Malaysian Journal of Medicine and Health Sciences (eISSN 2636-9346)

REVIEW ARTICLE

Finite Element Modelling and Simulation for Lower Limb of Human Bone: A Review

Muhammad Hanif Ramlee[1,2]

[1] Medical Devices and Technology Centre (MEDiTEC), Institute for Human Centered Engineering (iHumEn), Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia
[2] School of Biomedical Engineering and Health Science, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia

ABSTRACT

Most orthopaedic cases that involved with bone fracture are normally treated with medical implants. To be noticed that some precautions in terms of biomechanical and biomaterial properties are necessary for a successful post-surgery process. The biomechanical evaluation of implants could be carried out using computing and engineering technologies. However, in the computer simulation, some assumptions are needed as the limitations on computer resources and data input. This review focuses on the current method of developing the finite element model for patients with specific values of material properties for lower limb part such as hip, knee and ankle joint. Previous literature was reviewed from which keywords and search engines were identified. In this review, inclusion and exclusion criteria were used to limit the literature search. We reviewed the state-of-the-art in this area and provide recommendations for future research. In conclusion, the previous published reports illustrated different methods to develop numerical models.

Keywords: Finite Element, Implant, Orthopaedics, Biomechanics, Fixator, Lower limb

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

INTRODUCTION

Most orthopaedic cases involving fractures, dislocation, and deformities of human bone are normally treated by medical experts using special implants with different materials (1-4). Such pathological problems could be caused by car accidents, falling from high altitudes, sports injuries and aging (2). Therefore, special surgical treatments and management are expected of medical surgeons to restore the normal human functions of the injured bone. However, precautions concerning biomechanics should be observed before surgery, as many complications reported from previous literature are related to misunderstanding the biomechanical behaviour of implants (3, 5). Understanding the biomechanical characteristics of the fixated implants is the key for a successful post-surgery procedure. When medical experts use the correct fundamental of the biomechanics and biomaterials, it can reduce complications such as mal-union and secondary fractures.

There are various concerns regarding the cost management and efficiency of the implants used in the future generation. To do that, various methods such as observational, functional score systems and experimental studies were implemented (6). Even though these methods are popular and reliable among researchers, however, detailed biomechanics of the inner human tissue, such as stress and deformation, are difficult to evaluate. For that reason, the computational finite element (FE) analysis has become a vital assistive tool in surgical planning and the biomechanical evaluation of human musculoskeletal structures (7, 8). The results in terms of stress distribution, displacement and micromovement can be used to monitor the conditions of patients. This can help to improve implant design, the choice of suitable biomaterials for implants as well as surgical techniques and help researchers develop more favourable techniques for long-term bone and tissue repair (9).

Computational FE model and analysis are rapidly closing on creating a biomechanical study of patient-specific model of the human lower limb. This review article summarises the state-of-the-art in this area. We focus on the methods for generating a patient-specific computational model via the FE method and area-specific models of the lower limbs of the human body. The current technologies and challenges in FE model development are included and biomechanical analysis of implants and its conclusion are described in the accessibility of orthopaedic applications.

METHODS

Strategy of literature search

In this review, searches were conducted using several keywords and search engines. The following specific search strategies were used to review current evidence from the literature:

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", "micromovement", "stress", "orthopaedics", "internal fixation", "external fixation".

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

Inclusion and exclusion criteria

Our investigation and review were conducted followed by inclusion and exclusion criteria. For the inclusion criteria, previous studies were considered and included if they met the following specific criteria: (1) Articles published in English only; (2) The studies were published between January 1980 and December 2019; (3) All types of implants used were included; (4) The studies included the biomechanical aspects; (5) The studies' observation endpoint was the onset of implant design.

For the exclusion criteria, these are based on the following: (1) Biomechanical reports for children were not included; (2) The studies examined comparison between geography or country, were excluded; (3) Duplicate publications; (4) Animal experiments were not considered.

RESULTS AND DISCUSSION

Overview of the finite element

Finite element models of the human body were normally developed from two-dimensional (2D) images into a three-dimensional (3D) model (5, 10). The 2D images can be obtained from a Computed Tomography (CT) scanner or Magnetic Resonance Imaging (MRI) system which can acquire more than 300 slices images per scan (4, 11, 12). In the image acquisition process, some authors used patient-specific subjects (8, 13) while others used synthetic bone (Sawbones) (14) to obtain the 2D images from the scanner. The 3D model can be developed using commercial software such as Mimics and Amira (15-17). This simulated model with simplified and partial structures was used to evaluate the force loading of the human body when implants are fixated on the bone (18, 19).

Reconstruction of human tissue

In the study of orthopaedic research using the FE method, the first step is to reconstruct a 3D model of a human bone. To date, the CT or MRI dataset of the subject-specific or synthetic bone will be used to develop a 3D model via a segmentation procedure using a huge number of 2D images (20-22). To be noted, bone and tissue in the human body consist of different attenuation coefficients resulting in different grey values. Bone can be developed based on these values in the process of segmentation. There are three available research software including (1) Mimics (Fig. 1) (5, 23), (2) Simpleware (24) and (3) Scan IP (25), that provide segmentation tools and other techniques to decrease noise effects of the 2D images. Other than this commercial software, there is also an open-source package (3D Slicer, OsiriX and Mechanical Finder) that can be used for image segmentation (26, 27). The segmentation of the images can be performed in many methods, and no uniform procedure is preferred. For example, a Hounsfield unit of 700 was set to distinguish between cortical and cancellous bones (Hu>700 is cortical and Hu<700 is cancellous) (28). In addition, there is some finite element models were assigned with grey data values in Hu for each elements (22, 27, 29). Other soft tissues such as cartilage and ligament were modelled manually using specific software. The cartilage was extruded from their respective bone layers via Mimics and 3-matic software (11, 30). While the ligament was modelled via FE packages (5, 30).

