Establishment and Validation of a Computed Tomography–Based Finite Element Model of Lumbar3-5 Vertebrae

Xin Xia¹, Lu Zhang¹, Bingzhi Chen² and Wenzhi Zhao¹*
¹Department of Trauma, Dalian Medical University Second Hospital, Dalian 116023, China
²Traffic and Transportation School, Dalian Jiaotong University, Dalian 116028, China

ABSTRACT

Objective: To describe a rapid method for establishing a finite element model of the lumbar vertebrae by using computed tomography (CT) images

Methods: CT images of the L3-L5 vertebrae of a 75-year-old man were acquired and used to establish a finite element model using the Minics 10.01 and Hypermesh 11.0 softwares. Soft tissues, such as the intervertebral discs and ligaments, were manually meshed. The model thus generated was subjected to a vertical force of 400 N and a torsion force of 10 N·m to simulate the conditions of the human spine in the standing position.

Results: The CT-based finite element model of the lumbar vertebrae was successfully established, accurately reflecting the anatomy of both vertebrae and attached soft tissues. The biomechanical response of the model to the external testing forces was within the accepted normal ranges.

Conclusion: We constructed a feasible and accurate CT-based finite element model of the lumbar vertebrae with manually meshing of the soft tissue data. This model showed good consistency with in vitro experimental data. Thus, our method appeared to be useful in the construction of a finite element model of the spine.

INTRODUCTION

A variety of diseases of the spine, such as trauma injury and fractures, are best treated with surgery. With the continued expansion of the aging population, osteoporosis-related spinal fractures have also become more frequent. Both in China and abroad, an increasing number of patients are being treated with spinal surgery. Continuous advancements are being made in diagnostic and therapeutic techniques, such as vertebroplasty and motion-preserving procedures. Compared with the previous treatment method of vertebral fusion, these new treatments provide better range of motion and flexibility for the spine. However, the outcomes of these treatment methods in terms of the biomechanical changes have to be assessed thoroughly before they can be safely applied in practice. Accordingly, several methodologies have been developed to gain insight into the behavior of the spine [1] when subjected to external forces. For a long time, in vitro experimentation on cadaveric spines remained the primary mode of biomechanical studies of the spine; this method is limited by the lack of sufficient specimens and interindividual variability. On the other hand, in vivo tests of the spine under physiological conditions can provide the most reliable information on the biomechanics of the spine, but these tests are restricted by the need for minimal invasiveness and by ethical issues. Therefore, over the last decade, finite element
analysis has been increasingly gaining ground as the preferred method for biomechanical studies of the vertebrate spine.

The finite element model is a powerful mathematical tool that can be used to analyze the patterns of stress, strain, deflections, and heat transfer in computer-generated models of different materials. Without FEM, many biomechanical questions would remain unexplained and intractable [1]. Various finite element models of the spine have been established for a repertoire of clinical and basic medicine questions. Models have been specifically designed for the investigation of fracture risks in the vertebrae and degenerative pathology in the intervertebral discs. In recent years, three-dimensional finite element models of the spine have been constructed to simulate the biomechanical response to various conditions [2-4] and treatment outcomes of traumas [5]. The prerequisite for the establishment of a finite element model is accurate geometrical information that must be captured and expressed digitally such that it can be turned into a mesh of finite elements that can then be loaded by external forces. The techniques for acquiring geometrical data include manual tracing, computed tomography (CT) scanning, and magnetic resonance imaging (MRI); the collected information is then input into a computer-aided design (CAD) program, from which meshable data can be exported. Touch-probe digitizers [6] and laser scanners [7] can provide highly accurate measurements of the surface, while CT [8] and MRI [9] can elucidate the internal structures and material properties. These methods have been used in combination for the simulation of a set of vertebrae or segments of the human spine [7].

