The Effect of Material and Model Type on Behavior of Femur Bone under: Static, Modal and Dynamic Analysis

تأثير مادة ونوع النموذج الرقمي لعظم فخذ سليم على سلوكه في التحليل الاستاتيكي والديناميكي والاهتزازي

Dr. Professor
Adnan Naji Jameel

Ph.D student
Mahmood Wael Saeed

Engineering College - Baghdad University

Abstract:

Three types of analysis are represented in this study for three intact finite element (FE) femur bones models with two material types. Static analysis studied stress distribution in intact femur bone under excessive load that was stumbling case (critical and dangerous load case). The dynamic load was assumed to be impact load and natural frequencies of FE models were calculated in order to specify the failure characteristic of bone in fractured mode. ANSYS workbench version .14 was used in these numerical analyses. The results represent the effect of changing material type where orthotropic material shows lower stresses values in both static and dynamic analysis with respect to isotropic material.

Keywords: Intact femur, stumbling case, isotropic, orthotropic, impact load, failure mode.

1. INTRODUCTION

Of all human long bones, femurs are the heaviest, longest, and strongest. In younger people with good bone quality, normal bone properties, femur fractures usually require high energy events such as the 6.5 million automobile accidents that occur annually in the U.S. alone, (U.S**.** Department of Transportation, 2007). In elderly people with poor bone quality, osteoporosis or osteopenia, low energy impact from falls is the most frequent cause of femoral neck fracture (Pankovich et.al. 2006), (Haidukewych et. al., 2008). Moreover, in the U.S. alone, about 231,000 hip replacement and 542,000 knee replacement surgeries are done annually due to femoral diseases like osteoarthritis and bone cancer, (DeFrances CJ et. al., 2006).

The finite element method has become a particularly useful tool in analyzing the stresses in structures of complex shapes, loading and material behavior. An overview of its application in orthopedics during the last ten years has been presented by (Prendergast 1997)**.** For a complete and accurate indication of the stresses in the bone, the model must be modeled in three dimension system.

The finite element method enables a great variety of loading conditions and design variables to be changed easily but it is only an approximate method of solution; it represents the object being modelled as a finite number of degrees of freedom. The model will not converge to the solution of the physical structure under consideration however, unless the model is a precise representation of the structure. The accuracy of a finite element model will depend on the type of element used in the model and the fineness of the mesh, and is best evaluated by observing the convergence of the solution as the number of elements defining the problem is increased. The model was built with the largest possible number of elements in order to improve the accuracy of the solution. (J.H. Keyak et.al.1992)**.**

2. FINITE ELEMENT MODEL

The finite element (FEM) can now be considered as the most popular theoretical technique ever known to man, and it has been applied successfully to many engineering disciplines, such as structural mechanics, computational fluid dynamics, tribology, heat transfer , electromagnetism, biomechanics,… etc.

Three famous models of femur bone had been used in this study first model (CAD Model [IGES], Sawbones, Vashon, WA, USA) was obtained which was consist from cortical bone, lower and upper cancellous bone as shown in **Fig.1** where in this figure introduced the parts of this model . Second model has two major parts (cortical and cancellous) [.IGES] as shown in **Fig.2**. Third model regarded as one part (solid model)[.IGES] as shown in **Fig.3**. These 3D solid models are available in public domain derived from a CT-scan dataset of a synthetic human femur.

These three models were imported from external file to (ANSYS workbench software program version .14). The modeling process for each femur was accrued by introducing the meshing properties and material properties and then by generating mesh to whole model. The IGES format was imported to the commercial pre- and post-processor, Hyper mesh software. Using hyper mesh built-in ANSYS template, it was possible to export the model as an ANSYS 3-D database, (Altair Inc., USA, 2000).

The femur in this stage consisted of 30176 key points and 1350.0 lines.

In several cases, there were non-continuous and non-smooth curves in the model that induced errors in the output model; hence, a curve smoothing approach was implemented to avoid such incidence and to allow mesh generators more flexibility.