Modelling and meshing

For the FE analysis, the 3D model from segmented images should mesh and it can be done with FE software such as ANSYS, Marc.Mentat, NASRAN, Catia and Abaqus (12, 31-36). The mesh generation can also be done by using image processing software (Mimics, U-GRAPH, and 3-Matic software) (34, 37). All of the software programs provide automatic mesh

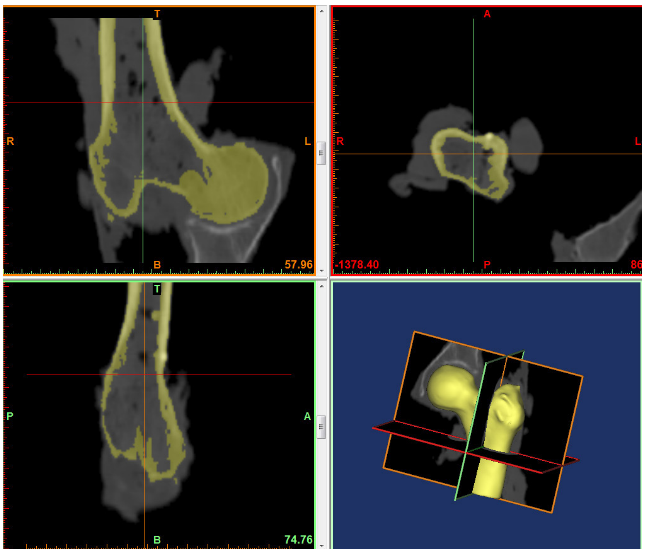

Figure 1: Bone development by using Mimics

generators which can mesh the human bone, according to the correct geometry. In addition, these packages also provide manual mesh control where the user can set up the element size and type (5, 38). However, to manually set the element size and type, a mesh convergence study should be performed first to ensure that any further FE results are not dependent on any parameter (5, 35, 39-42). The convergence study can be done either using h-refinement (for element size) as shown in Fig. 2 (5), or the r-refinement method (for element type). For implant modelling, Solidworks and Autodesk programs are usually used to design and mesh the model (11, 43, 44). The type of element used in the FE model depends on the human tissue. Most authors used triangular tetrahedral for bone (11, 19, 45) and a hexahedral element for soft tissue such as meniscus or skin (17). For FE model of implants, the triangular tetrahedral element was used for analyses (5).

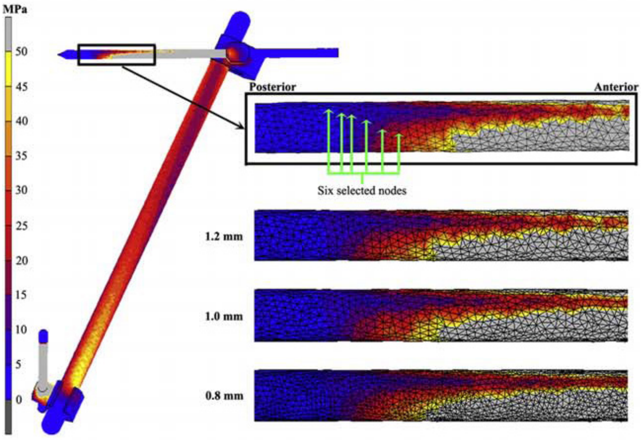

Figure 2: H-refinement for convergence study. Adopted from (5)

The modelling of ligaments is useful for FE analyses of implants at joint areas such as knees and ankles. This could mimic the real behaviour of the human body which can simulate a better distribution of loading. The mechanical behaviour of ligaments such as non-linear elastic spring (46), truss and link element was used in the previous FE model.

Assigning materials properties

The most common method used to assign material properties of bones and implants is by using Young's modulus values (11, 31, 37). Young's modulus is associated with density-elasticity relationships, and the values can be assigned in the FE software (24, 32). For orthopaedic implant, this step is a straightforward method as it is a linear isotropic and homogeneous material (47-49). However, the human bone and other tissue are neither linear isotropic nor homogeneous. Hence many scholars simplify the model as linear isotropic (5, 28, 30, 39, 43). Other tissues such as cartilage and skin were assigned with hyper-elastic behaviour (11, 30). Some of Young's modulus and hyper-elastic properties were developed by specific experiments, and others were obtained from previous literature (43). However, another method of assigning material properties is by using gray scale values from CT images (50).

