Biomechanical Analysis of a Three Dimensional Finite Element Model

C S Aparna, R P Tewari & A K Govil

Dept. of Applied Mechanics, MNNIT Allahabad, India
E-mail : csaparna88@gmail.com, rptewari@mnnit.ac.in, ashokkgovil@yahoo.com

Abstract - **Low back pain has for long been associated with the lumbar spine biomechanics. A three dimensional model of segments L2 to L5 of lumbar spine is generated form CT images of a healthy human male. Hounsfield units of the image slices are used to derive material properties throughout the 3D model. This model is subjected to finite element analysis to study stresses developed during naturally found loading phenomena.**

The stresses developed both for flexion and extension are found to be highest in the pedicle region. Also, while flexion produces more stress concentrations, extension balances the compressive body weight, thereby reducing the stress levels.

Keyword - Lumbar Spine, Biomechanics, Finite Element Analysis.

I. INTRODUCTION

Low back pain, a leading cause of disability, occurs uniformly across all cultures, interferes with quality of life and work performance, and is the most common reason for medical consultations [1]. Relation between degeneration of lumbar vertebrae and low back pain has been widely studied. Lumbar spine, especially its 4^{th} and 5^{th} vertebrae, is subjected to significantly greater loads than the rest of the spine. Hence it continues to receive more attention clinically and experimentally [2].

Many studies have been carried out on lumbar spine to biomechanically analyse the causes and effects of stresses on it. The experimental methods either based on cadaveric specimens or synthesized models not only pose difficulties in simulating loading conditions within the body, but also are costly and cumbersome. Hence computational models are highly sought after, especially because information that is inaccessible experimentally can be obtained through this method. Apart from that, various possibilities of normal and abnormal situations can be simulated with ease.

Finite element analysis has been found to be very useful to study anatomical models that usually have complex geometries and highly non-linear properties. This paper discusses the generation of a finite element model of four segments of lumbar vertebra based on CT image data, followed by its biomechanical analysis.

Recent times have witnessed a growing research interest in generating biomechanically viable models of lumbar as well as the rest of the spine. The major concern here is to replicate the material property and geometry of the original object as closely as possible. One widely used method is to use values of dimensions and material constants obtained from previous experimental works. The utilisation of Computed Tomography in developing finite element model facilitates accurate representation of three-dimensional geometry and heterogeneous distribution of material properties.

Tobias Pitzen etal generated an anisotropic, three-dimensional, nonlinear finite element model of an intact human L3/4 spinal segment from slices of a computed tomography and using a finite element analysis software [3]. The material properties were taken from the literature. The results of the finite element analysis were validated with an in-vitro biomechanical study on cadaveric spinal segments. It was proposed that the finite element model be used to predict the biomechanical behaviour of the spine.

M A Tyndyk etal focused on the development of a robust and efficient procedure for generating anatomical 3D FE models, directly from a series of medical images, i.e., CT data [4]. They concluded that for anatomical structures such as the vertebra, which consist of simple and complex geometry regions, the optimum method for solid model generation involves the combination of CAD and CAD-STL methods.

Janko D Jovanovic developed a model of L4-L5 segment of human vertebra using subject specific finite element modelling based on Quantitative Computed Tomography, wherein a vertebra and a special CT number calibration phantom are scanned simultaneously [5]. The relation obtained from QCT together with relation from the literature is used to assign material properties to the model.

In this work, both the geometry and the material properties of the lumbar vertebra are obtained from the CT image data. Such a model is subjected to finite element analysis to dwell into the biomechanics of lumbar spine, which shares the greatest loads, forces and moments and provides the greatest mobility within the human spine.

II. MATERIALS AND METHODOLOGY

The process adopted in this work can be divided into 3 sections as discussed below:

A. Generation of 3D model from CT images

The images in DICOM format of the CT Scan of Lumbar vertebra of an 18 year old healthy male are obtained. The scanning was done on .These CT slice images are imported into Materialise MIMICS software version 10.01. CT slices are segmented according to the Hounsfield Unit thresholds for vertebra (226-1674) and disc(50-90) obtained from literature [6]. Then individual slices are edited to reduce noise and improve the geometry. Region growing is done wherever necessary to avoid holes or gaps in masks.

