with and without end caps. Injury. 2016;47(10):2339-46. doi: 10.1016/j.injury.2016.07.001.
77. Walch G, Young AA, Melis B, Gazielly D, Loew M, Boileau P. Results of a convex-back cemented keeled glenoid component in primary osteoarthritis: multicenter study with a follow-up greater than 5 years. Journal of Shoulder and Elbow Surgery. 2011;20(3):385-94. doi: 10.1016/j.jse.2010.07.011
78. Büchler P, Ramaniraka NA, Rakotomanana LR, Iannotti JP, Farron A. A finite element model of the shoulder: application to the comparison of normal and osteoarthritic joints. Clinical Biomechanics. 2002;17(9):630-9. doi: 10.1016/s0268-0033(02)00106-7.
79. Orr TE, Carter DR, Schurman DJ. Stress analyses of glenoid component designs. Clin Orthop Relat Res. 1988(232):217-24. doi: 10.1097/00003086-198807000-00029
80. Stone KD, Grabowski JJ, Cofield RH, Morrey BF, An KN. Stress analyses of glenoid components in total shoulder arthroplasty. Journal of Shoulder and Elbow Surgery. 1999;8(2):151-8. doi: 10.1016/s1058-2746(99)90009-5.
81. Murphy LA, Prendergast PJ, Resch H. Structural analysis of an offset-keel design glenoid component compared with a center-keel design. Journal of Shoulder and Elbow Surgery. 2001;10(6):568-79. doi: 10.1067/mse.2001.118630.
82. Lacroix D, Murphy LA, Prendergast PJ. Three-Dimensional Finite Element Analysis of Glenoid Replacement Prostheses: A Comparison of Keeled and Pegged Anchorage Systems. Journal of Biomechanical Engineering. 2000;122(4):430-6. doi: 10.1115/1.1286318.
83. Aldinger PR, Raiss P, Rickert M, Loew M. Complications in shoulder arthroplasty: an analysis of 485 cases. International Orthopaedics. 2010;34(4):517-24. doi: 10.1007/s00264-009-0780-7
84. Fox TJ, Cil A, Sperling JW, Sanchez-Sotelo J, Schleck CD, Cofield RH. Survival of the glenoid component in shoulder arthroplasty. Journal of Shoulder and Elbow Surgery. 2009;18(6):859-63. doi: 10.1016/j.jse.2008.11.020
85. Severt R, Thomas BJ, Tsenter MJ, Amstutz HC, Kabo JM. The influence of conformity and constraint on translational forces and frictional torque in total shoulder arthroplasty. Clin Orthop Relat Res. 1993(292):151-8. doi: 10.1097/00003086-199307000-00019
86. Anglin C, Wyss UP, Pichora DR. Shoulder prosthesis subluxation: Theory and experiment. Journal of Shoulder and Elbow Surgery. 2000;9(2):104-14. doi: 10.1067/mse.2000.105139
87. Oosterom R, Herder JL, van der Helm FCT, Święszkowski W, Bersee HEN. Translational stiffness of the replaced shoulder joint. Journal of Biomechanics. 2003;36(12):1897-907. doi: 10.1016/s0021-9290(03)00192-1.
88. Hopkins AR, Hansen UN, Amis AA, Taylor M, Gronau N, Anglin C. Finite element modelling

of glenohumeral kinematics following total shoulder arthroplasty. Journal of Biomechanics. 2006;39(13):2476-83. doi: 10.1016/j.jbiomech.2005.07.031
89. Antoniac IV, Stoia DI, Ghiban B, Tecu C, Miculescu F, Vigaru C, et al. Failure Analysis of a Humeral Shaft Locking Compression Plate—Surface Investigation and Simulation by Finite Element Method. Materials. 2019;12(7):1128. doi: 10.3390/ma12071128.
90. Konrad G, Bayer J, Hepp P, Voigt C, Oestern H, Kääb M, et al. Open Reduction and Internal Fixation of Proximal Humeral Fractures with Use of the Locking Proximal Humerus Plate: Surgical Technique. JBJS. 2010;92(Supplement_1_Part_1):85-95. doi: 10.2106/JBJS.I.01462.
