Patient-specific Finite Element Modeling of the Human Lumbar Motion Segment

Bian Li Ming
(B.Eng (Hons))

A DISSERTATION SUBMITTED
FOR THE DEGREE OF MASTER OF SCIENCE
GRADUATE PROGRAM IN BIOENGINEERING
NATIONAL UNIVERSITY OF SINGAPORE
JUNE, 2004
Acknowledgement

The author would like to extend his greatest gratitude towards following people:

Professor Teoh Swee Hin, the main supervisor of this thesis project, for his enlightening and instructive guidance during the process of this project.

Associate Professor Ong Sim Heng, the co-supervisor of this thesis project, for his informative advices on image processing technologies which is a crucial component of this project.

Dr. Wong Hee Kit, the co-supervisor of my thesis project, for his advices on spinal anatomy, a good understanding of which is extremely important to the construction of the spine finite element models.

Dr Yan Chye Hwang, a member of the Thesis Steering Committee, for his effective advices in elastic mapping and registration.

Associate Professor Wang Shih Chang, a member of the Thesis Steering Committee, for his insightful advices on the medical imaging of the human spine.

Mr Jeremy Teo, Mr Zhang Jin and Mr Wang Zhenlan, the author’s fellow colleagues in VSW group, for their indispensable help in various works involved in this project.

Mr Feng Wei, for his help in providing the floating license of the software MIMICS.

The project group is funded by Virtual Spine Workstation grant (Grant No.: 397-000-006-305) from the Agency for Science, Technology and Research (A*STAR), Singapore.
3.3 Material properties in FE modeling of biological structures........................................33
3.4 Conclusion.....................................................................................................................34

Chapter 4 Normal line searching TPSI method ...................................................37

4.1 Introduction ..................................................................................................................37
4.2 Materials & methods .................................................................................................38
  4.2.1 Lumbar Template FE mesh ..................................................................................38
  4.2.2 The patient-specific data ..................................................................................38
  4.2.3 Thin-plate spline interpolation ..........................................................................39
  4.2.4 Local Surface Normal ......................................................................................42
  4.2.5 Normal line searching method ..........................................................................43
  4.2.6 Pre-mapping adjustments ................................................................................47

4.3 Results of mesh mapping ..........................................................................................49
4.4 Discussion ..................................................................................................................52
4.5 Conclusion ..................................................................................................................54

Chapter 5 FE simulation of a lumbar motion segment ........................................55

5.1 Finite element model .................................................................................................55
5.2 Biomechanical responses of the lumbar MS under pure loadings ..................58
  5.2.1 Axial compression ..........................................................................................58
  5.2.1.1 Parametric study on the axial stiffness of the MS ........................................58
  5.2.1.2 Comparison with experimental results .......................................................61
  5.2.1.3 Analysis of stress distribution pattern under axial compression .............63
  5.2.2 Flexion .........................................................................................................68
  5.2.2.1 Flexural stiffness of the lumbar MS ..............................................................68
  5.2.2.2 Comparison with the experimental results .................................................69
  5.2.2.3 Analysis of the stress distribution pattern .................................................70
  5.2.3 Lateral bending ...............................................................................................71

5.3 Conclusion ..................................................................................................................72

Chapter 6 Recommendations & Conclusions ......................................................74

6.1 Recommendations ....................................................................................................74
  6.1.1 Patient-specific geometry ...............................................................................74
  6.1.2 Patient specific material property ..................................................................75

6.2 Conclusion ..................................................................................................................76

References ......................................................................................................................78

Appendix ..........................................................................................................................83
List of figures

Fig 2.1 Lateral (side) view of a normal spine. The drawing shows the locations of the five major spinal levels. The cervical region has seven vertebrae (C1 through C7), the thoracic region has 12 vertebrae (T1 through T12) and the lumbar region has five vertebrae (L1 through L5). The sacral region consists of five vertebrae, all fused together to form one continuous bone mass known as the sacrum. The coccygeal region consists of four vertebrae, all fused together to form the coccyx or tailbone (http://www.espine.com/normal_anatomy.html)

Fig 2.2 Detailed lateral (side) view of a segment of three lumbar vertebrae (http://www.espine.com/normal_anatomy.html)

Fig 2.3 Detailed top and lateral (side) view of a lumbar vertebra.