Once the curve smoothing was performed, the Boolean operations have been implemented in order to only consider the proximal femur as a cortical shell bone. The models were meshed with 10 nodes tetrahedral elements with the best possible mesh refinement. No contact element was considered in this work.

Three types of analysis were used in this study under different load cases where each of these models were analyzed under static, modal, and dynamic for the most important and dangerous load case on femur bone (stumbling load) where the load reached to 8.7 times the body weight **El'Sheikh, et.al. 2003.** Based on common geometry, it is practical to compare results from different FEM studies worldwide and besides, every FE models could be calibrated with data from experimental tests available in the literature. The latter one is of great importance as it is not always possible in biomechanics to do experimental tests for validating and verifying the numerical tests. More information about the physical object from which the standardized femur model has been derived is available from (M. Viceconti, 2001)**.**

3. MATERIAL PROPERTIES

The material properties of each type of femur bones that are used in present work are illustrated in Table1.Where in type1 the bone was modeled as an isotropic material A. (Completoa ,et. al., 2007) type2 the bone was assumed to be as an orthotropic material, S.A. (Asgaria,et.al., 2004)

In this study each of the above models was treated as isotropic properties and as orthotropic properties.

4. LOADING AND BOUNDARY CONDITION

For dynamic analysis, the load time curve during walking that applied as time history of the dynamic load components for 5 s shows that the maximum load applied on hip prosthesis reached to 8.7 times the body weight during stumbling case so the case with this excessive load should be studied (El'Sheikh, et.al. 2003).

For the same load value the above process was repeated but for static analysis. In this work the body weight assumed to be equal to 70kg. The applied load on the femoral head was taken to represent terminal stance during horizontal walking (El'Sheikh, et.al. 2003). Distal end of intact femur for the three models was fixed .For each reconstruction, the force was applied vertically on the upper cortical part (S. Shaha, et. al., 2012)**.**

5. RESULTS

For static analysis distributions at load of 8.7 times body weight are shown for two femur models in Fig.4, where these models assumed to be as an isotropic material. The standard isotropic three parts model experienced peak stresses which approximately less than the standard isotropic two parts by 150Mpa.

For dynamic analysis Fig.5 shows the maximum equivalent von misses stress for two models. In dynamic analysis the difference between the maximum von Misses equivalent stresses for these two models was not more than 4Mpa. The maximum value of von Misses stresses accrued at different time where in the standard isotropic two parts model the maximum value occurred at 0.00068sec while in the standard isotropic three parts model at 0.00031sec.

In Fig.6 the modal analysis results are represented for orthotropic three parts intact femur bone and for orthotropic solid intact femur bone respectively. In modal analysis six modes shapes was modeled for each intact femur models.

Figures 7-8 show the maximum von misses stresses in static analysis for both isotropic and orthotropic respectively. In these figures it is clear that the maximum stress value was in orthotropic two parts model while minimum value of equivalent stress was in isotropic three parts model.

In figures 9-10 results of dynamic stress analysis are represented for both isotropic and orthotropic material respectively, minimum stress in dynamic analysis was in orthotropic three parts model while the maximum value was in model of isotropic two parts.

In modal analysis the results show that the minimum frequency value was in orthotropic two parts model while the maximum value in isotropic three parts model, values shown in figures 11, 12 respectively.

From both static and dynamic results we can easily calculate an important designing factor that is dynamic load factor (DLF), its values shown in table.2 .

6. DISCUSSION

The aim of this study was to develop a practical FE model and to estimate the risk of bone failure during gait based on three analysis types firstly in static then in dynamic and modal under excessive load case that was stumbling in order to simulate this critical load type that any femoral bone can be loaded by it.

