International Journal of Automotive and Mechanical Engineering (IJAME)
ISSN: 1985-9325 (Print); ISSN: 2180-1606 (Online); Volume 3, pp. 256-264, January-June 2011

PROBABILISTIC FINITE ELEMENT ANALYSIS OF VERTEBRAE OF THE LUMBAR SPINE UNDER HYPEREXTENSION LOADING

A. Zulkifli[1,a], A.K. Ariffin[1,b] and M.M. Rahman[2,c]

[1]Department of Mechanical & Materials Engineering
Faculty of Engineering & Built Environment, Universiti Kebangsaan Malaysia
46300 UKM, Bangi, Selangor, Malaysia
Phone: +603-89250200, Fax: +603-89216106
E-mail: zulkola@eng.ukm.my[a]; kamal@eng.ukm.my[b]
[2]Faculty of Mechanical Engineering, Universiti Malaysia Pahang
26600 Pekan, Kuantan, Pahang, Malaysia
Phone: +609-4242346, Fax: +603-4242202
Email: mustafizur@ump.edu.my[c]

ABSTRACT

The major goal of this study is to determine the stress on vertebrae subjected to hyperextension loading. In addition, probabilistic analysis was adopted in finite element analysis (FEA) to verify the parameters that affected failure. Probabilistic finite element (PFE) analysis plays an important role today in solving engineering problems in many fields of science and industry and has recently been applied in orthopaedic applications. A finite element model of the L2 vertebra was constructed in SolidWorks and imported by ANSYS 11.0 software for the analysis. For simplicity, vertebra components were modelled as isotropic and linear materials. A tetrahedral solid element was chosen as the element type because it is better suited to and more accurate in modelling problems with curved boundaries such as bone. A Monte Carlo simulation (MCS) technique was performed to conduct the probabilistic analysis using a built-in probabilistic module in ANSYS with 100 samples. It was found that the adjacent lower pedicle region depicted the highest stress with 1.21 MPa, and the probability of failure was 3%. The force applied to the facet (FORFCT) variable needs to be emphasized after sensitivity assessment revealed that this variable is very sensitive to the stress and displacement output parameters.

Keywords: Probabilistic, finite element analysis, lumbar spine, stress, hyperextension.

INTRODUCTION

In engineering, uncertainties are the most important thing to measure in order to make the analysis as real as in nature. Neglecting the existence of uncertainties in the biological system and environment can make the application fail even when the calculation suggests it is safe enough. However, the values of the variables that are working on the system cannot be predicted with certainty. Structural geometric properties, mechanical properties and the external loads are all uncertain in nature. In particular, the uncertainties in the external loads are very serious (Qiu and Wu, 2010). However, Taddei et al. (2006) found that bone stresses and strains in the proximal femur were more sensitive to uncertainties in the geometric representation than material properties. In the probabilistic approach, all uncertain variables are considered to be random and the uncertain problems are analysed based on their statistical properties (Qiu and Wu, 2010).

Hyperextension is a straightening movement that goes beyond the normal, healthy boundaries of the joint and often results in orthopaedic injury. This movement will produce an extreme condition and create a failure in the vertebra. It may occur during training by athletes and or sometimes by accident. The pedicle is most commonly the part where fractures are observed during trauma, experimentally and clinically (Xia et al., 2006). Occasionally, a pedicular fracture may occur that suggests a causative relationship with the patient's hyperactivity (Sirvanci et al., 2002).

Finite element analysis is one of the most advanced simulation techniques and has been used in orthopaedic biomechanics for many decades (Kayabasi and Ekici, 2008). Up to now, many finite element (FE) simulations as well as in vivo or in vitro studies have been conducted for biomechanical analyses of the lumbar spine (Kuo et al., 2010). They can also be successfully applied for the simulation of biomechanical systems (Odin et al., 2010). FE methods have become an important tool to evaluate mechanical stresses and strains in bone (Hernandez et al., 2001) and have been widely used to investigate the mechanical behaviour of bone tissue (Herrera et al., 2007). The purpose of this study is to determine the highest stress on the vertebra due to the hyperextension condition and calculate the probability of failure for the current model. The sensitivity analyses were incorporated with probabilistic analysis to support the results and verify the input random variables that are sensitive to the output parameters. The hypothesis for this study is that the pedicle is the most critical region that affects the vertebrae when the facet joints are subjected to hyperextension loading.