91. Alexandru L, Haragus H, Deleanu B, Timar B, Poenaru DV, Vlad DC. Haematology panel biomarkers for humeral, femoral, and tibial diaphyseal fractures. International Orthopaedics. 2019;43(7):1567-72. doi: 10.1007/s00264-019-04305-1.
92. Pidhorz L. Acute and chronic humeral shaft fractures in adults. Orthopaedics & Traumatology: Surgery & Research. 2015;101(1, Supplement):S41-S9. doi: 10.1016/j.otsr.2014.07.034
93. Davidovitch R. Shoulder and Elbow Fractures of the Humeral Shaft. Cancer Therapy Advisor. 2017. https://www.cancertherapyadvisor.com/
94. Kumar V, Rathinam M. Fractures of the shaft of humerus. Orthopaedics and Trauma. 2013;27(6):393-402. doi: 10.1016/j.mporth.2013.09.001
95. Masih C, Francis A, Shriwastava A, Diwedi N, Tiwari P, Nareliya R, et al. Biomechanical evaluation of human humerus and scapula bone: a review. Journal of Biomedical and Bioengineering. 2012;3(1):63-6. Retrieved from http://www.bioinfo.in/contents.php?id=87
96. Masih C, Nareliya R, Kumar V, editors. Finite Element Application to Human Humerus Bone: A Biomechanical Study2013; India: Springer India. doi: 10.1007/978-81-322-0970-6_12
97. Calori GM, Colombo M, Bucci MS, Fadigati P, Colombo AIM, Mazzola S, et al. Complications in proximal humeral fractures. Injury. 2016;47:S54-S8. doi: 10.1016/j.injury.2016.07.039
98. Lungu R, Borgazi E, Lungu M, Popa D, Tutunea D, MX C. New methods for the simulation with finite element of the human elbow. Proceedings of the international conference on circuits, systems, signals. 2010:pp 45–50. Retrieved from https://www.scimagojr.com/journalsearch.php?q=19900191924&tip=sid&clean=0
99. Lee BP, Adams RA, Morrey BF. Polyethylene Wear After Total Elbow Arthroplasty. JBJS. 2005;87(5):1080-7. doi: 10.2106/JBJS.D.02163.
100. Banagan KE, Murthi AM. Current concepts in total elbow arthroplasty. Current Opinion in Orthopaedics. 2006;17(4):335-9. doi: 10.1097/01.bco.0000233729.71880.dc
101. Loebenberg MI, Adams R, O'Driscoll SW, Morrey BF. Impaction Grafting in Revision Total Elbow Arthroplasty. JBJS. 2005;87(1):99-106.doi: 10.2106/JBJS.B.00038
102. Müller SA, King GJW, Johnson JA. Total Elbow Arthroplasty: Design Considerations. In: King GJW, Rizzo M, editors. Arthroplasty of the Upper Extremity: A Clinical Guide from Elbow to Fingers. Cham: Springer International Publishing; 2021. p. 3-19. doi: 10.1007/978-3-030-68880-6_1
103. Morrey BF, Askew LJ, Chao EY. A biomechanical study of normal functional elbow motion. J Bone Joint Surg Am. 1981;63(6):872-7. doi: 10.2106/00004623-198163060-00002
104. BF M. The elbow and its disorders. Philadelphia: WB Saunders. 2000. doi: 10.1016/S0025-6196(12)60750-2
105. Boone DC, Azen SP. Normal range of motion of joints in male subjects. JBJS. 1979;61(5):756-9. doi: 10.2106/00004623-197961050-00017
106. Tarnita D, Boborelu C, Popa D, Tarnita D-N, editors. Design and Finite Element Analysis of a New Spherical Prosthesis-Elbow Joint Assembly2018; Cham: Springer International Publishing. doi: 10.1007/978-3-319-79111-1_12
107. TarniŢă D. Wearable sensors used for human gait analysis. Rom J Morphol Embryol. 2016;57(2):373-82. Retrieved from https://www.rjme.ro/
108. TarniŢă D, TarniŢă DN. Experimental measurement of flexion-extension movement in normal and corpse prosthetic elbow joint. Rom J Morphol Embryol. 2016;57(1):145-51. Retrieved from https://www.rjme.ro/
109. Tarnita D, Popa D, Boborelu C, Dumitru N, Calafeteanu D, Tarnita DN, editors. Experimental Bench Used to Test Human Elbow Endoprosthesis 2015; Cham: Springer International Publishing. doi: 10.1007/978-3-319-09411-3_71
110. Radakisnin R, Mamat N, Majid MSA, Nasir NFM, editors. A FINITE ELEMENT ANALYSIS OF ELBOW JOINT IN DAILY ACTIVITIES2015. Retrieved from http://www.arpnjournals.com/jeas/
111. Waters PM, Mintzer CM, Hipp JA, Snyder BD. Noninvasive measurement of distal radius instability. The Journal of Hand Surgery. 1997;22(4):572-9. doi: 10.1016/S0363-5023(97)80111-6.