An adult human intact femur bone in three famous constructions is studied in present work. Direct comparison of the results from the present study to the literature is difficult because of inter-study methodological differences and the subject-specific nature of the results. However, three dimensional subject specific FE models were used to calculate the stress patterns due to loads derived from daily activities. The present work was compared with them where comparison shows large agreement in results.

In static analysis even though the results shown in previous images are obtained using a load case obtained from literature as well as average material properties, it is considered that they give an indication of critical places considering equivalent stress in a femur using this modeling approach, a next course of action, considering medical treatment of the patient or surgery planning, may be undertaken.

With respect to dynamic analysis for FE models each model shows different values of effective stress where in isotropic material for all models lower values of effective stress when they had been changed to orthotropic type. Where that changing leads to distribute the load and stress smoothly across the femur, same thing with respect to static analysis. In general the orthotropic material shows lower stresses values with compare to isotropic one. Where the difference between values of stresses under same load case but different material type leads to decrease effective stress value by 400Mpa for three parts model.

Ranges of natural frequencies had been calculated for each femur bone and compared with each other to show the effect of material so it is clear from that any changing or difference in frequency of intact femur bone could help the surgeon to predict if there was any disease or damage in bone construction or not. From table.2 values of magnification factor for each model is presented, these values can be considered as an important guide for designer to avoid critical load case by proper modifying in design process, these values give indication on range and relation of both dynamic and static stresses values.

It is recommended to utilize these values for designing total artificial femur bone with taking into account the compatibility situation with human body by trying on reducing high stresses values.

The essential purpose of this work was to give an image on magnification amount of dynamic stress compared to static stress, where the amount of magnification represented by magnification factor.

7. CONCLUSIONS

In this paper a full process of static, dynamic and modal structure analysis of femur is described. The process includes generating 3D model from CT-Scan data, meshing, assigning material properties and analyzing the structure. During the process, we also study the methods for mesh quality enhancement and apply this idea using specific software. In the presented work, the internal zone of 3D FE model, corresponding to cancellous bone, is further subdivided into a number of subzones of lower or larger stiffness, which is an approach until now only used in 2D analysis.

This paper may be considered as an introduction to analysis of biological structures by applying finite element method and FEM software. Also presents ranges of effective stress for static analysis.

1-In general the lowest values of effective stress were in orthotropic material for static and dynamic stress analysis.

2-Modal analysis is a precise method in predicting the bone strength and health which depends highly on its shape and distribution of its cortical or cancellous component.

3- Representing of bone as orthotropic material gives the nearest results to those for natural tested bones.

4- Difference in stresses values for static analysis reached to 57.1429% between isotropic and orthotropic three parts model.

5- In dynamic analysis the percentage difference in stresses values reached to 40% where bone of orthotropic properties shows less stress values than isotropic one.

6- Orthotropic model of three parts shows least DLF value comparing to other models and comparing to three parts model when it assumed to be isotropic material, so it is recommended to avoid modeling bone as an orthotropic material.