The final mask is used to calculate the 3D object. A primitive surface mesh with triangular elements is automatically generated by the software. The surface mesh is of very poor quality. Here quality is measured based on height/base parameter. This parameter measures the ratio between the height and the base of a triangle and normalizes the value. A perfect equilateral triangle has a quality of 1 and a very bad triangle has a quality of 0. There are 83412 elements in the original mesh. More than half of these have a quality less than 0.4. In addition, there are many intersecting triangles, marked in FIG.2. These are removed by a sequence of processes beginning with triangle reduction. Then quality is improved with split based method. All the intersecting triangles are manually removed. This is followed by quality preserving triangle reduction. It is ensured that all the bad edges are removed. At this stage the meshing is checked for any shell formation.

B. Mesh and material properties

Finally, the 3D object is obtained with high quality mesh containing 37980 elements. Less than 20% of these elements have a quality less than 0.4. This model devoid of any intersecting triangles and shells is imported into Abaqus for volume meshing. Here, the surface mesh of the 3D object is converted into a volume mesh consisting of tetrahedral elements.

After volume meshing is done, the object is imported back to MIMICS as finite element model. The elements of the volume mesh is assigned material properties using the relationships between Hounsfield Unit values and density, and density and modulus of elasticity. These relationships for lumbar vertebra given below are determined by J Y Rho etal, using an ultrasonic transmission technique [7].

$$\rho = 47 + 1.122\ CT. \qquad (1)$$

$$E = 0.63 * \rho^{1.35}. \qquad (2)$$

Eq.1 relates the density, ρ in kg/m3 with the Hounsfield values, CT. The relationship between modulus of elasticity, E in MPa and density, ρ in kg/m3 is given by Eq. 2. The values thus obtained was confirmed to be within range obtained from literature for cortical bone, cancellous bone, disc and transverse processes[8].The 3D model constitutes of 132628 quad SOLID185 elements having 28927 nodes. It has a non-linear variation of material properties as shown in Fig.1.

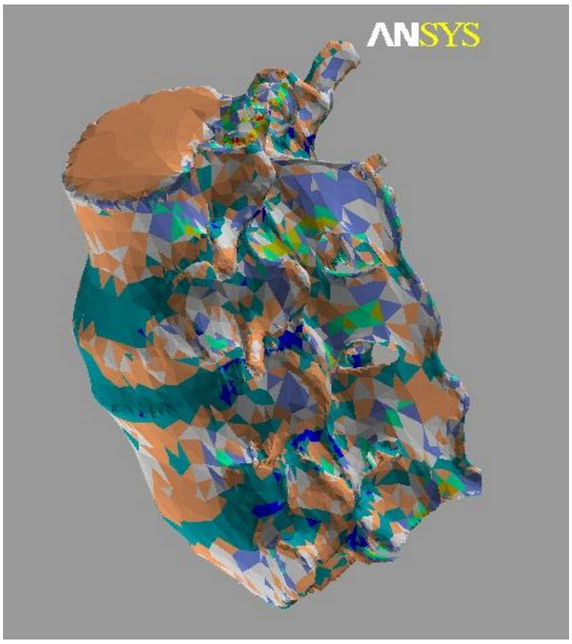

Fig. 1 : Distribution of material properties in the 3D finite element model

C. Loads and boundary conditions

The finite element analysis of this model under various loading conditions is done using ANSYS 11. The lower end of the L5 vertebra is uniformly restricted in all the three directions.

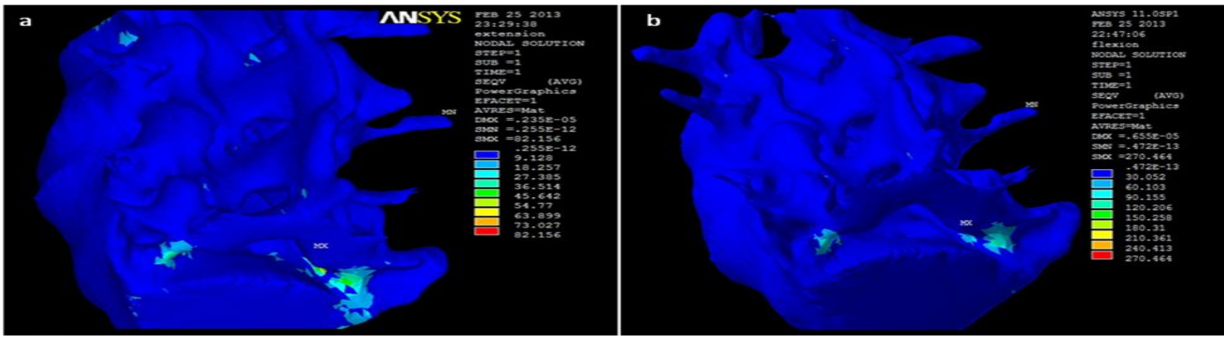

Fig. 2 : Von Mises Stress distribution in (a) extension (case 3) and (b) flexion (case 4)

Four cases are considered for simulating flexion and extension.