112. Winemaker MJ, Chinchalkar S, Richards RS, Johnson JA, Chess DG, King GJW. Load relaxation and forces with activity in hoffman external fixators: A clinical study in patients with Colles' fractures. The Journal of Hand Surgery. 1998;23(5):926-32. doi: 10.1016/S0363-5023(98)80175-5.
113. Jupiter JB. Fractures of the distal end of the radius. J Bone Joint Surg Am. 1991;73(3):461-9. doi: 10.2106/00004623-199173030-00019
114. Graff S, Jupiter J. Fracture of the distal radius: Classification of treatment and indications for

external fixation. Injury. 1994;25:SD14-SD25. doi: 10.1016/0020-1383(95)90125-6.
115. Short WH, Palmer AK, Werner FW, Murphy DJ. A biomechanical study of distal radial fractures. The Journal of Hand Surgery. 1987;12(4):529-34. doi: 10.1016/s0363-5023(87)80202-2.
116. Pike LM, SW W. Alternatives to bone graft in the treatment of distal radius fractures. Atlas Hand Clin. 1997:2:125–50.
117. Court-Brown CM, Caesar B. Epidemiology of adult fractures: A review. Injury. 2006;37(8):691-7. doi: 10.1016/j.injury.2006.04.130
118. Disseldorp DJ, Hannemann PF, Poeze M, Brink PR. Dorsal or Volar Plate Fixation of the Distal Radius: Does the Complication Rate Help Us to Choose? J Wrist Surg. 2016;5(3):202-10. doi: 10.1055/s-0036-1571842
119. Lin C-L, Lin Y-H, Chen AC-Y. Buttressing angle of the double-plating fixation of a distal radius fracture: a finite element study. Medical and Biological Engineering and Computing. 2006;44(8):665-73. doi: 10.1007/s11517-006-0082-9
120. Liu HC, Jiang J-S, Lin C-L. Biomechanical investigation of a novel hybrid dorsal double plating for distal radius fractures by integrating topology optimization and finite element analysis. Injury. 2020;51(6):1271-80. doi: 10.1016/j.injury.2020.03.011
121. RIIS J, FRUENSGAARD S. Treatment of Unstable Colles' Fractures by External Fixation. Journal of Hand Surgery. 1989;14(2):145-8. doi: 10.1016/0266-7681_89_90115-0.
122. Cooney WP, 3rd, Dobyns JH, Linscheid RL. Complications of Colles' fractures. J Bone Joint Surg Am. 1980;62(4):613-9. doi: 10.2106/00004623-198062040-00016
123. S. K. Muscle strength. London, United Kingdom: Taylor & Francis Ltd. 2004:p. 224. doi: 10.1201/9780203503591
124. Brown CJ, Wang CJ, Yettram AL, Procter P. Intramedullary nails with two lag screws. Clinical Biomechanics. 2004;19(5):519-25. doi: 10.1016/j.clinbiomech.2004.01.004.
