Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Development and Verification of Three-Dimensional Model of Femoral Bone: Finite Element Analysis

To cite this article: Aishah Umairah Abd Aziz *et al* 2019 *J. Phys.: Conf. Ser.* **1372** 012014

Development and Verification of Three-Dimensional Model of Femoral Bone: Finite Element Analysis

Aishah Umairah Abd Aziz[1,4], Hong Seng Gan[2], Ahmad Kafrawi Nasution[5], Mohammed Rafiq Abdul Kadir[3,4] and Muhammad Hanif Ramlee[1,4*]

[1] Medical Devices and Technology Centre (MEDITEC), Institute of Human Centered Engineering (i-Humen), Universiti Teknologi Malaysia, 81310, Johor, Malaysia.
[2] Universiti Kuala Lumpur, British Malaysian Institute, Batu 8 Jalan Sg. Pusu, 53100 Gombak, Selangor, Malaysia
[3] Sport Innovation & Technology Centre (SITC), Institute of Human Centered Engineering (i-Humen), Universiti Teknologi Malaysia, 81310, Johor, Malaysia.
[4] Bioinspired Innovation Group (BIOINSPIRA), School of Biomedical Engineering and Health Sciences, Faculty of Engineering, Universiti Teknologi Malaysia, 81310, Johor, Malaysia.
[5] Faculty of Engineering, Universiti Muhammadiyah Riau, Pekan Baru, Indonesia.

[*] muhammad.hanif.ramlee@biomedical.utm.my

Abstract. Development of reconstructed three dimensional (3D) model of bones has successfully emerged from time to time. It is used in many engineering applications especially by researchers to help them in the simulation process through finite element analysis (FEA). Besides, the 3D model can also be used by surgeons with the help of engineers, in improving their pre-surgical planning and designing of patient-specific implants or surgical instruments. Focusing on the femur, this paper proposes a method in developing a reconstructed three-dimensional femoral bone model and verifying it through analysis by using the Finite Element Method. Load of 600N was applied on the most proximal point of the femoral head under axial loading. The strain value observed in the model were 0.35 mε, 0.242 mε, and 0.146 mε. These values were slightly higher than the value measured in cadaveric specimen in the previous study, but it showed the same pattern. Hence, it can be concluded that the development method is reliable.

1. Introduction

Femoral bone is the longest and heaviest bone in the human body. It helps to bear the body weight and hence, it plays significant roles in most of the daily activities. On that account, vast studies have been continuously done to better understand the femur. Researchers need to simulate the bone in various circumstances that sometimes, are not feasible in a normal test setup but, is possible through a numerical study by using Finite Element Method (FEM) [1]. FEM helps to observe the mechanical effects (stress, strain, displacement and etc.) at any desired point through its finite elements. For that, various studies have successfully employed Finite Element Analysis (FEA) as their research method [2-7]. A close up to one of the cases, researchers are studying the healing of a broken bone. There are many variables that need to be considered and observed to ensure the study is in the correct direction. To begin the analysis, the bone needs to be remodeled appropriately. An accurate and reliable three-dimensional model (3D) will help in producing reliable results [8]. Previously, researchers opted for a simple rod or tube to represent the bone. Henderson D. J. et. al., claimed that the model was symmetric and uniform thus, any variability throughout the testing could be minimized [9]. This method had raised several issues. In most of the cases where the rod or tube was chosen, the actual bone anatomy and mechanical properties were almost totally neglected, in which it will affect the reliability and accuracy of the result [10]. On the result and analysis part, strain is always observed in evaluating bone mechanical behavior [11]. For example, in the bone healing process, the callus formation is dependent on the strain value [12-16]. Thus, it is important for researchers to observe the strain of the bone and fundamentally, the strain is measured following equation (1) [15].

$$\text{strain}, \varepsilon = \frac{\Delta L}{L} \quad (1)$$

The accuracy and reliability of the modeling could be improved by using a correct anatomical model of bone through the reconstruction method. The stacks of two dimensional (2D) ct data of a real bone will be reconstructed into a 3D model, which will be used in the analysis, superseded the tube. To ensure that the developed model is correct and reliable, it needs to be validated and verified. Validated means a direct comparison with the physical occurrence, whereas, for verification, it relates to the actual solving of the equations, which is influenced by modeling representation [1]. Thus, in this paper, it is aiming to propose a method of developing a reconstructed 3D model of femoral bone and also to provide a verified analysis on the developed model by comparing the results with other literature. The model was tested under axial loading through FEM.