CT is particularly suitable for the modeling of the spine because of its ability to provide three-dimensional images of bone geometry and the spatial distribution of bone minerals. CT provides high inherent image contrast between bone and soft tissues, which enables clear segmentation of the bone from soft tissue and allows the generation of a geometrically accurate volumetric dataset of the examined spine. The basic concept is to overlay CT scan images representing the outline of each vertebra on the spinal axis. The cortical and trabecular bone compartments can be accurately modeled using the CT images for the three-dimensional analyses of bone structure and strength [10]. For the computational manipulation of finite element analysis, once the overall geometry of the spine is established, the model is divided into smaller finite elements, and the resulting assembly is called a mesh. The mesh consists of finite elements of various shapes, sharing nodes at each corner (and sometimes along their edges as well). Since CT images are usually divided into small cubic or cuboid volumes called voxels, voxel-based meshes are often used. These meshes directly convert the rectangular voxels into brick-shaped elements or combine multiple voxels into slightly larger brick-shaped elements. External forces causing stress (concentration of mechanical force) and strain (amount of physical deformation) on each element of the model are simulated to analyze the mechanical behavior of the spine. The bone mineral concentration of each voxel can be calculated and has a specific relationship with bone mechanical properties [11, 12]. Thus, it is quite convenient to convert a three-dimensional CT image to a three-dimensional finite element model, which makes CT-based finite element models very useful for biomechanical studies of the spine.

In this study, we first obtained the CT images of the L3-5 vertebrae from a male subject. These images and reconstructions of the intervertebral discs and ligaments were then meshed to establish a finite element model of the spine. The mechanical behavior of this model was tested and compared with previous in vitro data.
MATERIALS AND METHODS

Study Protocol

In this study, we obtained CT images of the lumbar vertebra of a 75-year-old man and prepared a finite element analysis model. The study protocol was approved by the Ethics Committee of Dalian Medical University Second hospital, and informed consent for the procedure was obtained from the subject.

CT Scanning

Plain radiographs of the subject’s spine were first obtained to rule out the presence of any deformities or lesions. Since no lesions were noted, CT images of the L3-L5 vertebrae of the subject were acquired in the axial plane by using a CT system (Aquilion 16, TOSHIBA, Japan). The following were the scan parameters: tube voltage, 120 kV; tube current, 200 mA; slice thickness, 0.5 mm; and slice interval, 0.5 mm. The CT images were saved in the DICOM format. Figure 1A shows the CT image of the L5 vertebra.

Establishment of the Infinite Element Model

The acquired CT images were then imported into the Mimics 10.01 software (Materialise, Belgium) (Figure 1B), and a rough model of the lumbar spine was established. The rough model was smoothed out according to the normal anatomic parameters (Figure 2A), by using the Remesh tool of the finite element analysis mode of the Mimics 10.10 software (Figure 2B). Closed polylines were created on the surface of the model and exported as an IGE file into the Hypermesh software (Version 11.0 Altair, US) (Figure 2C).

The meshes for the intervertebral discs between L3-4 and L4-5 were manually added into the three-dimensional model, which was imported into the SolidWorks software (Dassault Systèmes, US) (Figure 2D). The model was saved in the IGES format with 89633 nodes and 404676 elements. Further, representations for the other normal anatomic components of the spine, namely, the anterior longitudinal ligament, posterior longitudinal ligament, ligamentum flavum, interspinous ligament, supraspinal ligament, and intertransverse ligament, were also added (Figure 2E), by using only the drawn rod element simulation. Simplified processing of small joint lines, defined as only tensile rod elements, surface contact selection surface-surface contact element simulation of articular cartilage layer. Material properties reference previous literature assignment[13].

Figure 1. A, CT image of L5 vertebra in the axial view; B, CT images imported into the Mimics 10.01 software.

Figure 2. A, Rough model of the lumbar spine established in the Mimics software; B, Smoothed model by using the Remesh tool; C, Closed polylines exported as an IGE file; D, Three-dimensional model imported into the SolidWorks software; E, Model with normal anatomic components added.
Figure 2. A, Rough model of the lumbar spine established on the basis of CT images; B, Surfaces of the model were smoothed; C, Model mesh established using the Hypermesh software; D, Manual addition of meshes for intervertebral discs between L3-4 and L4-5. E, Addition of the meshes for anterior longitudinal ligament, posterior longitudinal ligament, ligamentum flavum, interspinous ligament, supraspinal ligament, and intertransverse ligament. F, Extension movement of the model.

Model Validation

The model was subjected to an axial compression force of 400 N to simulate the pressure exerted by the upper torso in the neutral position. A rigid element is established by using the surface node of the L3 vertebral body, and the torque of the 10 N. M is applied to simulate the physiological motion of the segmental spine on 6 degrees of freedom: extension, flexion, left/right bending, and torsion. (Figure 2F) shows the model under extension movement. The applied load has proved to be sufficient to produce a physiological range of motion, without causing instability in the spine [14]. The model and testing forces were setup and imported into the ANSYS 12.0 software (ANSYS, US) and analyzed. And obtain the stress cloud of lumbar spine flexion and extension, (Figure 4) and to observe the displacement of lumbar spine.