METHODOLOGY

A three-dimensional finite element model of a lumbar vertebra was constructed using SolidWorks software and analysed by ANSYS 11.0. The lumbar segment has five vertebrae that stack each other vertically, but a single vertebra was focused on in this study due to the similarity of analysis. So, the analysis target was the second lumbar vertebra (L2), since it seems to be responsible for bone fractures (Sances et al., 1984) and has also been reported on by Woodhouse (2003).

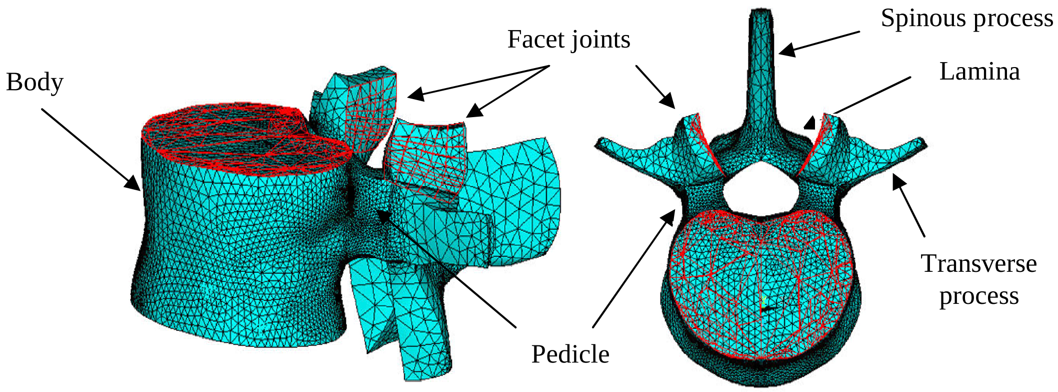


Figure 1. Anatomy of the vertebræ of the lumbar spine

The vertebrae are composed of six components. These are the vertebra body, spinous process, transverse process, lamina, pedicle, and facet joints. Figure 1 shows the anatomy of a lumbar spine vertebra from various different angles. The vertebra has two layers, of cortical and cancellous bone, which are generally considered as one integrated region of body material. In fact, the surface of the lumbar vertebra is not regular, and the simplified model was developed by removing the unnecessary surface and smoothing the irregular surface during the trimming process. Three-dimensional meshes with tetrahedral 20 node quadratic elements (SOLID186) were constructed using an automatic mesh function of ANSYS. The area of the critical region is refined using finer meshes so that reliable results are necessarily produced especially in the vertebra body.

To evaluate the effects of the hyperextension condition, a simple compressive loading was applied to the vertebral model shown in Figure 1. The lower vertebral body is fully constrained in all degrees of freedom, whereas the upper body and upper facet (indicated in red) represent the area subjected to a load based on the weight of an 80 kg person. This weight converts to a force of 460 N or 59% of total weight, to represent the upper body comprising the head, trunk and limbs, as reported by Langrana et al. (1996).

To quantitatively assess the changes of the hyperextension condition, the portion of load applied to the facet joints was calculated. The value of pressure applied to the vertebra was defined as Eq. (1):

$$\rho_{(i)} \begin{cases} F_t(1-(i/10))/A_b & \text{(body)} \\ F_t\left(i/10\right)/2A_f & \text{(facet)} \end{cases} \sum_{i=1}^{4} i \tag{1}$$

where ρ (Pa) denotes the pressure applied to the vertebra, F_t is total force, A_b and A_f are the surface area of the body and facet respectively.

Material Properties

In nature, bone is a non-linear, inhomogeneous and anisotropic material and varies in the boundary regions between cortical and cancellous bone (Xia et al., 2006; Yang et al., 2010; Peng et al., 2006). However, most studies performed in this area have been based on the assumption that bone material has an isotropic and inhomogeneous distribution of material properties due to its simplicity (Yang et al., 2010; Peng et al., 2006). Therefore, this study was conducted on linear isotropic and the whole vertebra is considered as having cortical bone properties. In this study, random input variables were arbitrarily assumed as defined in Table 1. Standard deviations were computed by assuming a coefficient of variation (COV) of 0.1 and distribution types were assumed based on experience.