Fig 2.4 A segment of lumbar spine revealing the internal structure of an intervertebral disc and spinal nerve system. (http://www.dcdoctor.com/pages/rightpages_healthconditions/yourspine)

Fig 2.5 Detailed anatomy of the intervertebral disc (courtesy from the Slide Library of the Orthopaedic Surgery Department of the National University Hospital)

Fig 2.6 Organization of the collagen fibrils in the annulus lamella (courtesy from the Slide Library of the Orthopaedic Surgery Department of the National University Hospital)

Fig 2.7 The nucleus pulposus gradually loses its water content and gel-like property as the disc ages. Left picture shows an adolescent disc; the middle one is a disc in its 2nd to 3rd decade; the right one is a disc of seriously aged spine. (Courtesy from the Slide Library of the Orthopaedic Surgery Department of the National University Hospital)

Fig 2.8 Distribution of forces in the normal and abnormal disc. A, when the disc functions normally, as in the early decades of life, the nucleus distributes the forces of compression and tension equally to all parts of annulus. B, with degeneration, the nucleus no longer functions as a perfect gel. Now the forces distributed to the annulus are less and not equal. C, with advanced degeneration of the nucleus, the distribution of forces to the annulus is completely lost. (Anthony, 1970)

Fig 2.9 The first model of an intervertebral disc and adjacent vertebrae developed by Belytschko et al. in 1974. The model is axisymmetric with a horizontal plane of symmetry. (Belytschko et al., 1974)

Fig 2.10 The three-dimensional non-linear model of the L2–L3 disc body unit developed by Shirazi-Adl et al. in 1983. Because of symmetry, only a quarter of the joint was modelled, with symmetry about the sagittal plane and mid-horizontal plane. (Shirazi-Adl et al., 1984)

Fig 2.11 An exploded view of a typical motion segment model showing the vertebrae, end plates and intervertebral disc. (Wang et al., 1997)

Fig 2.12 A full three-dimensional finite element model of the L2–L3 motion segment showing typical ligament attachments included in many of today’s models (Lu et al., 1996).

Fig 2.13 Finite element model of two motion segments from the lumbar spine which includes both ligaments and muscles. (Goel et al., 1993)

Fig 2.14 Finite element model of human lumbar spine section from L2 to L4 with spinal ligaments (Smit, 1996)
Fig 3.1 A geometrically accurate model of a vertebra generated directly from CT data (Fagan, 2002) 28
Fig 3.2 Voxel mesh of a C4 vertebra obtained directly from CT data, where each voxel was converted directly into a finite element. (Bozic et al., 1994) 30
Fig 3.3 Smoothened voxel mesh of a femur head used for contact analysis (Nicole, 2003) 30
Fig 3.4 Finite element model of a human skull base, a) original voxel mesh with discontinuous surface, b) mesh surface smoothened using the centroid-based algorithm. (Daniel et al., 1997) 30
Fig 3.5 A reference MRI dataset was manually segmented first (upper left). The template segments (lower left picture) were then registered to a volunteer image (lower right) to generate a volunteer 3-D image. Differences can be seen in the shape of the tibial plateau and femoral condyle as indicated. The resultant mapping function is used to transform the template mesh into a patient-specific mesh (upper right). 32
Fig 3.6 Superimposition of the reference 3D mesh (grey) with the 3D surface target points (black) from one femur b) 3D mesh of the volunteer’s femur (shaded grey) generated from the reference 3D mesh (wire frame). 32
Fig 3.7 Application of mesh-match algorithm in generation of patient-specific facial soft tissue FE models (left: reference mesh before mapping; right: new mesh generated after mapping) 33
Fig 4.1 a) the templates lumbar vertebra mesh b) the template mesh with the surface normal vectors of each surface node represented as short truss elements 38
Fig 4.2 (a) A thin steel plate tacked at 4 points. (b) Application of TPSI in a 2-D interpolation problem. 39
Fig 4.3 Creating local unit vector surface normal from all neighboring vertices. 43
Fig 4.4 Normal line searching 46
Fig 4.5 Iterative local adjustment of the template mesh 47
Fig 4.6 mesh generate for a pig vertebra. Left, central and right column are the top, side and perspective view of the target pig vertebra geometry, the template or template mesh; and the transformed mesh, respectively. 50
Fig 4.7 Mesh generate for a human L3 vertebra. Left, central and right column are the top, side and perspective view of the target L3 vertebra geometry, template mesh and the transformed mesh, respectively. 51
Fig 4.8 The results of the element shape correction procedure 51
Fig 5.1 First and second row are different views of segmented lumbar motion segment and its FE model constructed, respectively. 57
Fig 5.2 The effect of Poisson’s ratio of the disc nucleus on the axial stiffness of the lumbar motion segment 60
Fig 5.3 The effect of Young’s Modulus of the disc nucleus on the axial stiffness of the lumbar motion segment 61
Fig 5.5 Von Mises stress distribution in the superior vertebral cortical shell of the motion segment model with a) a healthy disc nucleus or b) a degenerated disc nucleus (red, yellow, green and blue colors in order represent stress of highest value to lowest value, following figures follow the same convention)  