Results

The maximum stress and strain exerted during different spinal movements simulated on the model are shown in TABLE I and Figure 3. A comparison of the obtained data with those reported previously from in vitro studies showed that the biomechanical behavior of this finite element model was consistent with the reported reference ranges for the various parameters [15]. According to the above test results, compared with in vitro biomechanical results, the numerical error is in the allowable range compared with the experimental results in this experiment. It is proved that the three-dimensional finite element model is effective and can be used for further experiments.

| TABLE I. MAXIMUM STRESS (MPa) AND STRAIN (MM) OF THE FINITE MODEL. |
|-------------------|---------------------|---------------------|---------------------|---------------------|---------------------|
| Flexion | Extension | Left bending | Right bending | Torsion |
| Stress (MPa) | 2.855 | 2.863 | 2.857 | 2.861 | 3.465 |
| Strain (mm) | 2.738 | 2.748 | 2.741 | 2.746 | 2.743 |
DISCUSSION

Finite element models can be used to reconstruct spinal segments for biomechanical studies of the spine. The reconstructions of the adjacent complex soft tissues, such as the ligaments and intervertebral discs, can then be meshed to accurately simulate the biomechanical response of the spine under various loading conditions. Models prepared in this manner can be evaluated on the basis of data published from cadaveric experiments and other studies. The introduction of finite element analysis has considerably advanced the biomechanical study of the spine. However, this method is limited by the need for the simplification of the structures and material properties. Variations in the properties of different tissue materials can lead to differences in the patterns of deformation in the models. Models with the highest degree of detailing in the modeling of the bone material properties have been shown to provide the best simulation of the in vivo conditions [16]. The process of collecting detailed data on the material properties and integrating them into a model is both time consuming and labor intensive, but it is mandatory for the construction of an accurate model. Another drawback of finite element models is the need for human input when defining the nodes, elements, and loading forces, which raises the possibility of arbitrary bias or errors and compromises the accuracy of the simulation.

Finite element models of the vertebrae can be generated using geometrical data obtained from in vitro specimens or by generic assessment of the average vertebra measurements. Generic models for various purposes have been developed by measuring the anatomical features of several vertebrae [17, 18]. The geometrical data of the generic models can be used to easily regenerate the morphology of various vertebral shapes symmetrically divided at the sagittal plane. This can reduce the computational time to about half. Even if the generic geometry is relatively simple, the generation of the mesh is time-consuming and tedious. The inclusion of the soft tissues such as intervertebral discs and ligaments may further complicate the reconstruction. The
stiffness and strength of the vertebral body are determined by the specific geometrical and material properties of the specimen, and a large number of single vertebral models have been generated using data on these properties using medical images. Currently, the most common method for generating a finite element model of the vertebral body is the extraction of the segmented area and conversion of the image voxels into hexahedral mesh elements [19, 20]. The material properties of each element are determined on the basis of the density data of the image. The model is validated if good similarity is noted between the reconstructed model and the corresponding in vitro specimen. Compared to the generic vertebral models, the CT-based finite element model is limited by the need for data on the geometries of real specimens, which are often scarce, and the time-consuming nature of the computational process because of the asymmetry of the shapes. However, the cubic mesh resolution is much lower than that of the source images, thereby reducing the number of elements and the extent of computation required. The most significant advantage of converting the vertebral models from voxels is the simplicity of the mesh-generation process. To achieve this simplicity, the cortical surfaces of the vertebrae in these models are generally rough, with flat endplates. Some voxel-based models are smoothed on the surface to improve the geometric fit [21]. The initial mesh is generated from the image voxels, creating hexahedral elements. The surface is then smoothed by converting them to tetrahedral elements where necessary. With this method, it is possible to account for the anatomical cortical shell and endplate shape in specimen-specific models. Our model was established in the same manner, with surface smoothing, and satisfactory improvement in the anatomy was achieved.