Table 1. Type of model random variables

Variables	Description	Mean	COV[a]	Distribution type	Ref.
YMODCOR	Young Modulus	12 GPa	0.21	Lognormal	(Thacker et al., 2001)
PSSNRAT	Poisson ratio	0.3	±0.017	Uniform	(Sarah et al., 2007)
FORBDY	Force to the body	414 N	0.1	Normal	[b]
FORFCT	Force to the facet	46 N	0.1	Normal	[b]
AREBDY	Body area	1298 mm^2	0.1	Lognormal	[b]
AREFCT	Facet area	166 mm^2	0.1	Lognormal	[b]

[a]COV= coefficient of variation [b]Arbitrarily assumed

Reliability and Probabilistic Analysis

A probabilistic analysis was conducted of a structural failure under uncertain material and geometric characteristics subject to random loads applied to the model. ***X*,** denotes a vector of random variables, with components $X_1, X_2, ... X_n$ representing the uncertainties in the load, material properties and geometry (Akramin et al., 2007). The probabilistic design system was modelled as Eq. (2):

$$Z(X) = Z(X_1, X_2, X_3, ..., X_n) \tag{2}$$

where $Z(\boldsymbol{X})$ is a random variable describing the system (e.g. stress, displacement) at a node or element. Each random variable is defined by a probability density function (PDF), which is commonly defined by parameters such as a mean value, standard deviation and distribution type. The structural uncertainties are generated by the Latin Hypercube Sampling (LHS) technique that requires fewer simulation loops to get better accuracy. The limit state function for lumbar $g(\boldsymbol{X})$ can be expressed as Eq. (3):

$$g(X) = Y(X) - S(X) \tag{3}$$

where $Y(X)$ is the yield strength of bone, $S(X)$ is the Von Mises stress computed from FEA and X is a random variable as defined earlier. Suppose that the model failure occurs if $g \le 0$, whereas no failure occurs if $g > 0$. The probability of failure (P_f) is the likelihood when the stress exceeds the yield strength of bone or satisfies the function $g \le 0$ (Sarah et al., 2007). The probability of survival, P_s is one minus the probability of failure and referred to as reliability, $P_s = 1 - P_f$.

A MCS was performed by a powerful computer to minimize cost and time consumption. This method will converge with the approximately correct solution but needs a lot of samples during analysis. The number of simulations necessary in a MCS to provide that kind of information is usually between 50 and 200. Thus, this study used 100 samples after considering the complexity of the model and range of simulation. However, the more simulation loops you perform, the more accurate the results will be.

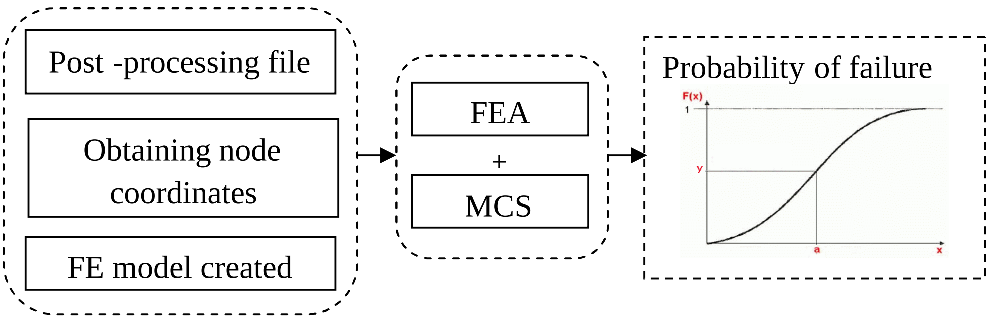


Figure 2. Work sequence of a probabilistic finite element program

The uncertainty of the mechanical properties of bone, especially the Young Modulus of vertebrae, depends on the person, since the physiological loading affects the stress distribution of the vertebra. Therefore, the PFE program has been developed using ANSYS software incorporating MCS. The work sequence of a patient-specific FEA using the ANSYS software program is shown in Figure 2.

RESULTS AND DISCUSSION

Figure 3 shows the stress distribution of the vertebra under compression loading where the contours represent the level of stress. It was found that the highest stress concentrations were at the adjacent lower posterior vertebral body, with Von Mises stress value 1.2117 MPa. Stress concentration will reduce the mechanical integrity of the bone, making it susceptible to fracture during trauma (Kasiri & Taylor, 2008). This critical area of the vertebra body tends to act as a pivot when another load is applied to the facet joints and creates a bending effect. A longer distance between the facet joints and the vertebral body causes an increase in the bending moment, as well as a stress concentration.