Boundary and load condition

Finite element studies should use relevance boundary and load condition to mimic the real behaviour of the human body. The load is applied at a place where the pressure or reaction force is applied in the physiological condition of healthy humans. The pressure or force can be defined from muscle force, weight bearing and joint force (5, 6). These forces can be either estimated and obtained from electromyography (EMG) data or motion analysis such as inverse dynamics that is more closely illustrate the real loading conditions. To be noted, the use of boundary conditions is not similar to all finite element models, nevertheless, it should mimic a similar condition to that of activity of daily living, pathological condition or rehabilitation procedures. For example in hip model, Perez et al. (48) used two different contact forces to the hip joint with additional forces of glutei, tensor fasciae latae, iliotibial tract, vastus lateralis and medialis muscle where all of them were simulating walking and stair climbing condition. Meanwhile, Ishak et al. (51) used three forces only to simulate hip joint load that representing stair climbing activity. It is similar happened to the other regions such as tibia bone where many researchers utilized different boundary conditions to simulate their finite element models. As shown in a numerical study by Izaham et al. (8), they used an axial force of 2500 N with a distribution of 40% to the lateral compartment of tibia bone to mimic single limb stance phase of a healthy adult. Unlike a study by Mehboob et al. (10) in their simulation work where they applied 10% of body weight (BW) to the tibia bone for the first eight weeks after surgery then the load was increased up to 200% BW between eight to 12 weeks post-surgery period. Based on these previous studies, the determination of boundary conditions was based on the specific patient's condition (i.e. walking or standing phase) and the value of specific forces (i.e. 10% BW or 2500 N) to be applied to the finite element model.

Validation of the finite element model

In the computational analysis, the validation of the FE model is a vital issue and should be conducted to assess the reliability of the predictions (13, 15, 41, 52-61). It can also provide evidence as well as recommendations for the improvement and modification of the model for better mimicking the real conditions of human tissue. Many scholars used cadaveric specimens to validate their FE model (6, 52, 53). For example, Cheung et al. (52) used a foot (right side) cadaveric (Fig. 3) and loaded with the mechanical testing machine (Mini Bionix) for the application of vertical compression forces up to 700 N. Similar load conditions were applied to the FE model and comparisons were made between both. The results of vertical deformation and strain of plantar fascias showed the FE model predicted a similar profile

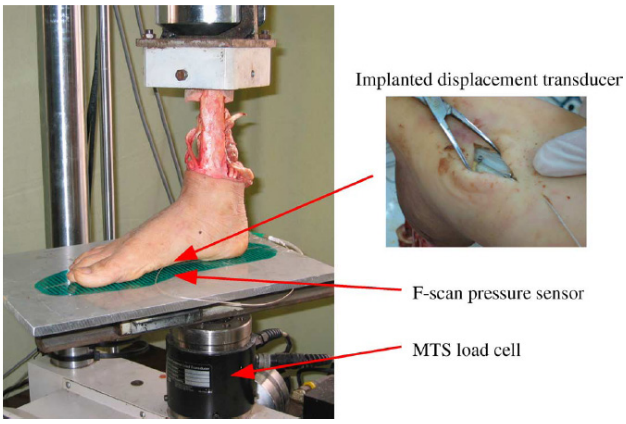

Figure 3: Cadaveric foot. Adopted from (52)

to that of cadaveric foot measurement. On top of that, Abdul Kadir et al. (6) compared micromotion values between FE model and experiment. They used four cadaveric femurs that were implanted with hip stems and placed in the universal materials testing machine (Instron Corp.). They found that distal and proximal micromotion of the hip in FE analyses is similar to the experimental measurement. Other than that, Tuncer et al. (13) used ten fresh-frozen cadaveric knees associated with different bone densities. The bones were then implanted with cementless medial Oxford UKRs and were CT scanned to obtain the geometry of the bone. FE models were developed and biomechanical evaluation was conducted to validate the computational model. Despite using human cadaveric specimen is favourable, nevertheless, the use of a commercialize synthetic model (i.e. Sawbones, Synbone or Orthobone) has received attention and appears to be an acceptable practice to validate FE models (62, 63).

Area-specific finite element model

Hip joint

Hip implants are becoming important medical devices to treat patients suffering from pathological problems (48). The most frequently reported complications associated with the use of hip stems are thigh pain, loosening (micromotion) and high stress, which can lead to implant failure (6, 23, 42, 64-68). This issue has been addressed by Abdul Kadir et al. (6, 16) through the FE model of the hip bone to evaluate interference fits on micromotion predictions. In another FE study, Dopico-Gonzalez et al. (35) compared different geometry of bone and implant design of the uncemented hip replacement. The finding shows bone variability significantly affected the sensitivities of maximum micromotion. On top of that, the femoral neck fracture of the hip is another issue that was investigated via the FE method. This hip fracture is normally treated with different size and design of implants and surgeons have been debating which configurations is better (36, 69, 70). By using FE analyses, this issue has an answer to 7.3 mm cannulated hip screw in a triangle configuration with a recommended 135° angle (51, 71).

Acetabular cup (Fig. 4) is a crucial implant for replacing defective human hip joints. Due to frequent movement after surgery, the wear rate and micromotions at the acetabular cup increases which will expose to loosening. Therefore, the use of the computational FE model to analyse the wear rate, micromotions, contact area and gap angle is preferred by many researchers (14, 45, 72, 73). A complex biomechanical analysis was conducted by Kunze et al. (72) where they simulated acetabular cup stability during the normal activity of getting up from different heights of the seat. This FE study found that low seat heights significantly prevent bone in-growth in acetabular cup implants.

