International Journal of Engineering & Technology, 7 (4.26) (2018) 20-24

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper



Simplified Femoral Bone Model for Numerical Analysis of Proximal Femur

M. H. Khan[1]*, R. Daud[1], M. K. Ali Hassan[1], A. F. Azizan[2]

[1] *Fracture and Damage Mechanics (FDM) SIG, School of Mechatronic Engineering, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia*

[2] *Hospital Tuanku Fauziah, 01000 Kangar, Perlis, Malaysia*

**Corresponding author E-mail: hazrafkhan@gmail.com*

Abstract

Numerous studies had been conducted to analyze the performance of implant for treating femoral head fractures utilizing finite element (FE) method. However, most studies use the whole femoral bone structure as the main model for conducting the numerical analysis. This will significantly increase the complexity to calculate the solution due to the higher number of meshing nodes and elements. Simplified femoral bone model without the shaft and distal region may be able to solve this issue. Thus, this study aimed to investigate the viability of simplified femoral bone model for conducting a numerical analysis of femoral head. In this study, two proximal bone models were constructed; by creating a cutting plane orthogonal to the shaft of the original model and another by cutting a plane orthogonal to the femoral head. The maximum stresses were measured along the femoral neck based from the simulated von Mises stresses. Based from the results, the proximal bone model with cut plane orthogonal to the femoral shaft showed the closes overall stress distribution to the original femoral bone model with 4.32% error. Thus, the simplified model is suitable to be used to replace the original femoral model for proximal femur FE analysis.

Keywords: *Proximal Femur; Finite Element Analysis; Simplified Model.*

1. Introduction

Hip fractures are among the most common and serious musculoskeletal injury presenting to orthopedic surgeons [1]. Femur bone can be divided into three main classification region which are the upper (proximal), body (shaft) and lower (distal) [2]. Although femoral head (proximal) fractures were labelled as a rare type of injury [3], the incidence of these fractures has increased proportionally with the incidence of high energy trauma such as motor vehicle accidents [1, 4]. Elderly individuals with osteoporosis are also at most risk to sustain this type of fracture typically due to a low energy fall [4]. It was estimated that the population aged over 50 will increase by 28% between the year of 2004 and 2031 [5]. Due to this estimation, it is predicted that there will be 45% to 75% increase in the number of hip fractures per annum since most of the individuals who sustained hip fractures were among the elderly population [1].

Finite element (FE) analysis is one of the most common methods for examining the structural stresses developed in engineering mechanics [6, 7]. This method has been used in many engineering applications due to its ability to repeatedly perform constant analysis with different sets of boundary conditions, ultimately reducing time and cost for analyzing certain mechanical behavior. FE analysis can also be used for calculating stresses in human bone as well as assist in optimizing implant design. Meshing is an important part of FE analysis where is can influence the accuracy as well as the speed for finding the solution. Mesh comprises of nodes and elements for analysis. Technically, the accuracy of the FE solution is directly correlate to the number of nodes and elements. However, increasing the number of nodes and elements can increase the difficulty for the program to find the solution thus extending the time for obtaining the FE results.

FE analysis had been applied in numerous studies involving the proximal femur of human bone [8, 9]. However, the whole femur bone including the shaft and distal were typically used for conducting these analyses since all these regions are a part of a single bone. Due to the larger surface volume of this model, higher processing power is needed for conducting the analysis. In this study, two simplified proximal femur models with significantly lower surface volume were introduced. The main objective of this study was to find out whether these models could provide the same accuracy as using the original femur for conducting the analysis on the proximal femur.

2. Materials & Methods

The original CAD model of the femoral bone was obtained from a three-dimensional (3D) scanning process where an actual bone specimen of a femur was scanned with a 3D laser scanner to obtain its external topology [10]. The CAD model of the original femur was then imported into Creo Parametric 3D modelling software (ProE) for modelling the simplified bone model. Figure 1 shows the original femoral bone model imported into ProE modelling software.

Copyright © 2018 Authors. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



Fig. 1: Original CAD model of the femoral bone

The first simplified proximal femur bone model (model A) was created by making a cut plane orthogonal to the femoral shaft. The whole bottom section of the bone (shaft & distal) was then removed, leaving only the proximal femur or the femoral head. The second bone model (model B) was also created by using the method but with a cut plane orthogonal to the femoral shaft. The models were labeled as model A and model B respectively for reference. Figure 2 shows the image of the simplified models along with their respective cut plane angle. The length of both proximal femur models was approximately 30% of the total length of the original model. The simplified models from ProE were then imported into ANSYS for conducting the FE analysis.