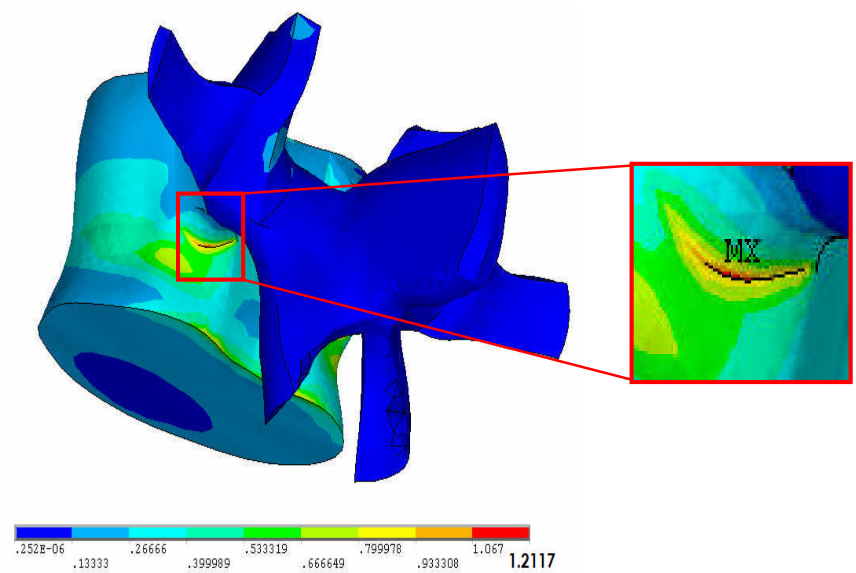


Figure 3. The highest stress distribution of the vertebra

The displacement of the model is very small, at about $0.24758e^{-08}$ mm. This happens due to the assumption that all that components act as one body with the same material, which is cortical. Cortical material is brittle compared with other materials, with the highest strength in the vertebra component. Failure or fracture of the bone starts at the highest stress concentration and it produces the weakest area of the bone. This result agrees well with research by El-Rich et al. (2009), which concluded that in extension loading, the maximum stress is located in the lower pedicle region of L2 and fractures start in the left facet joint, then expand into the lower endplate.

In Figure 4, the stress distributions for different types of ratio represent the effect of hyperextension. The comparisons between these ratios are the proportion load applied to the vertebral body and facet joints. For the ratio *i=1*, there are some stresses in the vertebral body, whilst for the ratio *i=3*, the vertebral body was not affected wholly. Hence, load ratio *i=3* means that hyperextension starts after the facet joint sustains in excess of 30% of the total load, as reported by Nabhani et al. (2002) and Hall (1995).