125. Lin YH, Lin CL, Kuo HN, Sun MT, ACY C. Biomechanical analysis of volar and dorsal locking plates for fixation in comminuted extra-articular articular distal radius fractures: a 3D finite element study. J Med Biol Eng. 2011.doi: 10.5405/jmbe.1003
126. Carrozzella J, Stern PJ. Treatment of Comminuted Distal Radius Fractures with Pins and Plaster. Hand Clinics. 1988;4(3):391-7. doi: 10.1016/S0749-0712(21)01156-2
127. Greatting MD, Bishop AT. Intrafocal (Kapandji) pinning of unstable fractures of the distal radius. Orthop Clin North Am. 1993;24(2):301-7. doi: 10.1016/S0030-5898(21)00018-3
128. A. RD, P. R. FRACTURES OF THE DISTAL END OF THE RADIUS TREATED BY INTERNAL FIXATION AND EARLY FUNCTION. The Journal of Bone and Joint Surgery British volume. 1996;78-B(4):588-92. doi: 10.1302/0301-620x.78b4.0780588
129. Naidu SH, Capo JT, Moulton M, Ciccone W, Radin A. Percutaneous pinning of distal radius fractures: A biomechanical study. The Journal of Hand Surgery. 1997;22(2):252-7. doi: 10.1016/S0363-5023(97)80159-1.
130. Jones DJ, Henley MB, Schemitsch EH, Tencer AF. A biomechanical comparison of two methods of fixation of fractures of the forearm. J Orthop Trauma. 1995;9(3):198-206. doi: 10.1097/00005131-199506000-00004.
131. Habernek H, Weinstabl R, Fialka C, Schmid L. Unstable distal radius fractures treated by modified Kirschner wire pinning: anatomic considerations, technique, and results. J Trauma. 1994;36(1):83-8. doi: 10.1097/00005373-199401000-00013.
132. Schuind F, Donkerwolcke M, Burny F. External Fixation of Wrist Fractures. Orthopedics. 1984;7(5):841-4. doi: 10.3928/0147-7447-19840501-09
133. Berger RA, Crowninshield RD, Flatt AE. The three-dimensional rotational behaviors of the carpal bones. Clin Orthop Relat Res. 1982(167):303-10. doi: 10.1097/00003086-198207000-00047
134. Zhang J, Ebraheim N, Lausé GE, Xiao B, Xu R. A comparison of absorbable screws and metallic plates in treating calcaneal fractures: A prospective randomized trial. Journal of Trauma and Acute Care Surgery. 2012;72(2):E106-E10. doi: 10.1097/ta.0b013e3182231811.
135. Zhang J, Xiao B, Wu Z. Surgical treatment of calcaneal fractures with bioabsorbable screws. International Orthopaedics. 2011;35(4):529-33. doi: 10.1007/s00264-010-1183-5
136. Pelto K, Hirvensalo E, Bustman O, Rokkanen P. Treatment of radial head fractures with absorbable polyglycolide pins: a study on the security of the fixation in 38 cases. J Orthop Trauma. 1994;8(2):94-8. doi: 10.1097/00005131-199404000-00003.
137. Shi X, Pan T, Wu D, Cai N, Chen R, Li B, et al. Effect of different orientations of screw fixation for radial head fractures: a biomechanical comparison. Journal of Orthopaedic Surgery and Research. 2017;12(1):143. doi: 10.1186/s13018-017-0641-9.
138. Stegeman M, Rijnberg WJ, van Loon CJM. Biaxial total wrist arthroplasty in rheumatoid arthritis. Satisfactory functional results. Rheumatology International. 2005;25(3):191-4. doi: 10.1007/s00296-003-0413-1.
139. Bajuri MN, Abdul Kadir MR, Murali MR, Kamarul T. Biomechanical analysis of the wrist arthroplasty in rheumatoid arthritis: a finite element analysis. Medical & Biological Engineering & Computing. 2013;51(1):175-86. doi: 10.1007/s11517-012-0982-9.