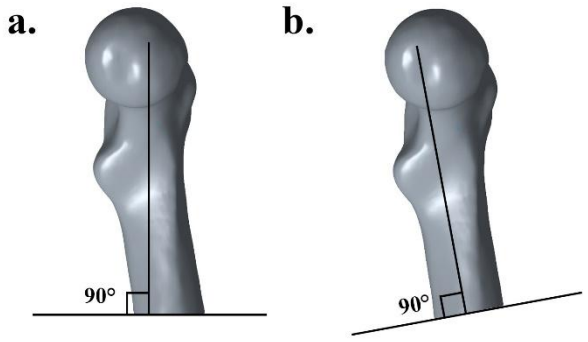

Fig. 2: Simplified proximal femur bone model (a) with cut plane orthogonal to the femoral head and neck (model A); (b) with cut plane orthogonal to the femoral shaft (model B)

The next step of modelling the femoral bone is to assign its material properties and the boundary conditions. Since bone is a complex biological structure consist of many internal structures such as cortical and cancellous bone, assuming the bone to be a solid with isotropic behavior would be inaccurate. Instead, the bone was assumed to be orthotropic to increase the accuracy of the final FE results. Unlike materials with isotropic behavior which have the same properties at different directions, orthotropic materials have properties which differ along its orthogonal directions. Table 1 shows the value of each properties of the orthotropic bone used in this study.

Forces with magnitudes of 1000 N and 150 N were applied along the horizontal and vertical axis on the femoral respectively. These forces were applied to simulate the static condition during a normal standing posture. The resultant force is 1012 N and acted 8.5° above the horizontal axis at the femoral head. Fixed support was also applied on all three models; on the distal for the original model and on the cut surface for the simplified models. Figure 3 shows the magnitude of forces applied on the femoral head whereas Figure 4 shows the location of fix support on all three models. 2 mm body sizing method was applied for meshing all three models; approximately 268,000 nodes and 191,000 elements for both model A and model B, whereas 830,573 nodes and 593,771 elements for the original femur model.

Table 1: Orthotropic elasticity and stress limits of human bone

Orthotropic Elasticity	Values	Orthotropic Stress Limits	Values
Young's Modulus X Direction (MPa)	19400	Tensile X Direction (MPa)	135
Young's Modulus Y Direction (MPa)	12600	Tensile Y Direction (MPa)	50
Young's Modulus Z Direction (MPa)	12600	Tensile Z Direction (MPa)	50
Poisson's Ratio XY	0.39	Compressive X Direction (MPa)	-250
Poisson's Ratio YZ	0.3	Compressive Y Direction (MPa)	-50
Poisson's Ratio XZ	0.39	Compressive Z Direction (MPa)	-50
Shear Modulus XY (MPa)	5700	Shear XY (MPa)	65
Shear Modulus YZ (MPa)	4850	Shear YZ (MPa)	65
Shear Modulus XZ (MPa)	5700	Shear XZ (MPa)	56

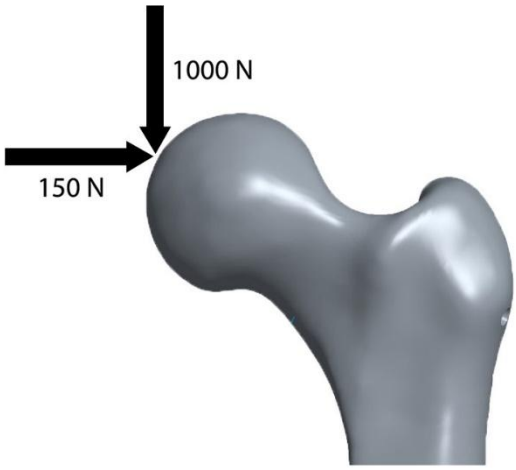

Fig3: Magnitude of forces applied on the femoral head

Solution for von-mises equivalent stress was then solved in ANSYS for each model based from the applied boundary conditions. The overall maximum stress as well as the stresses along the top and bottom section of the femoral neck of each model were compared and analyzed. Figure 5 shows the 13 labels marked with the red dots for data collection on both the top and bottom section of the femoral neck. The stress from each of these labels were then plotted and compared with the data from the original femoral model. Equation 1 was used to calculate the error percentage between the stress from each simplified model (model A and model B) and the original femoral model.

$$\%Error(i) = \left| \frac{\sigma_{original,i} - \sigma_{simplified,i}}{\sigma_{simplified,i}} \right| \quad (1)$$

Definition 1: $\sigma_{original,i}$ is the stress on the original femur model and $\sigma_{simplified,i}$ is the stress on the simplified femoral model.