Case 1: A pure moment of 23 Nm applied to the uppermost plane of L2 to simulate flexion

Case 2: A pure moment of 23 Nm applied to the uppermost plane of L2 to simulate extension

Case 3: Gravity load of 1000 N applied along with the flexion moment

Case 4: Gravity load of 1000N applied along with the extension moment

III. RESULTS AND DISCUSSIONS

In all the four loading cases analysed, the pedicle region is found to bear the maximum stress. Within the pedicles, the highest stresses are found at L4-L5, as seen in Fig. 2. This inference is in agreement with past works [9]. This region is susceptible to injuries due to sudden impacts and large loads. Also, the pedicle has much smaller thickness compared to the vertebral body and the transverse processes. This geometry is conducive to high stress concentrations.

Another important observation is that though the maximum stress developed for same value of pure moment in flexion and extension is same, they are very different when applied together with the gravitational compressive load. The maximum stress values obtained are summarised in Table 1 below.

Sl. No.	Load Case	Maximum Stress [N/m^2]
1	Pure moment in flexion	173.02
2	Pure moment in extension	173.02
3	Flexion moment with gravitational load	270.46
4	Extension moment with gravitational load	82.156

Table 1. Maximum Von Mises stress in different load cases

This makes sense since extension, or straightening of muscle is associated with healthier spine, lower cases of low back pain and reduced risk of fracture. There are many works suggesting this by implicating flexion activities like sitting and bending as more likely to be associated with back pain than extension activities such as walking and standing [10].

IV. CONCLUSION

An efficient combination of methods has been employed in this work to generate the three dimensional model of the lumbar vertebra that can be utilised for finite element analysis. Its worthiness is tested by using the model to simulate the two basic motions of spine, viz., flexion and extension. The results obtained conform to the already existing knowledge on causes and effects of stresses developed in lumbar spine. This modelling technique can also be used for developing robust models for other biomechanically significant body segments.

Further work is in progress to solve stability related issues in the model of lumbar vertebrae developed here to solve instability related to buckling by including the role of muscles.

V. REFERENCES

[1] George E. Ehrlich, Low back pain**,** Bulletin of the World Health Organization 2003; 81(9):671-676, 2003

[2] Margareta Nordin, Victor H Frankel, Basic Biomechanics of the Musculoskeletal System, 3rd edition, Lippincott

[3] Tobias Pitzen etal, A finite element model for predicting the biomechanical behaviour of the human lumbar spine, Control Engineering Practice 10 (2002) 83–90, Elsevier Science Ltd, 2002

[4] M A Tyndyk etal, Generation of a finite element model of the thoracolumbar spine, Acta of Bioengineering and Biomechanics, Vol. 9, No. 1, 207

[5] Janko D. Jovanović, Finite Element Modelling Of The Vertebra With Geometry And Material Properties Retrieved From CT-Scan Data, Facta Universitatis Series: Mechanical Engineering Vol. 8, No 1, pp. 19 – 26, 2010

[6] Paul Butler etal, Applied Radiological Anatomy, Cambridge University Press, 1999

[7] J.Y. Rho etal, Relations of mechanical properties to density and CT numbers in human bone, Med. Eng. Phys. Vol. 17. No. .5. pp. 347-355, Elsevier Science Ltd., 1995

[8] Marta Kurutz, Finite Element Modelling of Human Lumbar Spine, Finite Element Analysis, David Moratal (Ed.), ISBN: 978-953-307-123-7, InTech, 2010

[9] F. Nabhani, M. Wake, Computer modelling and stress analysis of the lumbar spine, Journal of Materials Processing Technology 127, 40–47, 2002

[10] S May, D Lomas, Posture, the Lumbar spine and back pain, In: JH Stone, M Blouin, editors, International Encyclopaedia of Rehabilitation, 2013

◈◈◈