Fig 5.6 Comparison of maximum Von Mises stress observed in the superior vertebral cortical shell under axial compression  

Fig 5.7 Medial sagittal section view of the Von Mises stress distribution in the cortical shell of the Degenerated model under axial compression load of 3000N.  

Fig 5.8 Medial sagittal section view of the Von Mises stress distribution in the cancellous core of a) the Normal model (the disc nucleus is not shown) and b) the Degenerated model under axial compression load of 3000N. (the grey color is used to represent the stress values which are higher than the stress value represented by the red color)  

Fig 5.9 Tensile principal stress distribution in the lower endplate of the superior vertebra  

Fig 5.10 The effects of dysfunctional collagen fibrils of the disc annulus on the motion segment flexural stiffness.  

Fig 5.11 Medial sagittal section view of the Von Mises stress distribution in the cancellous core and posterior elements of the Degenerated model a) with spinal ligaments and b) without spinal ligaments when the motion segment is subjected to flexural bending moment. (the grey color is used to represent the stress values which are higher than the stress value represented by the red color)  

Fig 5.12 Comparison of leftward and rightward lateral bending stiffness of scoliotic motion segment. (L/R-NL: leftward/rightward bending without spinal ligaments; L/R-Lig: leftward/rightward bending with spinal ligaments)
List of tables

Table 3.1 Comparison of different meshing techniques 36

Table 5.1 The material properties assigned to various spinal structures in the Finite element model. 57
Summary

Finite Element Method (FEM) has been widely used in the field of orthopedic biomechanics to investigate biomechanical behavior of human spine, especially the lumbar region of the spine, which is associated with various forms of spinal pathologies. Patient-specificity of a finite element model of the lumbar spine in both geometry and material property proves crucial to the study of the pathogenesis of lumbar spinal disorders and to the enhancement of the simulation realism of medical surgeries on spine as well. The objectives of this dissertation project are: firstly, to develop a method capable of rapidly creating FE meshes with patient-specific geometry; secondly, to study the unique biomechanical response of a lumbar motion segment as a result of its special geometry.

After a thorough review on the previous mesh generation methods, this dissertation proposes a novel method, named Normal Line Searching TPSI Method. This method can be used to rapidly generate FE meshes of human lumbar spine motion segments with accurate geometry by mapping a template mesh onto patient-specific geometry derived from analysis of medical images, such as CT or MR images. Successful mapping of human and pig lumbar vertebrae demonstrated the robustness and versatility of this method. The major advantages of the proposed mesh mapping method are: mapping accuracy onto complex geometry is very good; element organization and mesh simplicity of the template mesh is inherited in the generated meshes after mapping; and the time needed for new mesh generation is short. Compared with a benchmark algorithm in performing 3D mesh mapping, the octree-spline mesh-matching algorithm, the proposed method is capable of accurately mapping elongated geometrical features, such as the processes of the lumbar vertebrae.
Using the proposed method, a lumbar spine motion segment (MS) including two lumbar vertebrae and one intervertebral disc (IVD) was constructed based on a human CT dataset. It was found that the scoliotic symptom manifested by the geometry of the lumbar MS has a significant impact on its biomechanical responses under pure loadings. One phenomenon is that the cortical shell of the lumbar vertebra at the scoliotic concave side sustains higher stress than that at the other side when the MS is subjected to axial compression. Furthermore, the tensile principal stress distribution in the lumbar vertebra was found to be modified by the geometrical abnormality of the lumbar vertebrae as a result of the extensive osteophyte formation. The mechanics of the IVD was also investigated in this study in order to improve the patient-specificity of the FE model.