The intervertebral disc is a complex structure that is non-homogenous and has anisotropic material properties. The mechanical behavior of the discs is determined by the biochemical as well as mechanical compositions. The simulation of intervertebral discs is a major challenge, and therefore, the geometry of the disc has been inevitably simplified in most of the previous studies. For example, the disc structure was assumed to be axisymmetric [22, 23] or symmetrical in the sagittal [24] or sagittal and transverse planes [23]. Further, the superior and inferior surfaces of the disc are also assumed to be flat, although some segmental models do consider a more realistic curvature [25]. The geometric information obtained by CT lacks the clear differentiation between the disc tissues evident in radiographic imaging. Compared to the finite element model of the bony spinal components, the finite element mesh for the disc can be generated in a more straight-forward manner. In this study, the material properties of the discs and ligaments were simplified, which introduces the risk of disparity between the results of the model simulation and the actual behavior of the spine under physiological conditions.

CT images provide information of the density of the tissue, which can be correlated with the material property in the finite element model. Image-based vertebral models are generated on the basis of density values determined for each element according to the degree of the image brightness. These density values are then used to derive the Young’s modulus value(s) for that element. No consensus has been reached regarding the most appropriate formulae to arrive at the density values. The material properties for most image-based models are heterogeneous throughout, and no distinction is made between the cortical shell and cancellous region. The use of a single conversion formula to determine Young’s modulus for both cancellous and cortical bone tissue is consistent with the widely accepted notion that the cortical shell in the spine is closer to the condensed trabecular bone than the cortical shell in other parts of the body. When the
cortical shell is explicitly segmented from the cancellous core and modeled as a
different material, the shell material is universally represented as isotropic, although the
constants of the material property vary. The difference between the cortical shell and
the endplate material properties are rarely acknowledged for models of the vertebra in
isolation.

The functional spinal unit, comprising two vertebrae, the interposed disc
components, facet joints, and spinal ligaments, is often used experimentally to examine
the behavior of the spine and the effect of treatment. An increasing number of finite
element models of the functional spinal unit and of larger spinal segments have been
developed for such purposes. The construction of such models is more challenging than
that of models discussed in the previous sections. The use of spinal segment models has
increased rapidly over the last five years in investigations directed towards gaining a
better understanding of the behavior of the healthy and damaged or diseased spine. This
section is not intended to include an exhaustive list of studies in this area. Segment
models have also been extensively employed to study the biomechanical effects of the
treatments, including disc replacement, anterior and posterior instrumentation, and
vertebroplasty. The development and basic constituents of the models used across these
applications have been similar; in fact, several research groups have used a single base
model with minor adaptations for various purposes. As is the case with the vertebra and
disc models, the morphology of the spinal segment is often extracted from images
acquired via CT, micro-CT, or MRI, and often, the material properties are assigned to
total regions rather than on an element-by-element basis; more recently, however,
element-specific vertebral tissue properties have been assigned. The inclusion of the
ligamentous tissue and facet joints into the segment model augments the uncertainty
regarding the accuracy of the models due to the limited experimental data available for
characterization. The ligaments play an important role in the segment behavior,
particularly in bending; therefore, further characterization of these tissues as well as a
more detailed assessment of how their functions can be accurately represented in the
segment simulation are necessary. The limited experimental data available on the facet
joint behavior has prompted the development of a number of different approaches for
simulating the facet interactions. Some researchers have explicitly assigned properties
to the cartilage layer [26], while others have incorporated its behavior into gap elements
exhibiting changes in the stiffness as the gap closes [27, 28]. The surfaces are generally
assumed to be frictionless [26, 29] or assigned a low coefficient of friction. Consistent
with these approaches, our study considers the facet joints in our model as frictionless
joint surfaces.

A newly established simulation model first needs to be validated and proved to
induce mechanical results corresponding to the actual values when subjected to various
conditions. The most convenient and most common method for the validation of a finite
element model is the comparison with in vitro experimental test results. Although in
vitro test conditions only mimic the actual physiological conditions, they are
consensually accepted as surrogates for in vivo tests for the validation of finite element
models. In vivo tests are often invasive and involve many complex, influencing factors,
thereby making the study significantly difficult. However, validating a model against in
vivo tests should be considered as the gold standard and undertaken whenever possible.
Most studies test finite element models by comparing the simulation test results and
experimental test results. Such models are developed to simulate the experiment as
closely as possible. The material properties and geometry of the model are derived
directly from the specimen; therefore, the in vitro tests are usually performed on the same specimen. In some other studies, the results of the model simulation tests are compared to the in vitro test results reported previously, as well as the results of clinical trials and data from the subjects’ medical history. In this study, we compared the results of our finite element model with previously published data and found good consistency between the two sets of results.