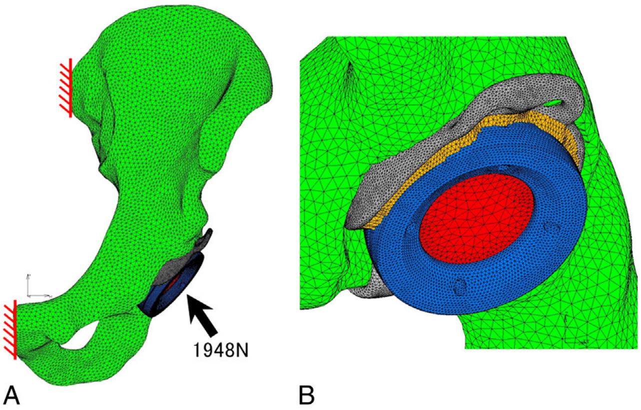

Figure 4: Acetabular cup. Adopted from (72)

Long femur

Various implants for long femur bone are receiving much attention in the current orthopaedic research (41, 74, 75). Comparative analyses between different positions and implants used for treating proximal or distal femur fractures have also been investigated via the computational FE model (41, 63, 76). For example, a complex computational modelling was conducted by Grujicic et al. (55) involving the analysis of a distal femoral fixation plate via a combination of FE method and design optimisation procedure. In this study, realistic physiological cycling loading condition and inverse dynamic analysis were carried out to optimise the design. The distal femoral fixation plate has also been compared with a nailing system to treat periprosthetic fractures (41). From the biomechanical perspective, the locking plate is better to reduce implant stresses by at least 2 times lower than nailing system (41). Samiezadeh et al. (57) used FE analyses to evaluate composite and metallic intramedullary nailing system for treating femoral shaft fractures. The finding showed that the composite nailing system provided a preferred biomechanical behaviour for tissue healing von Mises stress and strain.

Knee joint

Knee implants are a common orthopaedic medical device used by patients suffering knee defects. When this implant is inserted into the knee, long-term stability of the interface can be achieved by providing adequate micromotions. Micromotion analyses on knee implants can be conducted using the FE method (77). In addition, the total knee replacement (TKR) and total knee arthroplasty (TKA) designs should be investigated as the current design focuses on subject-specific geometry. This may interrupt the rotation and translation of the knee joint if the implant does not fit correctly (34, 47, 54, 78, 79). Some studies measured and predicted principal strain in order to distinguish the stability of various femoral stems (33, 80). Moreover, explicit dynamic FE analysis is a more efficient method to predict TKR kinematics and contact behaviour during dynamic loading conditions (12). Nevertheless, there are some studies evaluate aseptic loosening of the tibial or femur component (7, 17, 39, 44, 46, 50, 61, 81-85), where this issue can be caused by stress shielding and strain distribution on the implant (81, 86). The impact of ageing and implantation on the biomechanical environment were also examined (44).

With the latest and modern technologies in orthopaedics application, implanted cartilage replacements (ICRs) have been developed. Due to cartilage injuries, the use of ICR was biomechanical evaluated using the FE method to assess its geometry that affects loads and deformations at the defect area (9). The results found that the ICR size as well as material properties have a notable effect on the failure of the fibrin tissue. For ligaments defect in the knee joint, there were some FE studies demonstrated the use of different materials of screws for Anterior Cruciate Ligament (ACL) reconstruction (87-89). For example, Kim et al. (88) conducted a computational FE study to evaluate the different knee flexion angles when the ACL was reconstructed with a number of bundles. The results indicated that certain areas experienced high contact stress.

Long tibia

Finite element studies have also been adopted to evaluate orthopaedic implants at long tibia bone fractures (21, 49, 62, 90-94). FE analyses were conducted to investigate the micromotions, displacement and implant stress of Puddu and Tomofix plate fixation for open wedge high tibial osteotomy (8). This study found that the Tomofix plate demonstrates superior stability for bony fixation. In addition, the open tibial fractures can be treated by applying external fixator at the bone (43). Comparative analyses in treating complex distal tibia using computational model are preferred (90). The study showed the medial distal tibia locking plate could reduce fragment displacement more than an anterolateral plate. There is also a study to assess the safety condition of deformed plate fit every distal tibia bone via the FE method (95). From the results of deformation, the conclusion is that it was not achievable to manually deform the distal tibia plate to fit every bone. On top of that, the use of interlocking nail on the tibia fracture bone was evaluated through the computational model (96). The assessment of the external fixator construct for oblique tibia fractures has also been investigated using the FE model (97).

A finite element study was also conducted to evaluate the healing performance of the bone plate for tibia fractures (10). The findings showed that the biodegradable composite bone plates had better healing performance. Another study was carried out in order to biomechanically analyse the bone healing process of a fractured tibia fixated with different composite intramedullary rods using the FE method (98, 99). By comparing average callus modulus, it was demonstrated that the initial loading becomes the most sensitive factor that affects the healing performance. The effect of implant material properties has been investigated by Simon et al. (100) to assess stress distribution and micromotion at the interface of bone defect implants They found that osseointegration conditions are necessary within in a range of its stress and micromotion.

Ankle and foot

Because of the complexity of joints, several treatment strategies for several ankle and foot fractures were established such as internal fixator, external fixator and leg cast to immobilise the joint for temporary. To assess the optimum stability of external fixator (Unilateral, Mitkovic and Delta frame), FE analyses were used to evaluate the different designs and configurations available in the market (5, 11). A total of 12 bones, 4 cartilages and 25 ligaments were constructed to simulate vertical compression using Marc.Mentat software. Through this method, the displacement, micromovement and stress were investigated to indicate a more suitable and effective treatment for ankle pilon fractures. The findings showed that the Delta external fixator gave more favourable options in terms of stability compared with others.