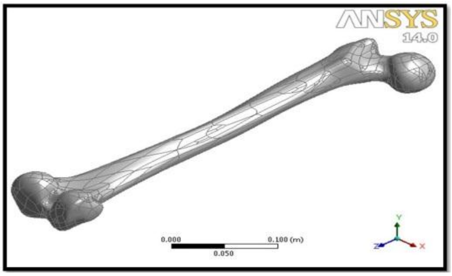

Figure.1 Three parts intact femur bone

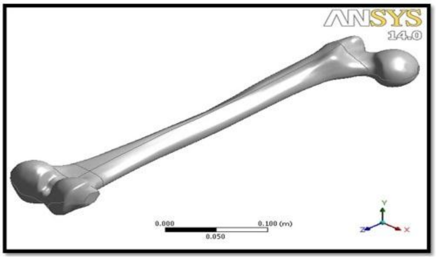

Figure.2 Two parts intact femur bone

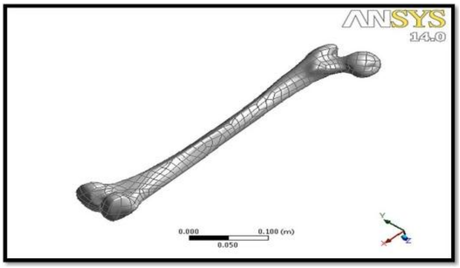

Figure.3 Solid intact femur bone

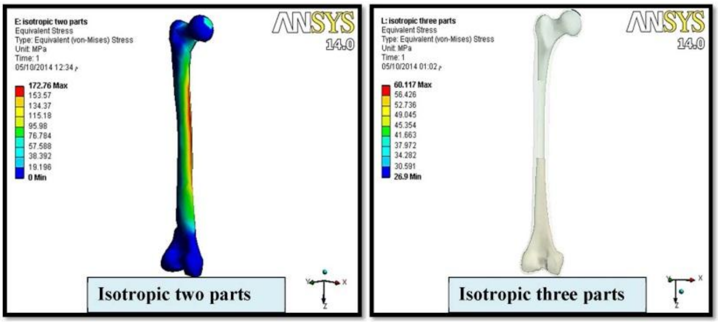

Figure.4 Maximum von misses equivalent stress for isotropic femur bone (Static analysis)

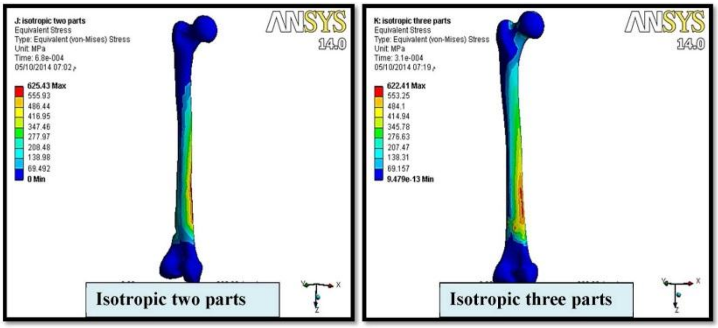

Figure.5 Maximum von misses equivalent stress for isotropic femur bone (Dynamic analysis)

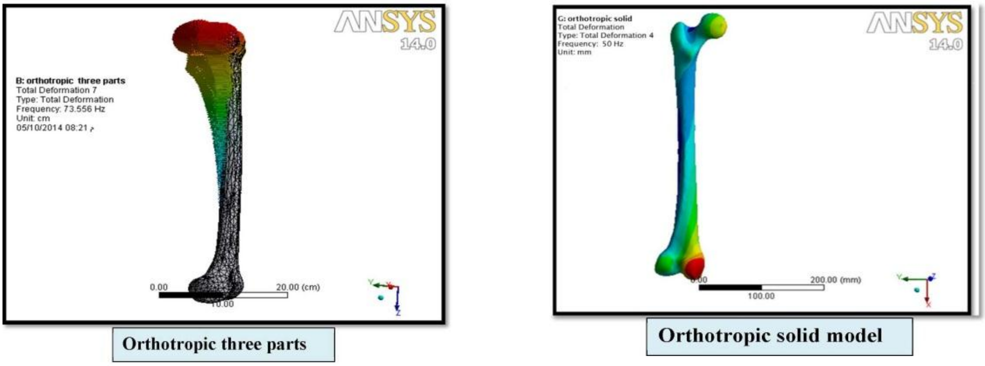

Figure.6 Modal analysis results for two intact femur bones

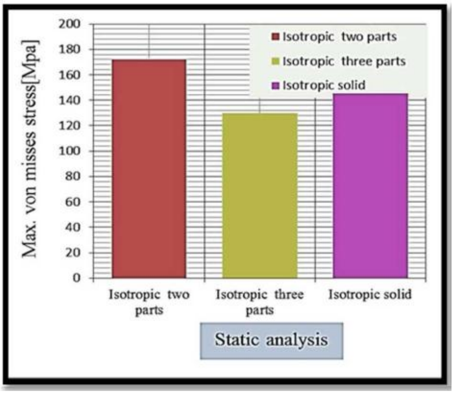

Figure.7 Maximum von misses stress for all models –static analysis- isotropic material