2. Methodology

Since this is basically a preliminary study, the modeling was made to mimic a healthy environment experienced by everyone most of the time. Hence, CT dataset of a healthy 27 years old man with 169 cm height and 75kg weight was chosen as the subject of the study. The data was obtained from Hospital Tengku Ampuan Afzan, Kuantan, Pahang, Malaysia. CT datasets, stored in DICOM format file, is a collection of grey scale images resulted from the attenuation of x-ray beams. The darker the image, the less attenuate the x-ray beam is. The ct images were captured for every 1.5 mm along the femur and since the attenuation rate is higher in bone, the image of the femur appears to be white.

2.1. Development of the reconstructed 3D femoral bone model

An image processing software Mimics (Materialise, Leuven, Belgium) was used in reconstructing and modeling of the femur. The ct data was loaded into the software and the bone was segmented based on the Hounsfield Unit (HU) set in 'thresholding' tool. The HU for bone ranges from 226 to 3071 HU. 2D image of the bone was then automatically calculated as shown in figure 1(a). To segment the region of interest (ROI), which is the femur, both 'edit mask' and 'region growing' tools were utilized (figure 1(b)). Then, the segmented femur was converted to a 3D form as in figure 1(c). All files are then saved in the stereolithographic (STL) format. The process can be best summarized in figure 1.

2.2. Mesh generation

As in figure 1(d), the next step done was mesh generation by using 3-MATIC software (Materialise, Leuven, Belgium). The tetrahedral element was used as it had been discussed to have higher accuracy in contrast to its equivalent lesser node element [17]. From the mesh convergence analysis, it was found that the results converge at 4.5mm. To highlight, mesh convergence analysis was done to determine the optimum mesh size to be used so that the analysis result does not change with any changes in mesh size. Smaller size results in more accurate and reliable results [18] but, it requires a longer running time and bigger computer resources. Hence, optimized mesh size is needed in order to balance them. Reducing the element size or *h*-refinement method was used throughout the modeling domain.

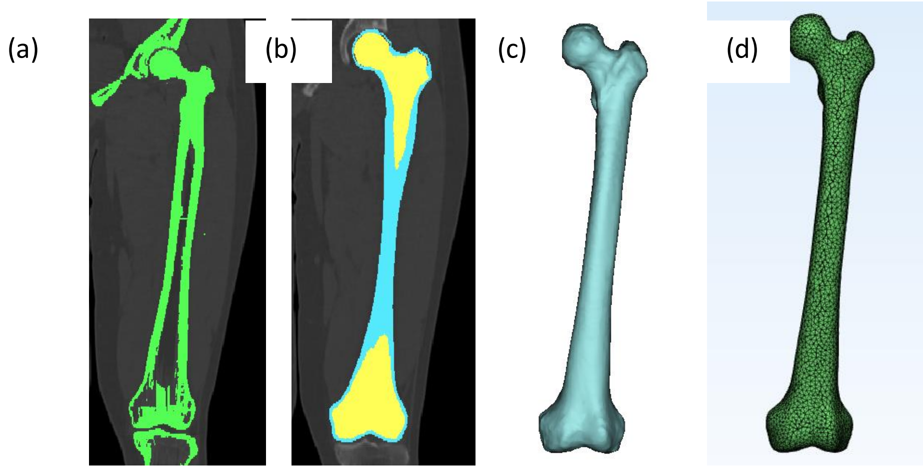

Figure 1. 3D femoral bone model development process from (a) 2D scanned image into a (b) segmented image based on the ROI, following by a (c) 3D reconstructed image and finally the (d) meshed model of the developed model.

2.3. Finite Element Analysis

The 3D model was then analyzed through Finite Element Method in Marc Mentat (MSC.Software, Santa Ana, CA). For simplification purpose, the material properties for bone were set as isotropic, homogenous and linearly elastic. The Young's modulus and Poisson's ratio were 7GPa and 0.3 respectively [7, 19]. The boundary condition applied in this study were as follows: distal end of the femur was set to fix in all cases, and to mimic the stance phase, as well as to verify the analysis with study made by Hensley et al [11], 600N point load was assigned at the femoral head in axial direction. Point load was chosen to represent a hip joint force in which it simplifies the hip muscle reaction as widely used in prior studies [3, 20, 21]. Figure 2 shows the developed 3D model of the femur with an illustrated boundary condition used in the analysis. The predicted linear strain was observed and compared in the analysis to verify the 3D femoral bone model. For that reason, three points for strain measurement were chosen following the cadaveric study made by Hensley S. et al [11]. They were located medially, starting with increment 25mm, starting 25mm below the lesser trochanter of the femur.

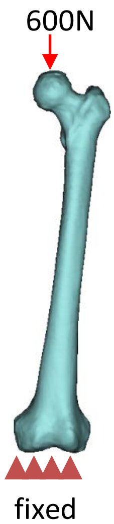

Figure 2. Boundary condition used in the Finite Element Analysis of developed femoral bone model.