The higher the number of components and the degrees of freedom in the model, the more difficult is the validation of the model outputs. Various indirect methods have been described for the validation of the models. Some of the methods involve the comparison of the whole segment behavior against already published data, which may not be sufficient if the model is then used to predict local phenomena such as stress maxima. Other methods involve the prediction of the results using in-house tests on cadaveric specimens [27, 30]. In such cases, the predicted response of the segment under the same loading conditions generally falls within one standard deviation of the mean of the experimental results. Simulation of the segment behavior during full spinal motion adds further complexities to the already daunting task of modeling the individual vertebral and disc components. Researchers have used CT and/or MRI data to derive the morphology, but the segmentation of the objects in the spine from imaging data and the subsequent generation of the finite element mesh are the primary bottlenecks in the construction of subject-specific models. Several methods have been successfully employed in speeding up the segmentation of objects within the spine [31, 32]; however, further advances are necessary towards making the image-to-mesh process fully automatic. Although currently, the in vitro testing of motion segments does not fully represent the in vivo conditions, many investigators have compared model simulations with laboratory experiments. Indeed, this does appear to be the most logical step since more measurements are possible, and the boundary conditions are more controllable than those in vivo. However, the large variance in the segment behavior from specimen to specimen implies that for proper validation of the simulations, direct validation using subject-specific models is necessary. To this end, some investigations now involve specimen-specific geometry and material properties being assigned to the hard tissues according to the imaging data. However, there still remains the challenge of modeling the soft tissues. Future direction would be to adopt image-based methods to determine both specimen-specific geometric parameters and material properties to the individual components.

CONCLUSIONS

In conclusion, we established a CT-based finite element model of the lumbar vertebrae and validated the accuracy of the model. The method of using CT-based imaging data and manually constructed meshes for the soft tissues was found to be convenient and afforded an accurate model of the spine.
ACKNOWLEDGEMENT

This study was supported by National Natural Science Foundation of China (No. 31270999 to Wenzhi Zhao) and Liaoning Provincial Natural Science Foundation of China (No. 20170540298 to Lu Zhang).

The authors gratefully thank the Traffic and Transportation School, Dalian Jiaotong University for technical assistance. This study was supported by the following grants (11272070, 2011J013-E, 2009BAG12A-04, 2010DFB80050-1).