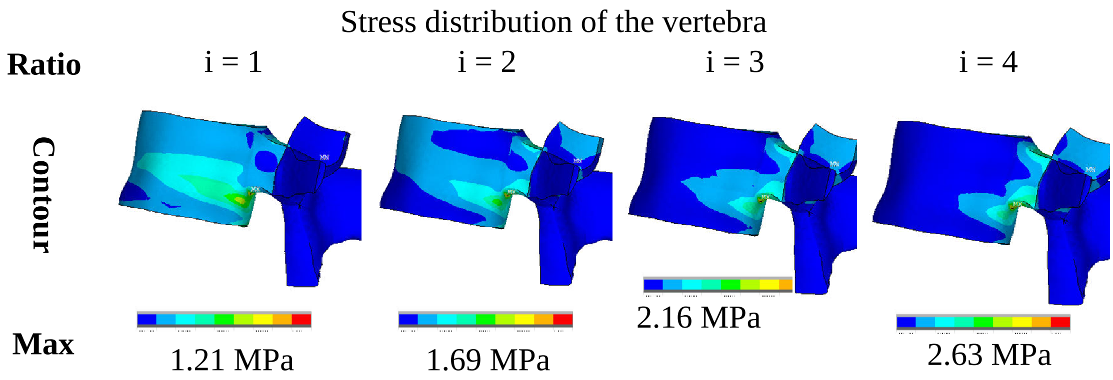


Figure 4. The stress distribution of the vertebra for different ratios

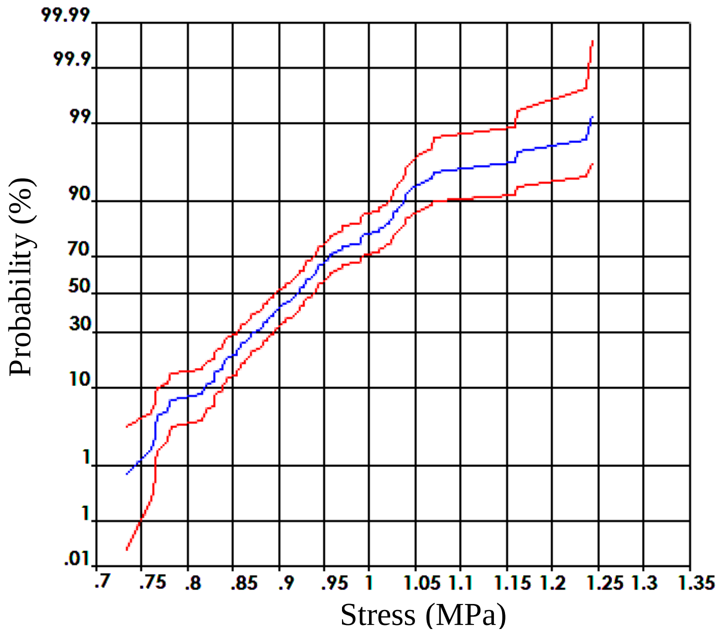


Figure 5. Probability of success for limit state function

The cumulative distribution function (CDF) offers a function to determine the probabilistic design variable. This feature is very helpful to evaluate the probability of failure or reliability of a component for a very specific and limited value given. For this study, the limit state function in Eq. (3) used the yield strength of the material as a limit value to determine the probability of failure if $g \leq 0$. The curve in Figure 5 indicates that the probability of success complies with the limit state function $g > 0$ and there is about a 97% or 0.97 probability that the stress remains below 1.2117 MPa. Therefore, the probability of failure can be calculated as 1-0.97=0.03 or 3% probability stress greater than 1.2117 MPa. From the result observation, 3% of the probability of failure indicates that the model is very reliable and safe to use. This means that the load applied to the model needs only be very low to induce the model to fail.

The probabilistic sensitivity diagrams in Figure 6 illustrate those variables that are sensitive to the maximum stress and maximum deflection respectively. The sensitivities are given as absolute values in the bar chart and the relative variables are represented in the pie chart. Four input variables are very sensitive to the stress and deflection, as shown in Figure 6. The most significant variable that strongly affected the maximum stress and maximum deflection is FORFCT, which is force applied to the facet joints. This means that a small change in the maximum stress of the important variables (AREBDY in this case) will result in a change in the computed probability. The positive sensitivity values indicate that a positive change in the mean value will result in an increase in the computed probability and negative sensitivities and vice versa. The insignificant or unimportant random variables have been eliminated from the sensitivity chart to improve the computational efficiency.