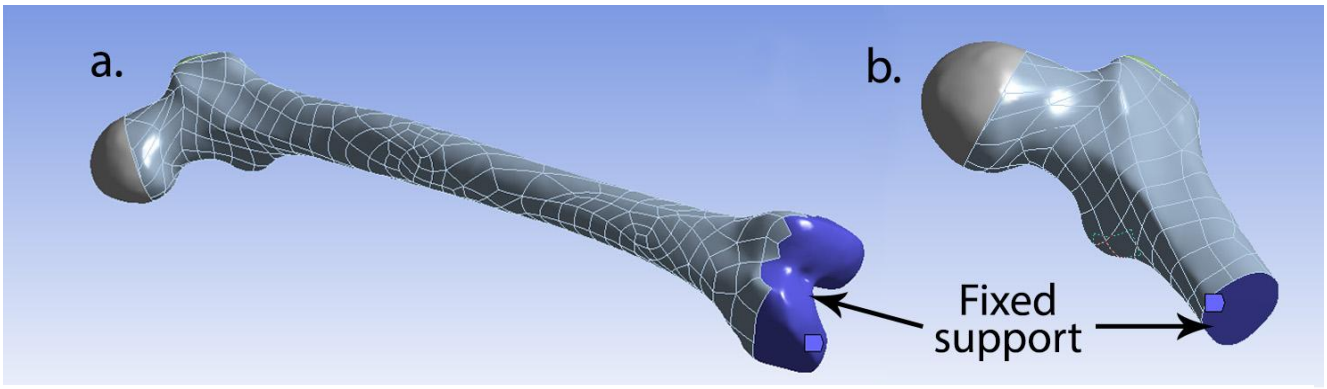

Fig. 4: Location of fixed support for (a) the original femur model; (b) model A and model B