3. Results and discussion

3.1. 3D model of the femur

The development steps in modeling the bone were successfully adopted. The model was visually compared to the standard anatomy of a femur to ensure the morphology of the model is correct. Through the proposed method, researchers are allowed to construct the bone in a faster way besides improving the reliability of their results as they are using actual bone anatomy. However, there are also some limitations and drawbacks in modeling the femur. For example, the model was assumed to be homogenous, linearly isotropic material. It contradicted the real bone composition which is non-linearly, orthotropic structure. Nonetheless, previous studies had successfully shown a good agreement between the cadaveric specimen and finite element model in spite of their simplified finite element material properties [22]. Unlike the real bone structure, the developed model was set as one material. It supposedly consists of both cortical and cancellous layer as done in many works of literature [3, 10, 23]. However, there is also literature that had been using a correct anatomical shape of the femoral model with a single material property [7, 17, 24, 25]. These mean that the simplifications of the bone material properties do give reliable results. As it is merely a preliminary study and considering the time restriction, only one subject was used throughout the analysis [26]. Thus, in the future, more subjects could be considered to further verified the proposed method.

3.2. Finite element analysis

From the analysis of the developed bone model, the maximum strain found at the three points were 0.35mε, 0.242mε, and 0.146mε respectively. These measurements were obtained at the white circle region as shown in figure 3. Low strain ranges from 0.082 mε and below was observed about half of the lower part of the model. This is due to the fix boundary condition at the distal condyle. Although the values obtained were higher than the cadaveric study by Hensley S. et al., it showed the same pattern with the DIC measured data. As the interested points went downwards, the strain values decreased. However, it was not in the case of strain gage measurement (table 1). Hensley S. et al., in their study, they were comparing the performance of DIC strain measurement techniques with strain gage for the use in biomechanical test materials [11]. They found that DIC measurement is more reliable compared to strain gage in measuring a material with more variability. Since the results from this study are in agreement to the DIC data, the 3D model developed is concluded to be verified and can be reliably used for future work.

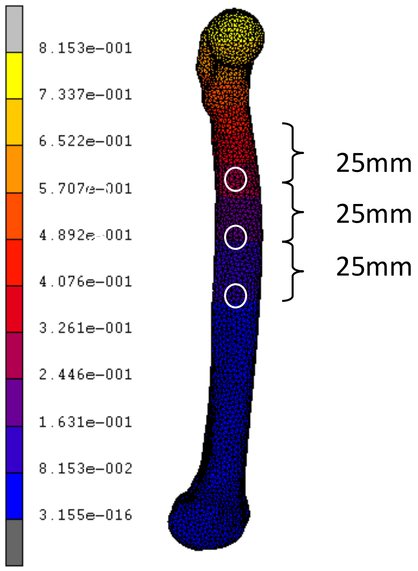

Figure 3. Maximum strain value for the analysis of the developed model.

Table 1. The maximum strain on the reconstructed model of the femur as compared to the model used by Hensley S. et al. [11].

	Point 1 (mε)	Point 2 (mε)	Point 3 (mε)
Hensley S. et al. (Strain gage)	0.125	0.120	0.125
Hensley S. et al. (DIC)	0.130	0.122	0.060
New method	0.350	0.242	0.146

4. Conclusion

The 3D femoral bone model was developed following the sequence of development steps. It was also analyzed through FEM, and despite some simplifications used in this study, the results obtained were still in agreement with the prior study. However, in the future, authors shall improve the modelling process by including more realistic material properties of human bone in an effort to obtain higher accuracy.

Acknowledgment

This work was financially supported by the Universiti Teknologi Malaysia under the Research University Grant (20H20, 4J358, 15J84 and 20H26), Fundamental Research Grant Scheme (FRGS) Ministry of Education Malaysia (278784-294883), Ministry of Energy, Science, Technology, Environment and Climate Change (EF0618I1166 and 4S144) and The Ministries of Research, Technology, and Higher Education Republic of Indonesia through Director of Research and Community Service with contract number: 009/L10/AK.04/KONTRAK-PENELITIAN/2019.

References

[1] MacLeod, A.R., Rose, H., Gill, H.S.: A validated open-source multisolver fourth-generation composite femur model. Journal of biomechanical engineering. **138**(12) (2016) 124501-124509

[2] Mehboob, A., Chang, S.H.: Biomechanical simulation of healing process of fractured femoral shaft applied by composite intramedullary nails according to fracture configuration. Composite structures. **185** (2018) 81-93

[3] Zhou, J.J., Zhao, M., Liu, D., Liu, H.Y., Du, C.F.: Biomechanical property of a newly designed assembly locking compression plate: Three-dimensional finite element analysis. Journal of healthcare engineering. (2017) 1-10

[4] Lopes, V.M.M., Neto, M.A., Amaro, A.M., Roseiro, L.M., Paulino, M.F.: FE and experimental study on how the cortex material properties of synthetic femurs affect strain levels. Medical engineering & physics. **46** (2017) 96-109

