FINITE ELEMENT ANALYSIS OF BONE BIOMECHANICS – A REVIEW

Ragini. B

Assistant Professor, Department of Biomedical Engineering, Karpaga Vinayaga College of Engineering and Technology, Kanchipuram, Tamilnadu, India.

ABSTRACT

In the recent times, the research on finite element modelling has emerged as an effective tool for modelling and simulation of the biomedical engineering system. Finite element modelling (FEM) is a computational technique which can be used to solve the biomedical engineering problems based on the theories of continuum mechanics. This paper presents the state of art review on finite element analysis (FEA) of bone biomechanics. This analysis can be used to assess the bone's stress and strain distribution, implant designs, fracture load prediction and also to determine the mechanical properties of the bone. The aim of this review is to provide a comprehensive detail about the development in the area of application of FEA in bone biomechanics during the last decades. It will help the researchers and the clinicians alike for the better treatment of patients and future development of personalized fixation designs.

Keywords: FEA, FEM, Bone, Biomechanics, stress and strain

1. INTRODUCTION

Finite Element Analysis of bone was begun as early as 1970s and its applications in biomechanics is constantly increasing every decade [1]. It was a promising tool in orthopedics and other clinical fields during the 1980s. Finite Element Modelling (FEM) is the most widely preferred computational technique used in the biomechanics for the mechanical analysis of hard tissue models. It has intensive technological developments via computer aided design, creation of three-dimensional models from imaging data, simulation of material properties [2]. It facilitates the picturization of superimposed structures, and the provision of the material properties of anatomic craniofacial structures [3]. It also allows to establish the location, magnitude, and direction of an applied force, and also assigns stress points that can be theoretically measured [4]. The finite element method (FEM) is basically a numerical method of analyzing stresses and deformations in the structures of any given geometry. The structure is discretized into the so called 'finite elements' connected through nodes. The type, arrangement and total number of elements impact the accuracy of the results [5]. FEM can solve unknowns of the model with complex constitutive laws of material behaviors (e.g., articular disk, ligament etc.), and various load and boundary conditions (e.g., external force, thermal change and magnetic field power by a series of computational procedures). In this process, the model in the global domain is divided into small and finite domains (elements) with differential equations [6]. The iterative incremental solution is done by solving the problem. If the model is not well-conditioned, the convergence will be slower. Mechanical analysis of biomechanical models with nonlinear load, material, geometry and contact properties can be carried out by FEA. Loading nonlinearity describes time-dependent loading that is causing large displacement. Material nonlinearity comes from plasticity, viscoplasticity properties of materials [2].

FEA has also proven to be the best method to study the stress patterns in implant components and on bone. It also helps in studying the biomechanical behavior of the implants and the effect of different implant materials to sustain the load. It helps in predicting the success rate of implants in clinical conditions [7]. It can also aid in customized implant design as per individual patient requirements. The biomechanics of fracture of bone can be simulated and studied easily using FEA. It is advantageous over real models in the way that the experiments can be repeated, no ethical considerations, no harm to patients, and the study requirements can be modified as and when required.

2. GENERATING 3D MODELS

The first requirement is the slices of anatomical structure on which analysis has to be performed. This can be obtained from digital imaging systems like Computer Tomography (CT) and Magnetic Resonance Imaging (MRI). CT and MRI help to image the bone at micro-level and help in obtaining reliable results due to the very accurate anatomic model. Thus, a 3D model depicting the biological structures can be obtained from these digital image slices [8]. The accuracy of analysis can be improved by constructing detailed geometry of the model, defining proper loading and boundary conditions, establishing true contact and joint points, as well as providing a well-structured mesh. The accuracy of the model determines the reliability of the FEA results. The mechanical properties of the internal structures are differentiated because different biological structures like bone, tissue, organ, ligaments absorb radiation at different levels and this is distinctly visible in CT as it is calculated in terms of Hounsfield numbers (HU) [9-10]. Thus, different anatomically significant areas can be converted into 3D volumetric models separately and subsequently a more detailed 3D model of the structure of study with specific material properties can be generated. This can be carried out through various Computer Aided Design (CAD) softwares like CATIA, MIMICS, SolidWorks, Pro/E, etc. Based on the HU values different structures like cortical bone, trabecular bone, muscles, tissues and fracture site can be separated from each other through thresholding. It is also possible to extract the solid body models of biological structures from MRI data that be directly processed by 3D reconstruction software. It is especially useful to separate soft tissues from whole scanned tissues of the body. The drawbacks of MRI is that images should be on the same plane and the extraction of the indented tissue from other tissues is a time-consuming process [11-13].