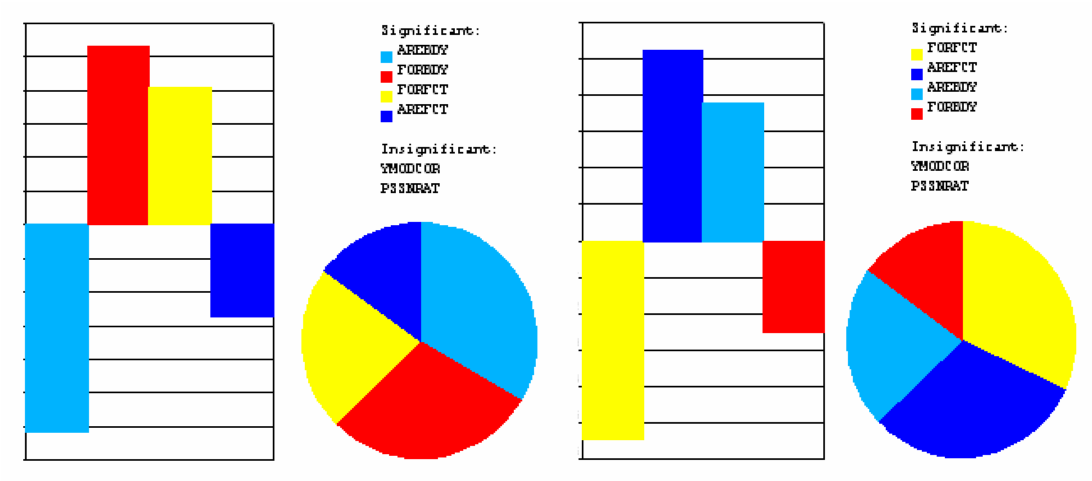
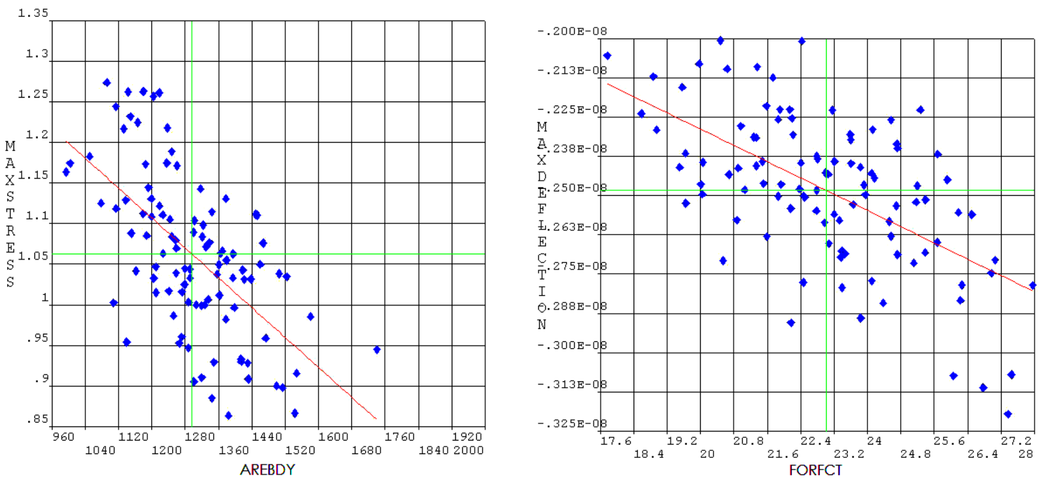


Figure 6. Sensitivity factors for (a) maximum stress and (b) maximum deflection



(a) (b)

Figure 7. Scatter plot for input variables (a) AREBDY and (b) FORFCT

The scatter plot in Figure 7(a) indicates the relationship between the AREBDY variable and maximum stress, while Figure 7(b) is between the FORFCT variable and maximum deflection. These two scatter plots represent the correlation between the input variables and output parameters that are generated by the same set. There are 100 blue dots to represent the 100 sampling points or samples that were used for this analysis. Probabilistic sensitivities measure how much the range of scatter of an output parameter is influenced by the scatter of the random input variables. The influences of probabilistic sensitivities are the slope of the gradient and the width of the scatter range of the random input variables. The slope of the gradient depends on the scatter range of the random input variables and output parameters. To improve the reliability, there are two options: 1) reduce the width of the scatter, and 2) shift the range of scatter. However, we do not discuss these options here as they are beyond the scope of this study. Since these variables contribute the most to the computed probability, improved estimates for the mean, standard deviation, and distribution will have the most impact on the computed probability (Thacker et al., 2001).

CONCLUSION

This study has achieved the objectives of determining the stress concentration and the probability of failure of the lumbar vertebra using finite element analysis. The probabilistic analysis method investigated here is useful to understand the inherent uncertainties and variations in biological structures.

ACKNOWLEDGEMENT

The author would like to thank Universiti Malaysia Pahang for supporting this research. Also, thanks to all the members who were involved in this study.

REFERENCES

Akramin, M.R.M., Abdulnaser, A., Hadi, M.S.A., Ariffin, A.K. and Mohamed, N.A.N. 2007. Probabilistic analysis of linear elastic cracked structures. *Journal of Zhejiang University SCIENCE A*, 8(11): 1795-1799.

El-Rich, M., Arnoux, P.J., Wagnac, E., Brunet, C. and Aubin, C.A. 2009. Finite element investigation of the loading rate effect on the spinal load-sharing changes under impact conditions. *Journal of Biomechanics,* 42: 1252-1262.