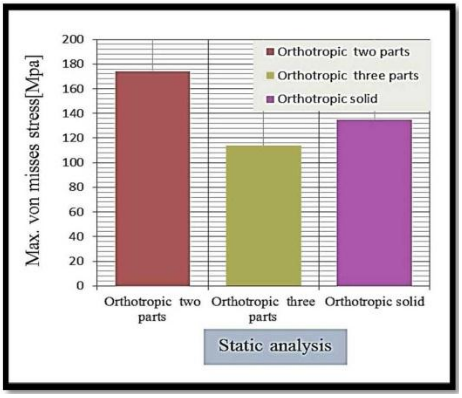

Figure.8 Maximum von misses stress for all models- static analysis- orthotropic material

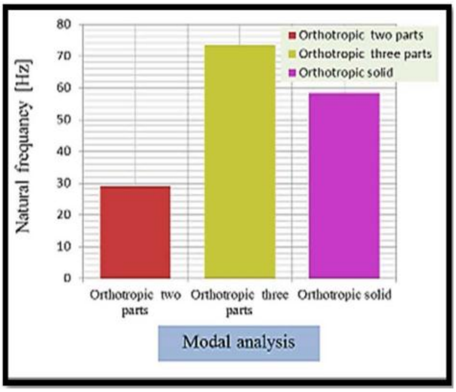

Figure.9 Maximum von misses stress for all models –dynamic analysis- isotropic material

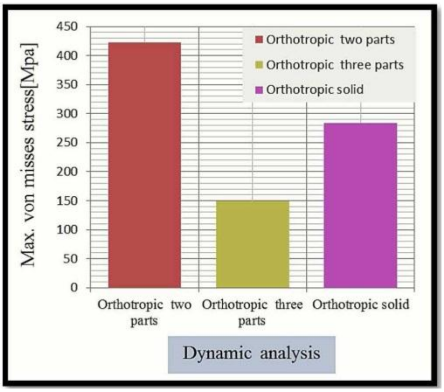

Figure.10 Maximum von misses stress for all models- dynamic analysis- orthotropic material

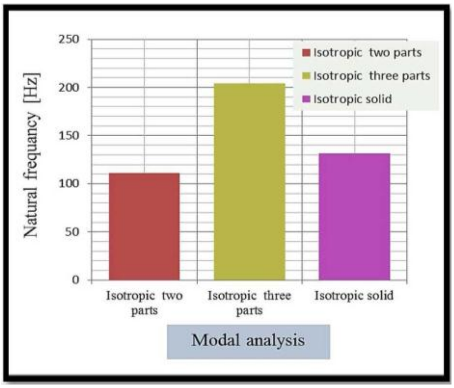

Figure.11 Natural frequency for all models –modal analysis- isotropic material

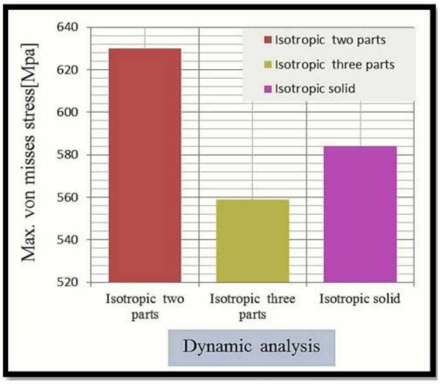

Figure.12 Natural frequency for all models –modal analysis- orthotropic material

Table.1 Material properties

Type	Young's modulus[Gpa]		Poisson's ratio	
isotropic	Cortical Bone	12.4	0.3	
	Cancellous Bone	0.104	0.3	
orthotropic	Young's modulus[Gpa]		Poisson's ratio	
	E_x	17.9	v_{xy}	0.26
	E_y	18.8	v_{xz}	0.31
	E_z	22.8	v_{yz}	0.37