To conclude, the works presented in this dissertation successfully achieved the objectives of this M.Sc project.
Chapter 1  Introduction

1.1 Background

The human spine is a composite anatomical structure including bone, muscle, ligaments, nerve system and a plethora of soft tissues. Spinal diseases, especially those associated with the lumbar spine are prevalent and inflict heavy social and economical losses. As many as 85% of adults experience back pain that disrupt their work or leisure activities and 25% of the people between the ages of 30-50 years report low back symptoms (Frymoyer, 1990). Results of epidemiology studies have associated degenerative changes in the lumbar section of the human spine resulted from frequent bending twisting, lifting, sudden violent incidents and physical heavy work, etc. to low back pain symptoms. Many other types of spinal pathology such as Spondylolisthesis, Lordosis and Osteoarthritis also occur at the lumbar region of the spine.

Determining the mechanical behavior of the lumbar motion segment is crucial to the study of pathogenesis of various spinal disorders. The experimental and computational simulations have been utilized to investigate the biomechanical behavior of the lumbar motion segment. The advantage of experimental study is that the data collected closely reflect the real characteristics of the tested anatomy structures provided that the testing conditions and test sample preparation procedures are verified to render no bias to the testing results. However, the validity of the experimental testing results is undermined by the number of samples tested due to difficulties in obtaining cadaver specimens. A more serious problem is that experimental study fails to provide insight into the internal stress/strain fields within the structure tested. Computational simulations like finite element analysis are able to delineate the internal stress/strain field of the tested object.
The conduction of computational simulations is not affected by the availability of the test specimens. In addition, different property parameters of the test anatomical structure, such as size, shape, elasticity, etc. can be varied and simulated to study the effects. Nevertheless, the computational models need to be validated by experimental results so as to ensure the accuracy of their results.

1.2 Finite Element Analysis/Method (FEA/FEM)

The terms “finite element method” (FEM) and “finite element analysis” (FEA) seem to be used interchangeably in most documentations. However, there is a difference, albeit a subtle one.

The finite element method is a mathematical method for solving ordinary and elliptic partial differential equations via a piecewise polynomial interpolation scheme. Put simply, FEM approximates the solution of a differential equation by using a number of polynomial curves. Each polynomial in the solution can be represented by a number of points and so FEM evaluates the solution at the points only. The points are known as node points or nodes.

FEA is an implementation of FEM to solve a specific problem. For example if we were intending to solve a 2D heat transfer problem. For the FEM mathematical solution, we would probably use the differential equation that governs heat conduction. In addition to this, a suitable type element with linear or higher order polynomials needs to be selected. Using a piecewise polynomial solution to solve the underlying differential equation is FEM, while applying the specifics of element formulation is FEA.
1.3 Finite Element meshing

In order to carry out a finite element analysis, the modeling domain, a 1D, 2D or 3D space must be divided into a number of small pieces known as finite elements. Since the model is divided into a number of discrete parts, FEA can be described as a discretization technique.

1.3.1 Mesh Density

A very important aspect of using FEM lies in choosing the correct mesh density according to the nature of the problems to be solved and the ultimate objectives of the solving the problems. If the mesh is too coarse, then the element will not allow a correct quantitative solution to be obtained but can provide qualitative predictions. Alternatively, if the mesh is too fine, the cost of analysis in computing time can be out of proportion to the level of accuracy needed.

1.3.2 Element Distortion

Every type of elements has its most ideal shape which gives accurate results. Due to the geometry of the modeled domain, elements may become distorted in an effort to force a mesh to comply with the boundary of the model. When elements are distorted from their ideal shape they become less accurate. Therefore, the shape of the elements should be kept as near to the ideal element shape as possible when creating a mesh.

1.3.3 Element Limitations

Triangles and tetrahedra can fit irregular boundaries and allow an adaptive and progressive change of element size without excessive distortion. There are fully automatic methods for generating triangular and tetrahedral meshes. However, linear tetrahedra are not desirable for
FEM because of over-stiffening effects and a high density of elements also leads to prolonged computation time. Quadratic quadrilateral and hexahedral elements are much more accurate elements for FEM. However, it is difficult to automatically generate all-hexahedral meshes.