140. Bajuri MN, Kadir MRA, Raman MM, Kamarul

T. Mechanical and functional assessment of the wrist affected by rheumatoid arthritis: A finite element analysis. Medical Engineering & Physics. 2012;34(9):1294-302. doi: 10.1016/j.medengphy.2011.12.020

141. McCullough MBA, Adams BD, Grosland NM. THE EFFECT OF ARTICULAR SURFACE SHAPE AND TENDON FORCES OF TOTAL WRIST ARTHROPLASTY SYSTEMS: A FINITE ELEMENT STUDY. Journal of Musculoskeletal Research. 2012;15(04):1250021. doi: 10.1142/S0218957712500212
142. Mianroodi M, Touchal S. Finite element study of a wrist prosthesis. Journal of Basic Research in Medical Sciences. 2019;6(3):49-55. Retrieved from https://jbrms.medilam.ac.ir/
143. Trieb K, Hofstätter S. Rheumatoid Arthritis of the Wrist. Techniques in Orthopaedics. 2009;24(1):8-12. doi: 10.1097/BTO.0b013e3181a32a36
144. Gupta A. Total wrist arthroplasty. Am J Orthop (Belle Mead NJ). 2008;37(8 Suppl 1):12-6. doi: 10.1186/1753-6561-9-S3-A86
145. Cavaliere CM, Chung KC. Total Wrist Arthroplasty and Total Wrist Arthrodesis in Rheumatoid Arthritis: A Decision Analysis From the Hand Surgeons' Perspective. The Journal of Hand Surgery. 2008;33(10):1744-55.e2. doi: 10.1016/j.jhsa.2008.06.022.
146. Herzberg G. Prospective study of a new total wrist arthroplasty: Short term results. Chirurgie de la Main. 2011;30(1):20-5. doi: 10.1016/j.main.2011.01.017
147. Adams BD. Total wrist arthroplasty for rheumatoid arthritis. International Congress Series. 2006;1295:83-93. doi: 10.1016/j.ics.2006.03.031
148. Radmer S, Andresen R, Sparmann M. Wrist arthroplasty with a new generation of prostheses in patients with rheumatoid arthritis. The Journal of Hand Surgery. 1999;24(5):935-43. doi: 10.1053/jhsu.1999.0935.
149. Gislason MK, Nash DH, Nicol A, Kanellopoulos A, Bransby-Zachary M, Hems T, et al. A three-dimensional finite element model of maximal grip loading in the human wrist. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine. 2009;223(7):849-61. doi: 10.1243/09544119JEIM527.
150. Gíslason MK, Stansfield B, Nash DH. Finite element model creation and stability considerations of complex biological articulation: The human wrist joint. Medical Engineering & Physics. 2010;32(5):523-31. doi: 10.1016/j.medengphy.2010.02.015
151. Carrigan SD, Whiteside RA, Pichora DR, Small CF. Development of a Three-Dimensional Finite Element Model for Carpal Load Transmission in a Static Neutral Posture. Annals of Biomedical Engineering. 2003;31(6):718-25. doi: 10.1114/1.1574027.
152. Cush JJ, Lipsky PE. Cellular basis for rheumatoid inflammation. Clin Orthop Relat Res. 1991(265):9-22. doi: 10.1097/00003086-199104000-00003
153. Ramlee MH, Gan KB. Function and biomechanics of upper limb in post-stroke patients - a systematic review. Journal of Mechanics in Medicine and Biology. 2017;17(6):1750099. doi: 10.1142/S0219519417500993
154. Zainal Abidin NA, Abdul Kadir MR, Ramlee MH. Three-dimensional finite element modelling and analysis of human knee joint - Model verification. Journal of Physics: Conference Series. 2019;1372(1):012608. doi: 10.1088/1742-6596/1372/1/012068
155. Clavert P, Zerah M, Krier J, Mille P, Kempf JF, Kahn JL. Finite element analysis of the strain distribution in the humeral head tubercles during abduction: comparison of young and osteoporotic bone. Surgical and Radiologic Anatomy. 2006;28(6):581-7. doi: 10.1007/s00276-006-0140-x.
156. Zainal Abidin NA, Abdul Kadir MR, Ramlee MH. Biomechanical effects of different lengths of cross-pins in anterior cruciate ligament reconstruction: a finite element analysis. Journal of Mechanics in Medicine and Biology. 2020;20(7):2050047. doi: 10.1142/S0219519420500475
157. Abd Aziz AM, Gan HS, Nasution AK, Abdul Kadir MR, Ramlee MH. Development and verification of three-dimensional model of femoral bone: finite element analysis. Journal of Physics: Conference Series. 2019;1372(1):012014. doi: 10.1088/1742-6596/1372/1/012014