The biomechanical analysis of total ankle replacement (TAR) is challenging when using a cadaveric specimen. Therefore, the FE method was used to evaluate TAR designs to compare the performance in terms of kinematics and contact pressure during normal daily activities (32, 101, 102). FE analysis was conducted by Elliot et al. (32) to determine the stress, wear characteristics and end point of optimisation model of TAR. The optimisation of the implant geometry gives lower stresses, thus decreasing wear rates.

Screw fixation in the ankle joint is the most popular and superior treatment by medical experts for several pathological conditions. To be noted, the study of syndesmosis injuries treating with screw fixation makes it difficult to obtain data of the stress and displacement when using cadaveric specimen. Therefore, the option of using the FE method to evaluate these injuries is favourable as it can demonstrate the changes stress distribution in ankle joint (103-105). By using FE analyses, Vazquez et al. (106) conducted a qualitative comparison of the initial stability for various screw configurations in ankle arthrodesis problem.

DIRECTIONS FOR FURTHER STUDY

Although the current literature provides valuable information and evidence, however, more research work is still needed to provide detailed prescriptive advice on the application of orthopaedic implant. Therefore, the following recommendation can assist in advancing new knowledge:

1. For modelling the FE model, it is suggested that the material properties should be similar to the real behaviour of human tissue. For example, is the use of anisotropic or inhomogeneous properties to assign the material of the cortical and cancellous bone as well as viscous-elasticity material to the ligament at joint (22, 27, 29). These methods, therefore, can predict more realistic outcomes compare to simple assumptions.

2. Due to the difficulty of assessing and securing cadaveric specimen in certain countries, therefore, the synthetic bone model (i.e. Sawbones, Synbone or Orthobone) or developed physical polyurethane model can be used for the validation process in FE model. It should be noted that the use of a synthetic bone have received attention and appears to be an acceptable practice to validate FE predictions (62, 63).

LIMITATIONS OF THIS REVIEW

In this review, we have attempted to highlight the use of finite element method in the biomechanical evaluation of orthopaedic implants. We believe this concise article can provide informative literature not only for engineers, but also to doctors and researchers. Nevertheless, there are limitations considered in this review. The main limitation is that we only reviewed FE analysis of implants for the lower limbs of the human body. Secondly, the review focused on current strategies to develop the computational model. Topics related to the FE method outside of those described earlier were not included in this review.

CONCLUSION

This review article has come out with a conclusion where the development process of finite element model from previous reported literature was based on the specific condition of data (CT or MRI), material properties (homogeneous or inhomogeneous), boundary conditions (walking or standing) and pathological conditions.

ACKNOWLEDGEMENT

The work has been carried out with the support of research funding from Universiti Teknologi Malaysia (UTM) under Tier 1 grant (Grant no: Q.J130000.2545.20H20 and Q.J130000.2545.20H26), Tier 2 (Grant no.: R.J130000.2651.15J84) and Fundamental Research Grant Scheme by Government of Malaysia (Grant no.: R.J130000.7851.5F135).

REFERENCES

1. Ansah M, Sella EJ. Treatment of complete open medial subtalar dislocation with an external fixator: a case report. Foot and Ankle Surgery. 2000;6:179-84.
2. Milenkovic S, Mitkovic M, Bumbasirevic M. External fixation of open subtalar dislocation. Injury. 2006;37:909-13.
3. 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;36:1480-7.
4. Cody DD, Hou FJ, Divine GW, Fyhrie DP. Short term in vivo precision of proximal femoral finite element modeling. Annals of Biomedical Engineering. 2000;28:408-14.
5. 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;36:1358-66.
6. 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;41:587-94.
7. Perillo-Marcone A, Ryd L, Johnsson K, Taylor M. A combined RSA and FE study of the implanted proximal tibia: correlation of the post-operative mechanical environment with implant migration. Journal of Biomechanics. 2004;37:1205-13.
8. Izaham RMAR, Abdul Kadir MR, Rashid AHA, Hossain MG, Kamarul T. Finite element analysis of Puddu and Tomofix plate fixation for open wedge high tibial osteotomy. Injury. 2012;43:898-902.
9. Vahdati A, Wagner DR. Implant size and mechanical properties influence the failure of the adhesive bond between cartilage implants and native tissue in a finite element analysis. Journal of Biomechanics. 2013;46:1554-60.
10. 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.
11. 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;36:1322-30.
12. Halloran JP, Petrella AJ, Rullkoetter PJ. Explicit finite element modeling of total knee replacement

mechanics. Journal of Biomechanics. 2005;38:323-31.