[5] Konya, M.N., Verim, O.: Numerical optimization of the position in femoral head of proximal locking screws of proximal femoral nail system; biomechanical study. Balkan medical journal. **34**(5) (2017) 425-431

[6] Giordano, V., Godoy-Santos, A.L., Belangero, W.D., Pires, R.E.S., Labronici, P.J., Koch, H.A.: Finite element analysis of the equivalent stress distribution in schanz screws during the use of a femoral fracture distractor. Revista brasiliera de ortopedia. **52**(4) (2017) 396-401

[7] Kumar, P.B., Parhi, D.R.: Vibrational characteristics and stress analysis in a human femur bone. Materials today: Proceedings. **4**(9) (2017) 10084-10087

[8] Fengjiao, G., Guanjun, Z., Jie, L., Shujing, W., Xu, L.: Study on validation method for femur finite element model under multiple loading conditions. IOP conference series: Materials science and engineering. **322**(2) (2018) 022061

[9] Henderson, D.J., Rushbrook, J.L., Harwood, P.J., Stewart, T.D.: What are the biomechanical properties of the taylor spatial frame™? Clinical orthopaedics and related research®. **475**(5) (2017) 1472-1482

[10] Mohd Aspar, M.A.S., Abd Razak, N.A., Abdul Kadir, M.R., Manap, H.: Development of 3-dimensional model of femur bone considering cortical and cancellous structures. International journal of engineering technology and sciences. **7** (1) (2017)

[11] Hensley, S., Christensen, M., Small, S., Archer, D., Lakes, E., Rogge, R.: Digital image correlation techniques for strain measurement in a variety of biomechanical test models. Acta of bioengineering and biomechanics. **19**(3) (2017)

[12] Epari, D.R., Taylor, W.R., Heller, M.O., Duda, G.N.: Mechanical conditions in the initial phase of bone healing. Clinical biomechanics. **21**(6) (2006) 646-655

[13] Boccaccio, A., Pappalettere, C. Mechanobiology of fracture healing: Basic principles and applications in orthodontics and orthopaedics. Theoretical biomechanics. Rijeka. (2011) 21-48.

[14] Pauwels, F.: “Grundriß einer biomechanik der frakturheilung. Verh dtsch orthop ges 34. Kongres 62-108,” ges abh. Springer, Berlin. (1980)

[15] Hak, D.J., Toker, S., Yi, C., Toreson, J., Stahel, P.F.: The influence of fracture fixation biomechanics on fracture healing. Orthopedics. **33**(10) (2010) 752-755

[16] Tan, B.B., Shanmugam, R., Gunalan, R., Chua, Y.P., Hossain, G., Saw, A.: A biomechanical comparison between taylor’s spatial frame and ilizarov external fixator. Malaysian orthopedic journal. **8**(2) (2014) 35-39

[17] N. Mughal, U., Khawaja, H., Moatamedi, M.: Finite element analysis of human femur bone. **9** (2015) 101-108

[18] Miura, M., Nakamura, J., Matsuura, Y., Wako, Y., Suzuki, T., Hagiwara, S., 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**(1) (2017) 536

[19] Vijayakumar, R., Madheswaran, M.: Modal analysis of femur bone using finite element method for healthcare system. 2017 conference on emerging devices and smart systems (2017) 224-228

[20] 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. **49**(7) (2016) 1064-1070

[21] Montanini, R., Filardi, V.: In vitro biomechanical evaluation of antegrade femoral nailing at early and late postoperative stages. Medical engineering & physics. **32**(8) (2010) 889-897

[22] Bettamer, A., Hambli, R., Allaoui, S., Almhdie-Imjabber, A.: Using visual image measurements to validate a novel finite element model of crack propagation and fracture patterns of proximal femur. Computer methods in biomechanics and biomedical engineering: imaging & visualization. **5**(4) (2017) 251-262

[23] Bayoglu, R., Okyar, A.F.: Implementation of boundary conditions in modeling the femur is critical for the evaluation of distal intramedullary nailing. Medical engineering & physics. **37**(11) (2015) 1053-1060

[24] Belaid, D., Bouchoucha, A.: Modeling of the mechanical behavior of the human femur: Stress analysis and strain 1702 (2015) 190023

[25] Dhanopia, A., Bhargava, M.: Finite element analysis of human fractured femur bone implantation with pmma thermoplastic prosthetic plate. Procedia engineering. **173** (2017) 1658-1665

[26] Ramlee MH, Sulong MA, Garcia-Nieto E, Penaranda DA, Felip AR, Abdul Kadir MR. Biomechanical features of six design of the delta external fixator for treating Pilon fracture: a finite element study. Medical & biological engineering & computing. **56**(10) 2018 1925-1938.