3. ANALYSIS OF MATERIAL PROPERTIES USING FEA

Material assignment in FEA is an important phase of the analysis. Isotropic materials have the mechanical and thermal properties that are the same in all directions. In contrast, the anisotropic materials show different mechanical and thermal characteristics along different directions under the same loading conditions. The orthotropic materials, which are a subgroup of the anisotropic materials, have the different material properties along three mutually-orthogonal axes [14].

In FEA software, specifying the elastic modulus (Young’s modulus) and Poisson’s ratio would be enough for identification of the material. The elastic modulus of bones can be predicted from CT data. The material properties can be characterized by HU values. These coefficients are related to tissue density [15]. Density values can be converted to elastic modulus values by means of different approaches by relating the HU value, elastic modulus and apparent bone density to each other [16]. HU values of the bone can be measured using certain software and the Young’s modulus can be estimated by some calculations [17]. This method of estimation of elastic modulus from HU values enables the researchers to apply different mechanical properties to the bone under study.

Materials used for the implants are selected to balance strength requirements and must be biocompatible. Commonly used implant materials include metals like Titanium alloys, stainless steel and cobalt-chromium alloys, Inert ceramic alloys like alumina, carbon and zirconia, bioactive ceramics like bioactive glasses and calcium phosphates and polymers like ultra high molecular weight polyethylene (UHMWPE) and PMMA. Thus, FEA can be used to extract accurate numerical results for different material properties of biological structures on realistic modelling.

4. STRESS AND STRAIN ANALYSIS USING FEM

To understand the fracture risk, implant failure and load bearing capabilities, stress and strain analysis of bones using FEM is highly essential. A study on stress analysis in human femur bone revealed that irregular stress distribution at the joints of the bone is likely to experience fracture in case of impact falling [18]. In another study, a computational approach based on cohesive finite element modeling was employed to evaluate the effect of strain rate on fracture toughness of human cortical bone. The results suggested that strain rates associated with falls lead to a dramatic reduction in bone’s resistance against crack propagation. The compromised fracture resistance of bone at loads exceeding normal activities indicates a sharp reduction and/or absence of toughening mechanisms in bone during high strain conditions associated with traumatic fracture [19]. Chu et al investigated the stress distribution of the femoral and rotating platform bearing components of New Jersey Contact Stress Knee, where the femoral component was made up of a titanium alloy and the bearing was made up of UHMWPE [20]. The author observed that the contact area decreased as the flexion angle increased, which led to the increase in compressive stress [21]. Another author analyzed the cartilage stresses in the hip joint and assumed the material of the bone to be rigid. In many papers the compact and spongy bone are modelled as isotropic and homogenous but in real conditions both type of bones have different material properties. It has been observed that the strain rate affects the human bone toughness [22].

5. IMPLANT DESIGN ANALYSIS USING FEM

There are two types of fracture fixation used to fix the fracture in bone such as external skeletal fixation (POP, clamp fixators, and ring fixators) and internal fixation of fractures (plate and screw, and intramedullary nail) [23]. Arbag et al, evaluated different miniplates for internal fixation of mandibular fractures using FEA. Simulated corpus fractures were fixed with 14 different fixation configurations of titanium miniplates and the FEA was performed with respect to displacement and stresses for these configurations [24]. In another study, analysis of femur bone fracture fixationplate was done using FEM and the results suggested that stresses at the bone plate decreases significantly when using the titanium material instead of stainless steel and cobalt chrome. This is because of flexibility of titanium plate compared to other plates. Additionally, the result indicates that the stresses at the fracture site increases when taking torsional load into consideration [25]. Kharazi et al used FEM to model and analyze a partially resorbable composite bone plate consisting of a poly L-Lactic acid matrix and textile bioglass fibers used as reinforcement. It was concluded that the composite plate system is suitable for forearm region and it was capable of reducing stress shielding effects at the fracture site [26].

6. FRACTURE LOAD ANALYSIS USING FEA

When the load in a particular region of a bone exceeds the ultimate strength of bone, then fracture occurs. Fracture means the continuity of bone being disrupted [23]. A study used FEA-based vertebral strength assessment and vertebral trabecular BMD for prediction of incident vertebral fractures in women [27]. And they found out that women and men in that study were at an equivalently high probability for fracture and that FEA can be used clinically to identify women and men at high risk of fracture. Another study developed a realistic model of the human clavicle that would respond just like a real clavicle would in an accident, independent of load direction. Three clavicle computer tomography (CT) scans were modeled in finite element analysis software. These models had dynamic loads applied to them and the results of the finite element analysis was compared to that of results from actual car crash data of individuals of similar age. This model was able to react to any load direction accurately rather than only a frontal or side load [28]. Lotz et al studied the FEA of proximal femur in fracture load prediction and found out that the calculated von misses strain proved to be the best indicator of bone yield and failure, predicting bone failure within 8 percent of the experimental fracture loads [29]. Miura et al [30], validated the proximal femur CT based specimen-specific FEA model with smaller mesh size using fresh frozen cadavers. The fracture load was calculated from force-displacement curve and it was found that there was a good correlation between fracture loads of mechanical testing and FEA prediction.