13. Tuncer M, Cobb JP, Hansen UN, Amis AA. Validation of multiple subject-specific finite element models of unicompartmental knee replacement. Medical Engineering & Physics. 2013;35:1457-64.
14. 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;28:1765-9.
15. 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;47:742-5.
16. Abdul Kadir MR. Computational biomechanics of the hip joint. Berlin: Springer; 2014.
17. Hopkins AR, New AM, Rodriguez-y-Baena F, Taylor M. Finite element analysis of unicompartmental knee arthroplasty. Medical Engineering & Physics. 2010;32:14-21.
18. Ramlee MH, Abdul Kadir MR, Harun H. Three-dimensional modelling and finite element analysis of an ankle external fixator. Advanced Materials Research. 2014;845:183-8.
19. 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. 2015;44(1):213-21.
20. Ramlee MH, Abdul Kadir MR, Harun H. Three-dimensional modeling and analysis of a human ankle joint. IEEE. 2013:74-8.
21. 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;48:1119-24.
22. Abdullah AH, Todo MN, Y. Stress and damage formation analysis in hip arthroplasties using CT-based finite element method. Journal of Engineering and Applied Sciences. 2017;10(12):2715-9.
23. 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;22:422-30.
24. van den Munckhof S, 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(3).
25. Qiu TX, Teo EC, Yan YB, Lei W. Finite element modeling of a 3D coupled foot-boot model. Medical Engineering & Physics. 2011;33:1228-33.
26. 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. Annals of Biomedical Engineering. 2015;44(1):3-15.
27. Izmin NAN, Todo M, Abdullah AH. Prediction of bone damage formation in resurfacing hip arthroplasty. International Journal of Engineering and Advanced Technology. 2019;9(1):5879-85.
28. Yosibash Z, Trabelsi N, Milgrom C. Realiable simulations of the human proximal femur by high-order finite element analysis validated by the experimental observations. Journal of Biomechanics. 2007;40:3688-99.
29. 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.
30. Wong DW-C, 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.
31. Dai XQ, Li Y, Zhang M, Cheung JT. Effect of sock on biomechanical responses of foot during walking. Clinical Biomechanics. 2006;21:314-21.
32. Elliot BJ, 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.
33. Completo A, Rego A, Fonseca F, Ramos A, Relvas C, Simoes 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;25:159-65.
34. 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;17:521-5.
35. Dopico-Gonzalez 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;43:512-20.
36. Wang CJ, Yettram AL, Yao MS, Procter P. Finite element analysis of a Gamma nail within a fractured femur. Medical Engineering & Physics. 1998;20:677-83.
37. Liu X, Zhang M. Redistribution of knee stress using laterally wedged insole intervention: Finite element analysis of knee-ankle-foot complex. Clinical Biomechanics. 2013;28:61-7.
38. 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;20:1-10.
39. 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.

40. Holzer A, Schroder 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;110:82-8.
41. Chen S-H, Chiang M-C, Hung C-H, Lin S-C, Chang H-W. Finite element comparison of retrograde intramedullary nailing and locking plate fixation with/without an intramedullary allograft for distal femur fracture following total knee arthroplasty. Knee. 2014;21(1):224-31.
42. Bah MT, Nair PB, Browne M. Mesh morphing for finite element analysis of implant positioning in cementless totap hip replacements. Medical Engineering & Physics. 2009;31:1235-43.
43. 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;44:1097-101.
44. 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;38:959-68.
45. 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. Journal of Orthopaedic Science. 2013;18:264-70.
46. Checa S, Taylor M, New A. Influence of an interpositional spacer on the behaviour of the tibiofemoral joint: A finite element study. Clinical Biomechanics. 2008;23:1044-52.
47. 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;46:1351-7.
48. Perez MA, Palacios J. Comparative finite element analysis of the debonding process in different concepts of cemented hip implants. Annals of Biomedical Engineering. 2010;36(6):2093-106.
49. 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;2:304-9.
50. Sawatari T, Tsumura H, Iesaka K, Furushiro Y, Torisu T. Three-dimensional finite element analysis of unicompartmental knee arthroplasty- the influence of tibial component inclication. Journal of Orthopaedic Research. 2005;23:549-54.
51. Ishak MF, Abdul Kadir MR, Rahman SA. The effect of cannulated hip screw size in the treatment of femoral neck fracture. Jurnal Teknologi. 2015;77(7):13-7.
52. Cheung JT-M, Zhang M, An K-N. Effect of Achilles tendon loading on plantar fascia tension in the standing foot. Clinical Biomechanics. 2006;21:194-203.
53. 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;39:2457-67.
54. Baldwin MA, Clary CW, Fitzpatrick CK, Deacy JS, Maletsky LP, Rullkoetter PJ. Dynamic finite element knee simulation for evaluation of knee replacement mechanics. Journal of Biomechanics. 2012;45:474-83.
55. Grujicic M, Arakere G, Xie X, LaBerge M, Grujicic A, Wagner DW, et al. Design-optimization and material selection for a femoral-fracture fixation-plate implant. Materials and Design. 2010;31:3463-73.
56. Completo A, Fonseca F, Simoes JA. Experimental validation of intact and implanted distal femur finite element models. Journal of Biomechanics. 2007;40:2467-76.
57. Samiezadeh S, Avval PT, Fawaz Z, Bougherara H. Biomechanical assessment of composite versus metallic intramedullary nailing system in femoral shalf fractures: A finite element study. Clinical Biomechanics. 2014;29:803-10.
58. Scholz R, Hoffmann F, von Sachsen S, Drossel W-G, Klohn C, Voigt C. Validation of density-elasticity relationships for finite element modeling of human pelvic bone by modal analysis. Journal of Biomechanics. 2013;46:2667-73.
59. 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):031016.
60. Gray HA, Zavatsky AB, Cristofolini L, Gill HS. A finite element model of a human tibia with an oxford knee replacement validated by experiments. Journal of Biomechanics. 2007;40:S63.
61. Siggelkow E, Kersh M, Muenchinger M. Femoral component design and positioning during implatation affects ACL tension- a finite element study. Journal of Biomechanics. 2007;40:S98.
62. Chen Y-N, Lee P-Y, Chang C-H, Chang C-W, Ho Y-H, Li C-T, et al. Computational comparison of tibial diaphuseal fractures fixed with various degress of prebending of titanium elastic nails and with and without end caps. Injury. 2016.
63. 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.
64. Colic K, Sedmak A, Grbovic A, Tatic U, Sedmak S, Djordjevic B. Finite element modeling of hip implant static loading. Procedia Engineering. 2016;149:257-62.
65. Kang YK, Park HC, Youm Y, Lee IK, Ahn MH, Ihn