1.4 Application of FEA in Biomedical Engineering

Finite element analysis was firstly developed 1943 by R. Courant. Early application of FEA was limited to aeronautics, automotive, defense, and nuclear industries due to expensive computational facilities required. Because of the rapid development in computer technologies the scope of FEA application has been dramatically broadened to numerous disciplines. One area in which FEA has been extremely useful is the field of orthopaedic biomechanics because of the advantages discussed in previous section.

However, a major limitation of FEA in studying the mechanics of biological structures is the difficulty in generating patient-specific FE models. This difficulty arises because biological structures can have significant variations in their geometrical shape and material property among different individuals. Even to a single individual, the characteristics of one biological structure changes with respect to age, time, pathological conditions, etc. for instance, an osteoporosis patient’s trabecular bone becomes more porous and weaker than his/her trabecular bone when he/she is at a younger age and free from osteoporosis symptom.

In the case of building FE models for lumbar motion segment, because of the complex shape of the lumbar vertebra and variations among patients an efficient and effective mesh generation method is needed in order to create FE mesh with accurate geometry for each patient. Besides
accurate representation of the geometry assigning patient-specific material property to the FE
model of human lumbar motion segment is another important issue.

1.5 Objectives

The objectives of this project are:

1. Develop a novel mesh mapping method which is capable of rapidly generating
   geometrically accurate hexahedral type FE meshes from a template mesh for a human
   lumbar spine motion segment based on medical images.

2. Perform FE simulations on a human lumbar motion segment model built with proposed
   method to investigate the unique biomechanical response of a lumbar motion segment as
   a result of its special geometry.

1.6 Overview

This dissertation consists of 6 chapters. Following this Introduction Chapter, Chapter 2 gives a
concise description on the fundamentals of spine anatomy which is important to the
understanding of the subsequently presented works. Literature reviews on the finite element
modeling of the lumbar motion segment and intervertebral disc is also presented in Chapter 2. In
chapter 3 previous studies in finite element meshing techniques are reviewed and compared.

Chapter 4 and Chapter 5 present the completed works aiming at achieving the objectives of this
dissertation project. Chapter 4 elaborates on the proposed novel mesh mapping method which
achieves the first objective of this master project. Chapter 5 describes the construction of a finite
element model for a lumbar motion segment using the proposed mesh mapping method and reports the results of a comprehensive finite element study on the constructed model, which is the second objective of this study.

Lastly, Chapter 6 concludes the dissertation and gives recommendations for the future works as well.
Chapter 2  Literature review on lumbar spine modeling

2.1  Brief introduction to spinal anatomy

The human spine is a very complicated musculoskeletal structure containing various soft and hard tissues. This chapter presents some important aspects of the human spine anatomy which are essential to the proper understanding of the following chapters of this dissertation. Section 2.1.1 gives a detailed description on the anatomical composition of the vertebral column. The anatomical features of lumbar vertebra are described in Section 2.1.2. In section 2.1.3 the function of the intervertebral disc (IVD) are explained and the effects of the IVD degenerations are discussed.

2.1.1  The vertebral column

The vertebral column or spinal column is a composite anatomical structure made of a string of 33 individual bones, each known as a vertebra, connected by a mass of soft tissue called intervertebral disc (Fig 2.1, Fig 2.2). The vertebral column is also the attachments site of various spinal muscles and ligaments which provide the structure stability to the entire vertebral column. The spinal canal located in the posterior region of the vertebral column functions as a protective shell of the delicate spinal chords inside (Fig 2.2).

According to the geometrical features of the vertebrae, the vertebral column are divided into five sections, namely, the cervical, thoracic, lumbar, sacrum and coccygeal regions (Fig 2.1). There are 7, 12 and 5 vertebrae in the cervical, thoracic and lumbar regions,
respectively. The sacrum actually consists of 5 fused vertebrae and the coccygeal is made of 4 fused vertebrae.