Table.2 Dynamic load factor (DLF) for all models

Model's Type	Isotropic	Orthotropic
Two parts	3.5	2.4
Three parts	3.3	1.34
Solid	5	2

الخلاصة:

قدمت الدراسة الحالية تصور نظري (عددي) عن تأثير مادة ونوع النموذج الرقمي لعظم فخذ سليم على سلوكه في التحليل الاستاتيكي والديناميكي والاهتزازي. ستاتيكيا سلط حمل التعثر احد اخطر احمال عظم الفخذ وتم ايجاد توزيع الاجهادات اما في الدراسة الديناميكية فسلط ذات الحمل باعتباره حمل صدمي لمدة قصيرة. أما في التحليل الاهتزازي فقد تم حساب الترددات الطبيعية لعظم الفخذ. تم حساب النتائج رقميا باستخدام برنامج (ANSYS workbench.14) واظهرت النتائج للتحليلين الاستاتيكي والديناميكي ان المواد من نوع Orthotropic. اظهرت اجهادات اقل من تلك التي من نوعIsotropic .

الكلمات الرئيسية: عظم الفخذ السليم، نموذج رقمي، تحليل استاتيكي، تحليل ديناميكي، حمل صدمي.

REFERENCES

- U.S. Department of Transportation. Transportation Statistics Annual Report 2007. Washington, DC, USA, available from www.bts.gov.
- Pankovich AM, Elstrom JA. Intracapsular fractures of the proximal femur. In: Elstrom JA, Virkus WW, Pankovich AM, editors. Handbook of fractures. Toronto, ON: McGraw-Hill; 2006. p. 264–92.
- Haidukewych GJ. Proximal femur fractures. In: Sanders R, editor. Trauma: core knowledge in orthopaedics. Philadelphia, PA: Mosby-Elsevier; 2008. p. 253–63.
- DeFrances CJ, Lucas CA, Buie VC, Golosinskiy A. 2006 national hospital discharge survey. Natl Health Stat Report 2008; 5(July):1–20.
- Prendergast, P. J., "Finite element models in tissue mechanics and orthopaedic implant design," *Clinical Biomechanics,* 12(6): 343-366, 1997.
- J.H. Keyak, H.B. Skinner, Three-dimensional finite element modeling of bone: e2ects of element size, J. Biomed. Eng. 14 (1992) 483–489.
- Altair Hypermesh, Altair Hypermesh Manual, Altair Inc., USA, 2000.
- ANSYS, Ansys Theory Reference, SAS IP Inc., USA, 2001.

- H.F. El'Sheikh, B.J. MacDonald, M.S.J. Hashmi, 2003, Finite Element Simulation of The Hip Joint During Stumbling: a Comparison Between Static and Dynamic Loading, Journal of Materials Processing Technology vol.144, No. 2, pp. 249–255.
- M. Viceconti, The ISB finite element repository, Instituti Rizzoli, accessed on August 2001, http://isb.ri.ccf.org/data, 1997.
- S.A. Asgaria, A.M.S. Hamouda, S.B. Mansora, H. Singhc, E. M. R. Wirzaa, B. Prakashc,2004, Finite element modeling of a generic stemless hip implant design in comparison with conventional hip implants, vol.40, pp. 2027–2047
- A. Completoa, F. Fonsecab, J.A. Simoesa, Experimental validation of intact and implanted distal femur finite element models, 2007,Journal of Biomechanics 40 2467–2476
- Suraj Shaha, Habiba Bougherara, Emil H. Schemitsch, Rad Zdero, Biomechanical stress maps of an artificial femur obtained using a new infrared thermography technique validated by strain gages ,2012, Medical Engineering & Physics.