JC. Three dimensional shape reconstruction and finite element analysis of femur before and after the cementless type of total hip replacement. Journal of Biomechanical Engineering. 1993;15:497-504.

66. Hirata Y, Inaba Y, Kobayashi N, Ike H, Fujimaki H, Saito T. Comparison of mechanical stress and change in bone mineral density between two types of femoral implant using finite element analysis. The journal of Arthroplasty. 2013;28:1731-5.
67. Pettersen S, Wik TS, Skallerud B. Subject specific finite element analysis of implant stability for a cementless femoral stem. Clinical Biomechanics. 2009;24:480-7.
68. Reimeringer M, Nuno N. The influence of contact ratio and its location on the primary stability of cementless total hip arthroplasty: A finite element analysis. Journal of Biomechanics. 2016;49(7):1064-70.
69. Chen W-P, Tai C-L, Shih C-H, Hsieh P-H, Leou M-C, Lee MS. Selection of fixation devices in proximal femur rotational osteotomy: clinical complications and finite element analysis. Clinical Biomechanics. 2004;19:255-62.
70. Helwig P, Faust G, Hindenlang U, Hirschmuller A, Konstantinidis L, Bahrs C, et al. Finite element analysis of four different implants inserted in different positions to stabilize an idealized trochanteric femoral fracture. Injury. 2009;40:288-95.
71. Ishak MF, Abdul Kadir MR, Harun H, Rahman SA. The effect of cannulated screw placement angle in the management of femoral neck fracture. Advanced Materials Research. 2014;845:216-20.
72. Kunze M, Schaller A, Steinke H, Scholz R, Voigt C. Combined multi-body and finite element investigation of the effect of the sear height on acetabular implant stability during the activity of getting up. Computer Methods and Programs in Biomedicine. 2012;105:175-82.
73. Spears IR, Morlock MM, Pfleiderer M, Schneider E, Hille E. The influence of friction and interference on the seating of a hemispherical press-fit cup: a finite element investigation. Journal of Biomechanics. 1999;32:1183-9.
74. Gong H, Kong L, Zhang R, Fang J, Zhao M. A femur-implant model for the prediction of bone remodeling behavior induced by cementless stem. Journal of Bionic Engineering. 2013;10:350-8.
75. Berney MJ, Gibbons J, Fitzgerald MK, Cardiff DP, Leonard M. Computational modelling of forces acting on the femur in acetabular fractures: A finite element analysis study. Journal of Orthopaedic. 2019;16(6):603-18.
76. Sim E, Freimuller W, Reiter TJ. Finite element analysis of the stress distributions in the proximal end of the femur after stabilization of a pertrochanteric model fracture: a comparison of two implants. Injury. 1995;26(7):445-9.
77. Berahmani S, Janssen D, Wolfson D, de Waal Malefijt M, Fitzpatrick CK, Rullkoetter PJ, et al. FE analysis of the effects of simplifications in experimental testing on micromotions of uncemented femoral knee implants. Journal of Orthopaedic Research. 2016;34(5):812-9.
78. Arsene CTC, Gabrys B. Probabilistic finite element predictions of the human lower limb model in total knee replacement. Medical Engineering & Physics. 2013;35(8):1116-32.
79. Amirouche F, Choi KW, Goldstein WM, Gonzalez MH, Broviak S. Finite element analysis of resurfacing depth and obliquity on patella stress and stability in TKA. The journal of Arthroplasty. 2013;28:978-84.
80. Completo A, Duarte R, Fonseca F, Simoes JA, Ramos A, Relvas C. Biomechanical evaluation of different reconstructive techniques of proximal tibia in revision total knee arthroplasty: An in-vitro and finite element analysis. Clinical Biomechanics. 2013;28:291-8.
81. Cawley DT, Kelly N, Simpkin A, Shannon FJ, McGarry JP. Full and surface tibial cementation in total knee arthroplasty: A biomechanical investigation of stress distribution and remodeling in the tibia. Clinical Biomechanics. 2012;27:390-7.
82. Taylor M, Tanner KE, Freeman MAR. Finite element analysis of the implanted proximal tibia: a relationship between the initial cancellous bone stresses and implant migration. Journal of Biomechanics. 1998;31:303-10.
83. Galloway F, Kahnt M, Ramm H, Worsley P, Zachow S, Nair P, et al. A large scale finite element study of a cementless osseointegrated tibial tray. Journal of Biomechanics. 2013;46:1900-6.
84. Rakotomanana RL, Leyvraz PF, Curnier A, Heegaard JH, Rubin PJ. A finite element model for evaluation of tibial prosthesis-bone interface in total knee replacement. Journal of Biomechanics. 1992;25(12):1413-24.
85. Hashemi A, Shirazi-Adl A. Finite element analysis of tibial implants- Effect of fixation design and friction model. COmputer Methods in Biomechanics and Biomedical Engineering. 2007;3(3):183-201.
86. Enab TA, Bondok NE. Material selection in the design of the tibia tray component of cemented artificial knee using finite element method. Materials and Design. 2013;44:454-60.
87. Abdullah AH, Rashid H, Mahmud J, Othman MF, Ibrahim MWA. Effects of screw materials in anterior cruciate ligament reconstruction using finite element analysis. Procedia Engineering. 2012;41:1614-9.
88. Kim HY, Seo YJ, Kim HJ, Nguyenn T, Shetty NS, Yoo YS. Tension changes within the bundles of anatomic double-bundle anterior cruciate ligament reconstruction at different knee flexion angles: a study using a 3-dimensional finite element model. Arthroscopy. 2011;27(10):1400-8.