7. CONCLUSION

In the recent past, the finite element modelling has been developed as an effective tool for bone biomechanics with some limitations. It is a computerized study in which clinical conditions may not be entirely replicated. The lack of anatomical detail in the modelling phase and the lack of information about the material properties of bone and bone structure impede its accuracy. With the recent advances in computer tomography some of these limitations have been overcome. In the future, a compact system, which has the ability to make all the processes in the same unit, could be designed and correspondingly operating systems would be improved. When specific FE models of subjects from CT images can be analyzed in a single system, it would consume less time and so a unique optimum customized implant can be practically designed and produced. Modeling and material of the implant could be determined for each bone and each fracturing type, individually. With regards to these limitations, further FEA research should be supplemented with clinical evaluation.

REFERENCES

1. Zysset, Philippe K., Enrico Dall'Ara, Peter Varga, and Dieter H. Pahr. "Finite element analysis for prediction of bone strength." *BoneKEy reports* 2 (2013).
2. Dogru, Suzan Cansel, Erol Cansiz, and Yunus Ziya Arslan. "A review of finite element applications in oral and maxillofacial biomechanics." *Journal of mechanics in medicine and biology* 18, no. 02 (2018): 1830002.
3. Sun, J., T. Jiao, Y. Tie, and D. Wang. "Three-dimensional finite element analysis of the application of attachment for obturator framework in unilateral maxillary defect." *Journal of oral rehabilitation* 35, no. 9 (2008): 695-699.
4. Gao, Jianxin, Wei Xu, and Zuquan Ding. "3D finite element mesh generation of complicated tooth model based on CT slices." *Computer methods and programs in biomedicine* 82, no. 2 (2006): 97-105.
5. Van Staden, R. C., Hong Guan, and Yew-Chaye Loo. "Application of the finite element method in dental implant research." *Computer methods in biomechanics and biomedical engineering* 9, no. 4 (2006): 257-270.
6. Geng, Jian-Ping, Keson BC Tan, and Gui-Rong Liu. "Application of finite element analysis in implant dentistry: a review of the literature." *The Journal of prosthetic dentistry* 85, no. 6 (2001): 585-598.
7. Trivedi, Shilpa. "Finite element analysis: A boon to dentistry." *Journal of oral biology and craniofacial research* 4, no. 3 (2014): 200-203.
8. Liang, Rui, Weihua Guo, Xiangchen Qiao, Hailin Wen, Mei Yu, Wei Tang, Lei Liu, Yongtao Wei, and Weidong Tian. "Biomechanical analysis and comparison of 12 dental implant systems using 3D finite element study." *Computer Methods in Biomechanics and Biomedical Engineering* 18, no. 12 (2015): 1340-1348.
9. Helgason, Benedikt, Egon Perilli, Enrico Schileo, Fulvia Taddei, Sigurður Brynjolfsson, and Marco Viceconti. "Mathematical relationships between bone density and mechanical properties: a literature review." *Clinical biomechanics* 23, no. 2 (2008): 135-146.
10. Morgan, Elise F., Harun H. Bayraktar, and Tony M. Keaveny. "Trabecular bone modulus–density relationships depend on anatomic site." *Journal of biomechanics* 36, no. 7 (2003): 897-904.
11. Benazzi, Stefano, Ian R. Grosse, Giorgio Gruppioni, Gerhard W. Weber, and Ottmar Kullmer. "Comparison of occlusal loading conditions in a lower second premolar using three-dimensional finite element analysis." *Clinical oral investigations* 18, no. 2 (2014): 369-375.
12. Demenko, V., I. Linetskiy, K. Nesvit, H. Hubalkova, V. Nesvit, and A. Shevchenko. "Importance of diameter-to-length ratio in selecting dental implants: a methodological finite element study." *Computer methods in biomechanics and biomedical engineering* 17, no. 4 (2014): 443-449.
13. Jaisson, Maxime, Philippe Lestriez, Redha Taiar, and Karl Debray. "Finite element modelling of the articular disc behaviour of the temporo-mandibular joint under dynamic loads." *Acta of Bioengineering & Biomechanics* 13, no. 4 (2011).
14. Boccaccio, Antonio, Luciano Lamberti, and Valentina Quarta. "Numerical and experimental evaluation of stresses in a human mandible." In *2005 SEM Annual Conference & Exposition on Experimental and Applied Mechanics*. 2005.
15. Wang, Hang, Baohui Ji, Wentao Jiang, Lei Liu, Ping Zhang, Wei Tang, Weidong Tian, and Yubo Fan. "Three-dimensional finite element analysis of mechanical stress in symphyseal fractured human mandible reduced with miniplates during mastication." *Journal of oral and maxillofacial surgery* 68, no. 7 (2010): 1585-1592.
16. Cansiz, Eroi, Suzan Cansel Doğru, and Yunus Ziya Arslan. "Mechanical evaluation of different fixation materials used for mandibular condyle fractures: Finite element analysis." In *2015 19th National Biomedical Engineering Meeting (BIYOMUT)*, pp. 1-6. IEEE, 2015.