Hall, S.J. 2007. *Basic Biomechanics (5th edition).* New York: McGraw-Hill Companies.

Hernandez, C.J., Beaupre, G.S., Keller, T.S. and Carter, D.R. 2001. The influence of bone volume fraction and ash fraction on bone strength and modulus. *Bone,* 29: 74-78.

Herrera, A., Panisello, J.J., Ibarz, E., Cegonio, J., Puértolas, J.A. and Gracia, L. 2007. Long-term study of bone remodeling after femoral stem: A comparison between dexa and finite element simulation. *Journal of Biomechanics,* 40: 15-25.

Kasiri, S. and Taylor, D. 2008. A critical distance study of stress concentrations in bone. *Journal of Biomechanics,* 41: 603-609.

Kayabasi, O. and Ekici, B. 2008. Probabilistic design of a newly designed cemented hip prosthesis using finite element method. *Materials and Design,* 29: 963–971.

Kuo, C.S., Hu, H.T., Lin, R.M., Huang, K.Y., Lin, P.C., Zhong, Z.C. and Hseih, M.L. 2010. Biomechanical analysis of the lumbar spine on facet joint force and intradiscal pressure - a finite element study. *BMC Musculoskeletal Disorders,* 11: 1-13.

Langrana, N.A., Edwards, W.T. and Sharma, M. 1996. Biomechanical analyses of loads on the lumbar spine. In: Wiesel, S.W., Weinstein, J., Herkowitz, H., Dvorak, J. and Bell, G. *The Lumbar Spine.* Philadelphia, W.B. Saunders Company. pp. 163-181.

Nabhani, F. and Wake, M. 2002. Computer modelling and stress analysis of the lumbar spine. *Journal of Materials Processing Technology,* 127: 40-47.

Odin, G., Savoldelli, C., Bouchard, P.O. and Tillier, Y. 2010. Determination of Young's modulus of mandibular bone using inverse analysis. *Medical Engineering & Physics,* 32: 630-637.

Peng, L., Bai, J., Zeng, X. and Zhou, Y. 2006. Comparison of isotropic and orthotropic material property assignments on femoral finite element models under two loading conditions. *Medical Engineering & Physics,* 28: 227-233.

Qiu, Z.P. and Wu, D. 2010. A direct probabilistic method to solve state equations under random excitation. *Probabilistic Engineering Mechanics*, 25: 1-8.

Sances Jr., A., Myklebust, J.B., Maiman, D.J., Larson, S.J., Cusick, J.F. and Jodat, R.W. 1984. The biomechanics of spinal injuries. *Critical Reviews in Biomedical Engineering,* 11: 1-76.

Sarah, K.E., Saikat, P., Tomaszewski, P.R., Petrella, A.J., Rullkoetter, P.J. and Peter, J.L. 2007. Finite element-based probabilistic analysis tool for orthopaedic applications. *Computer Methods and Programs in Biomedicine*, 85: 32-40.

Sirvanci, M., Ulusoy, L. and Duran, C. 2002. Pedicular stress fracture in lumbar spine. *Journal of Clinical Imaging,* 26: 187-193.

Taddei, F., Martelli, S., Reggiani, B., Crictofolini, L. and Viceconti, M. 2006. Finite-element modeling of bones from CT data: sensitivity to geometry and material uncertainties. *IEEE Transactions on Biomedical Engineering,* 53: 2194-2200.

Thacker, B.H., Nicolella, D.P., Kumaresan, S., Yoganandan, N. and Pintar, F.A. 2001. Probabilistic finite element analysis of the human lower cervical spine. *Math Model Sci Computer,* 13: 12-21.

Woodhouse, D. 2003. Post-traumatic compression fracture. *Clinical Chiropractic,* 6: 67-72.

Xia, Q.T., Tan, K.W., Lee, V.S. and Teo, E.C. 2006. Investigation of thoracolumbar T12-L1 burst fracture mechanism using finite element method. *Medical Engineering & Physics*, 28: 656-664.

Yang, H., Ma, X. and Guo, T. 2010. Some factors that affect the comparison between isotropic and orthotropic inhomogeneous finite element material models of femur. *Medical Engineering & Physics*, 32: 553–560.