89. Au AG, Raso VJ, Liggins AB, Otto DD, Amirfazli A. A three-dimensional finite element stress analysis for tunnel placement and buttons in anterior cruciate ligament reconstructions. Journal of Biomechanics. 2005;38:827-32.
90. Aizat RM, Abdul Kadir MR, Rahman SA, Shihabudin TMTM, Robson N, Kamarul T. Biomechanical comparative analyses between the anterolateral and medial distal tibia locking plates in treating complex distal tibial fracture: A finite element study. Journal of Medical Imaging and Health Informatics. 2013;3:532-7.
91. Dahmen T, Roland M, Tjardes T, Bouillon B, Slusallek P, Diebels S. An automated workflow for the biomechanical simulation of a tibia with implant using computed tomography and the finite element method. Computers & Mathematics with Applications. 2015;70(5):903-16.
92. Kim S-H, Chang S-H, Son D-S. Finite element analysis of the effect of bending stiffness and contact condition of composite bone plates with simple rectangular cross-section in the bio-mechanical behaviour of fractured long bones. Composites: Part B. 2011;42:1731-8.
93. Fouad H. Assessment of function-graded materials as fracture fixation bone-plates under combined loading conditions using finite element modelling. Medical Engineering & Physics. 2011;33:456-63.
94. Completo A, Simoes JA, Fonseca F. A finite element analysis of long stemmed distal tip fixation conditions in tibial knee replacement. Journal of Biomechanics. 2006;39:S517-S8.
95. Harith H, Schmutz B, Malekani J, Schuetz MA, Yarlagadda PK. Can we safely deform a plate to fit every bone? Population-based fit assessment and finite element deformation of a distal tibial plate. Medical Engineering & Physics. 2016;38(3):280-5.
96. Raunest J, Kynast W, Lesch V, Kukulies U, Hacklander T, Schwarting KH, et al. Geometric properties of the fractured tibia stabilized by unreamed interlocking nail: Development of a three-dimensional finite element model. Computers and Biomedical Research. 1996;29:259-70.
97. Cirovic S, Nutma A. Computer model of the external fixation of oblique tibia fractures. Journal of Biomechanics. 2008;41:S449.
98. Son D-S, Mehboob H, Jung H-O, Chang S-H. The finite element analysis for endochondral ossification process of a fractured tibia applied with a composite IM-rod based on a mechano-regulation theory using a deviatoric strain. Composites: Part B. 2014;56:189-96.
99. Mehboob H, Son D-S, Chang S-H. Finite element analysis of tissue differentiation process of a tibia with various fracture configurations when a composite itramedullary rod was applied. Composite Science and Technology. 2013;80:55-65.
100. Simon U, Augat P, Ignatius A, Claes L. Influence of the stiffness of bone defect implants on the mechanical conditions at the interface- a finite element analysis with contact. Journal of Biomechanics. 2003;36:1079-86.
101. Reggiani B, Leardini A, Corazza F, Taylor M. Finite element analysis of a total ankle replacement during the stance phase of gait. Journal of Biomechanics. 2006;39:1435-43.
102. Vo H, Tuvel B, Nguyen B, Le T. Determination of the neutral axis in total ankle replacement. IFMBE Proceedings 2015;46:121-4.
103. Liu Q, Zhang K, Zhuang Y, Li Z, Yu B, Pei G. Analysis of the stress and displacement distribution of inferior tibiofibular syndesmosis injuries repaired with screw fixation: A finite element study. Plos One. 2013;8(12).
104. Anderson RT, Pacaccio DJ, Yakacki CM, Carpenter RD. Finite element analysis of a pseudoelastic compression-generating intramedullary ankle arthrodesis nail. Journal of the Mechanical Behavior of Biomedical Materials. 2016;62:83-92.
105. Jiang D, Zhan S, Wang Q, Ling M, Hu H, Jia W. Biomechanical comparison of locking plate and cancellous screw techniques in medial malleolar fractures: a finite element analysis. The Journal of Foot and Ankle Surgery. 2019;58:1138-44.
106. Vazquez AA, Lauge-Pedersen H, Lidgren L, Taylor M. Finite element analysis of the initial stability of ankle arthrodesis with internal fixation: flat cut versus intact joint contours. Clinical Biomechanics. 2003;18:244-53.