REFERENCES

1. Richmond, B.G., et al. 2005. “Finite element analysis in functional morphology,” The Anatomical Record Part A: Discoveries in Molecular, Cellular, and Evolutionary Biology, 283A(2): 259-274.
2. Little, J.P., et al. 2010. “The mechanical response of the ovine lumbar anulus fibrosus to uniaxial, biaxial and shear loads,” J. Mech Behav Biomed Mater, 3(2): 146-57.
3. Ezquerro, F., et al. 2004. “Combination of finite element modeling and optimization for the study of lumbar spine biomechanics considering the 3D thorax-pelvis orientation,” Med Eng Phys., 26(1): 11-22.
4. Wang, J.P., et al. 2006. “Finite element analysis of the spondylolysis in lumbar spine,” Biomed Mater Eng., 16(5): 301-8.
5. Polikeit, A., et al. 2003. “Factors influencing stresses in the lumbar spine after the insertion of intervertebral cages: finite element analysis,” Eur Spine J., 12(4): 413-20.
6. Takayama, Y., et al. 2009. “Finite element model based on a mandibular cast and a waxed complete denture: evaluation of the accuracy and the reproducibility of analysis,” J. Prosthodont Res, 53(1): 33-7.
7. Herrera, A., et al. 2012. “Applications of finite element simulation in orthopedic and trauma surgery,” World J Orthop., 3(4): 25-41.
8. Klinder, T., et al. 2009. “Automated model-based vertebra detection, identification, and segmentation in CT images,” Med Image Anal., 13(3): 471-82.
9. Wang, S., et al. 2014. “In vivo loads in the lumbar L3-4 disc during a weight lifting extension,” Clin Biomech (Bristol, Avon), 29(2): 155-60.
10. Lang, T.F. 2010. “Quantitative computed tomography,” Radiol Clin North Am., 48(3): 589-600.
11. Les, C.M., et al. 1994. “Estimation of material properties in the equine metacarpus with use of quantitative computed tomography,” J. Orthop Res, 12(6): 822-33.
12. Kopperdahl, D.L., E.F. Morgan, and T.M. Keaveny. 2002. “Quantitative computed tomography estimates of the mechanical properties of human vertebral trabecular bone,” J. Orthop Res, 20(4): 801-5.
13. Zhao W. Z., Su J., and Chen B. Z., et al. 2013. “The Application of Contact Theory in Finite Element Model of Human L1-L5 Lumbar Segments,” J. Applied Mechanics & Materials, 330:712-718.
14. Yamamoto I., Panjabi M.M., and Crisco T., et al. 1989. “Three-dimensional movement of the Whole lumbar spine and lumbosacral joint,” Spine, 14:1256-1260.
15. Dreischarf, M., et al. 2014. “Comparison of eight published static finite element models of the intact lumbar spine: predictive power of models improves when combined together,” J. Biomech, 47(8): 1757-66.
16. Strait, D.S., et al. 2005. “Modeling elastic properties in finite-element analysis: how much precision is needed to produce an accurate model?” Anat Rec A Discov Mol Cell Evol Biol., 283(2): 275-87.
17. Overaker, D.W., N.A. Langrana, and A.M. Cuitino, 1999. “Finite element analysis of vertebral body mechanics with a nonlinear microstructural model for the trabecular core,” J. Biomech Eng., 121(5): 542-50.
18. Higgins, K.B., et al. 2007. “Biomechanical alterations in intact osteoporotic spine due to synthetic augmentation: finite element investigation,” J. Biomech Eng., 129(4): 575-85.
19. Crawford, R.P., C.E. Cann, and T.M. Keaveny. 2003. “Finite element models predict in vitro vertebral body compressive strength better than quantitative computed tomography,” Bone, 33(4): 744-50.
20. Crawford, R.P., W.S. Rosenberg, and T.M. Keaveny. 2003. “Quantitative computed
tomography-based finite element models of the human lumbar vertebral body: effect of element size
on stiffness, damage, and fracture strength predictions,” *J. Biomech Eng.*, 125(4): 434-8.
21. Wijayathunga, V.N., et al. 2008. “Development of specimen-specific finite element models of
human vertebrae for the analysis of vertebroplasty,” Proc Inst Mech Eng H, 222(2): p. 221-8.
22. Schroeder, Y., et al. 2006. “Osmoviscoelastic finite element model of the intervertebral disc,” *Eur
Spine J.*, 15(3): S361-71.
23. Ferguson, S.J., K. Ito, and L.P. Nolte. 2004. “Fluid flow and convective transport of solutes within
the intervertebral disc,” *J. Biomech.*, 37(2): 213-21.
24. Argoubi, M. and A. Shirazi-Adl. 1996. “Poroelastic creep response analysis of a lumbar motion
segment in compression,” *J. Biomech.*, 29(10): 1331-9.
25. Schmidt, H., et al. 2006. “Application of a new calibration method for a three-dimensional finite
element model of a human lumbar annulus fibrosus,” *Clin Biomech (Bristol, Avon)*, 21(4): 337-44.
26. Williams, J.R., R.N. Natarajan, and G.B. Andersson. 2007. “Inclusion of regional poroelastic
material properties better predicts biomechanical behavior of lumbar discs subjected to dynamic
loading,” *J. Biomech.*, 40(9): 1981-7.
27. Goel, V.K., et al. 2005. “Effects of charite artificial disc on the implanted and adjacent spinal
segments mechanics using a hybrid testing protocol,” *Spine (Phila Pa 1976)*, 30(24): 2755-64.
28. Rohlmann, A., et al. 2007. “Comparison of the effects of bilateral posterior dynamic and rigid
fixation devices on the loads in the lumbar spine: a finite element analysis,” *Eur Spine J.*, 16(8):
1223-31.
29. Kim, Y. 2007. “Finite element analysis of anterior lumbar interbody fusion: threaded cylindrical cage
and pedicle screw fixation,” *Spine (Phila Pa 1976)*, 32(23): 2558-68.
30. Kumaresan, S., et al. 1999. “Finite element modeling of the cervical spine: role of intervertebral disc
under axial and eccentric loads,” *Med Eng Phys.*, 21(10): 689-700.
31. Wilcox, R.K., et al. 2004. “A dynamic investigation of the burst fracture process using a combined
experimental and finite element approach,” *Eur Spine J.*, 13(6): 481-8.
32. Schileo, E., et al. 2007. “Subject-specific finite element models can accurately predict strain levels in
long bones,” *J. Biomech.*, 40(13): 2982-9.