17. Yoda, N., Zhipeng Liao, J. Chen, K. Sasaki, M. Swain, and Q. Li. "Role of implant configurations supporting three-unit fixed partial denture on mandibular bone response: biological-data-based finite element study." *Journal of oral rehabilitation* 43, no. 9 (2016): 692-701.
18. Basu, P. K., A. G. Beall, D. J. Simmons, and M. Vannier. "3-D femoral stress analysis using CT scans and p-version FEM." *Biomaterials, medical devices, and artificial organs* 13, no. 3-4 (1985): 163-186.
19. Ural, Ani, Peter Zioupos, Drew Buchanan, and Deepak Vashishth. "The effect of strain rate on fracture toughness of human cortical bone: a finite element study." *Journal of the mechanical behavior of biomedical materials* 4, no. 7 (2011): 1021-1032.
20. Chu, Taiming. "An investigation on contact stresses of New Jersey Low Contact Stress (NJLCS) knee using finite element method." *Journal of systems integration* 9, no. 2 (1999): 187-199.
21. Carr, Brandi C., and Tarun Goswami. "Knee implants–Review of models and biomechanics." *Materials & Design* 30, no. 2 (2009): 398-413.
22. Anderson, Andrew E., Benjamin J. Ellis, Steve A. Maas, and Jeffrey A. Weiss. "Effects of idealized joint geometry on finite element predictions of cartilage contact stresses in the hip." *Journal of biomechanics* 43, no. 7 (2010): 1351-1357.
23. Parashar, Sandeep Kumar, and Jai Kumar Sharma. "A review on application of finite element modelling in bone biomechanics." *Perspectives in Science* 8 (2016): 696-698.
24. Arbag, Hamdi, Hasan Hüsnü Korkmaz, Kayhan Ozturk, and Yavuz Uyar. "Comparative evaluation of different miniplates for internal fixation of mandible fractures using finite element analysis." *Journal of oral and maxillofacial surgery* 66, no. 6 (2008): 1225-1232.
25. Das, Sandeep, and Saroj Kumar Sarangi. "Finite element analysis of femur fracture fixation plates." *International Journal of Basic and Applied Biology* 1, no. 1 (2014): 1-5.
26. Kharazi, A. Zargar, M. H. Fathi, and F. Bahmany. "Design of a textile composite bone plate using 3D-finite element method." *Materials & Design* 31, no. 3 (2010): 1468-1474.
27. Kopperdahl, David L., Thor Aspelund, Paul F. Hoffmann, Sigurdur Sigurdsson, Kristin Siggeirsdottir, Tamara B. Harris, Vilmundur Gudnason, and Tony M. Keaveny. "Assessment of incident spine and hip fractures in women and men using finite element analysis of CT scans." *Journal of Bone and Mineral Research* 29, no. 3 (2014): 570-580.
28. Jones, Travis. "Finite Element Analysis of Human Clavicle Bone." PhD diss., The Ohio State University, 2012.
29. Lotz, J. C., E. J. Cheal, and Wilson C. Hayes. "Fracture prediction for the proximal femur using finite element models: part I—linear analysis." (1991): 353-360.
30. Miura, Michiaki, Junichi Nakamura, Yusuke Matsuura, Yasushi Wako, Takane Suzuki, Shigeo Hagiwara, Sumihisa Orita et al. "Prediction of fracture load and stiffness of the proximal femur by CT-based specimen specific finite element analysis: cadaveric validation study." *BMC musculoskeletal disorders* 18, no. 1 (2017): 1-8.