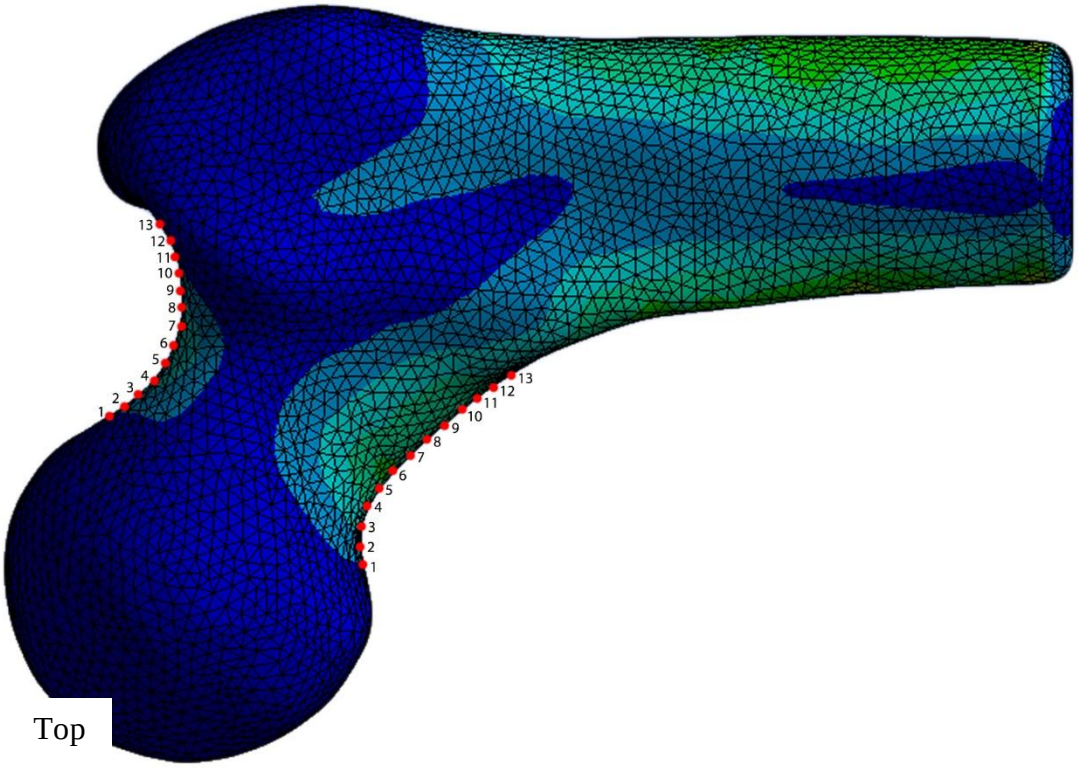

Fig. 5: Labels which were marked with the red dots for data collection

3. Results and discussions

The stress along the top section of the femoral necks is shown in Figure 6. Based from Figure 6, the distribution of stresses for all three models seems to show the same bell curve pattern where the maximum stress is located at the middle section of the femoral neck. However, the gap between the stress data of model A and the original model seems to be larger than the gap between model B and the original femur. This observation can be proved by calculating the percentage of error between these data. Based on the calculation using Eq. 1, the average percentage of error between the stress of model A and the original model is 18.3% whereas between model B and the original model is 8.44%. Thus, this proved that for the top section of femoral neck, model B have closer overall stress distribution to the original femur compared to model A.

Figure 7 shows the stress along the bottom section of the femoral necks. Based from this figure, it can be observed that the overall stress distribution of model B are much similar to the original femur compared to model A. The average percentage of error for stress data between model A and the original model is 10.27% whereas between model B and the original model is 4.32%. Similar to the result from Figure 6, the overall stress distribution on the bottom section of the femoral neck of model B also showed a lower percentage of error compared to model A. Due to the closer overall stress distribution with the original model, model B is much more preferable than model A for conducting the numerical analysis of proximal femur.

The maximum stress on the original model, model A and model B are 16.49 MPa, 19.30 MPa and 21.36 MPa respectively. The maximum stress for both model A and model B are located at the bottom section of the femoral head where it was cut. The higher maximum stress of both models was probably due to the stress concentration on the cut section. The original model has larger surface area to distribute the stress which explains its lower maximum stress. Although the maximum stress values of both simplified models differ a lot from the original model, it will not affect the reliability of using these simplified models for the numerical analysis of femoral neck.

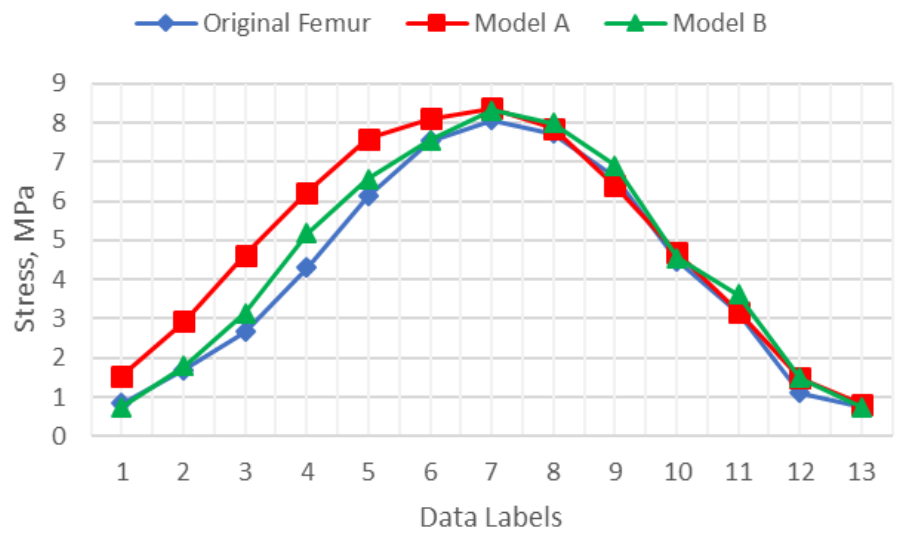

Fig. 7: Stress at each data labels along the top section of femoral neck

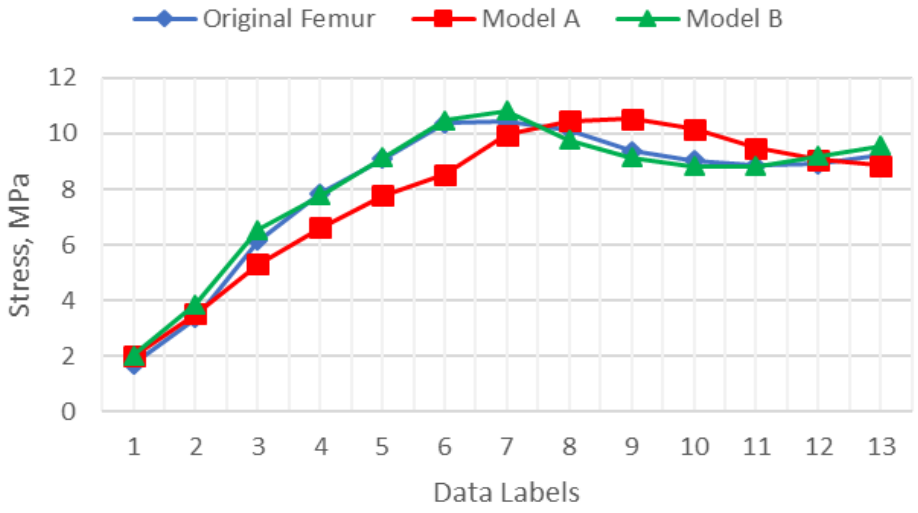

Fig. 6: Stress at each data labels along the bottom section of femoral neck

Although the results seem to conform with the initial hypothesis of this study, further analysis needs to be done to find the best criteria of the simplified model for replacing the original model. These analyses include varying the cut angle and cut length of the femoral head as well as the force applied on it. Minimal manipulated variables were introduced in this study to prove the initial hypothesis that the simplified model can be used to replace the original model for the FE analysis of femoral neck.

4. Conclusions

Two simplified models of femoral head were created by cutting the original femoral bone at two different angles; by making a cut plane orthogonal to the femoral head (model A) and a cut plane orthogonal to the femoral shaft (model B). Stress distribution along the top and bottom section of the femoral neck of each model (including the original femur) were compared and analyzed. The results showed that model B had the closest stress distribution on the femoral neck with the original femur, with average error of 8.44% and 4.32% along the top and bottom section of the femoral neck respectively. The results also proved that the simplified models, especially model B, can be used to replace the whole femoral model for the FE analysis of the femoral neck. This will help to reduce the time and difficulty for the software to calculate the solution while still producing accurate results.

Acknowledgement

This work was financially supported by the Ministry of Higher Education Malaysia grant, Fundamental Research Grant Scheme (FRGS 9003-00632).

References

[1] A. Winter, H. Bradman, C. Fraser, and G. Holt, "The management of intracapsular hip fractures," *Orthopaedics and Trauma*, vol. 30, no. 2, pp. 93-102, 2016.

[2] O. Jones. (2017, December 22). *The femur*. Available: http://teachmeanatomy.info/lower-limb/bones/femur/

[3] J. R. Ross and M. J. Gardner, "Femoral head fractures," *Curr Rev Musculoskelet Med*, vol. 5, no. 3, pp. 199-205, 2012.

[4] J. A. John Keating, "The management of intracapsular fracture of the femoral neck," *Orthopedics and Trauma*, vol. 24, no. 1, pp. 42-52, 2009.

[5] B. W. Robert KC, Jevsevar DS, Sevarino K., "Management of hip fractures in the elderly.," *J Am Acad Orthop Surg*, vol. 23, 2015.

[6] T. Stolarski, Y. Nakasone, and S. Yoshimoto, *Engineering Analysis with ANSYS Software*. Butterworth-Heinemann, 2011, p. 480.

[7] U. N. Mughal, H. A. Khawaja, and M. Moatamedi, "Finite element analysis of human femur bone," *Int. Jnl. of Multiphysics*, vol. 9, no. 2, pp. 101-108, 2015.
[8] M. Qasim *et al.*, "Patient-specific finite element estimated femur strength as a predictor of the risk of hip fracture: the effect of methodological determinants," *Osteoporos Int*, vol. 27, no. 9, pp. 2815-2822, Sep 2016.
[9] J. H. Keyak *et al.*, "Effect of finite element model loading condition on fracture risk assessment in men and women: the AGES-Reykjavik study," *Bone*, vol. 57, no. 1, pp. 18-29, Nov 2013.
[10] E. Isaza, L. García, and E. Salazar, "Determination of Mechanic Resistance of Osseous Element through Finite Element Modeling," presented at the Comsol Conference, Boston, 2013.