Fig 2.1 Lateral (side) view of a normal spine. The drawing shows the locations of the five major spinal levels. The cervical region has seven vertebrae (C1 through C7), the thoracic region has 12 vertebrae (T1 through T12) and the lumbar region has five vertebrae (L1 through L5). The sacral region consists of five vertebrae, all fused together to form one continuous bone mass known as the sacrum. The coccygeal region consists of four vertebrae, all fused together to form the coccyx or tailbone (http://www.espine.com/normal_anatomy.html)

Fig 2.2 Detailed lateral (side) view of a segment of three lumbar vertebrae (http://www.espine.com/normal_anatomy.html)
2.1.2 The lumbar vertebra

The vertebrae in different sections of the spine have distinctively different geometrical shapes and hence biomechanical behaviors. The lumbar section of the human spine has been under the focus of intensive research because it is the main load-bearing region of the entire vertebral column and its abnormality contributes to the development of an array of the pathological symptoms, such as low back pain.

The lumbar vertebrae can be divided into two major parts, which are the anterior vertebral body and the posterior elements (Fig 2.3).

Vertebral body: A lumbar vertebral body has an elliptical cross-section in axial plane except at the posterior cavity which accommodates the spinal chord. The size of the axial cross-sections is smaller at the mid-height of the vertebral body and gradually increases towards both inferior and superior facet of the vertebral body resulting in an overall hourglass shape in sagittal plane. The lumbar vertebral body is made of a porous cancellous bone core enclosed in a dense cortical bone shell which has much higher stiffness than the cancellous bone inside. The cortical shell extends above and below the superior and inferior surfaces of the vertebral body to form rims.

Posterior elements: “posterior elements” is a general term used to refer to the remaining components of the lumbar vertebra attached posteriorly to the vertebral body. These components include:
**Pedicles:** connect the lamina to the upper part of the vertebral body

**Lamina:** is a flat plate acting as the outer wall the spinal canal

**Transverse process:** extend laterally from the junction of the lamina and pedicles and provide attachment site for the intertransverse ligaments and muscles.

**Spinous process:** protrudes posteriorly from the middle of lamina and provides attachment site for the supraspinous and interspinous ligament.

**Superior/inferior articular facet:** the inferior articular facets of the superior lumbar vertebra and the superior articular facets of the inferior lumbar vertebra form two synovial type joints, called Zygapophyseal or Apophyseal Joints.

---

Fig 2.3 Detailed top and lateral (side) view of a lumbar vertebra.

(http://www.fpnotebook.com/ORT101.htm)
2.1.3 The intervertebral disc (IVD)

2.1.3.1 Overall anatomy

A lumbar intervertebral disc connects two adjacent lumbar vertebral bodies and acts as an elastic cushion to absorb shock and facilitate spinal column movements. A normal lumbar intervertebral disc consists of a soft gel-like core called Nucleus Pulposus (NP), a tough outer ring named Annulus Fibrosis (AF) surrounding the NP, and two endplates covering the superior and inferior surface of the intervertebral disc (Fig 2.5).

Fig 2.4 A segment of lumbar spine revealing the internal structure of an intervertebral disc and spinal nerve system. (http://www.dcdoctor.com/pages/rightpages_healthconditions/yourspine)

Fig 2.5 Detailed anatomy of the intervertebral disc (courtesy from the Slide Library of the Orthopaedic Surgery Department of the National University Hospital)
2.1.3.2 The Nucleus Pulposus

The nucleus consists of a highly hydrated gel of proteoglycans containing some collagen. Proteoglycans possess negatively charged sulfate group which give rise to high osmotic pressure. As the result of water attraction by osmosis the NP has much higher water content (up to 80% of its total weight) than the AF leading to swelling pressure in the NP being twice as great as that of the AF. The swelling pressure and high water content enables the NP to transform compressive force applied on the IVD into hydrostatic pressure and distribute it equally to all sides of the annulus which can produce tangential stress (hoop stress) to counter-balance the hydrostatic pressure applied.

2.1.3.3 The Annulus Fibrosus

The annulus fibrous is formed of concentric lamella of collagen fibrils embedded in proteoglycan ground substance (Fig 2.6). In each lamella the collagen fibrils are parallel and tilted with respect to the axis of the spine (60-70°). The direction of tilt alternates in successive lamellas. Outer lamellae contain large proportion of type I collagen attaching to the vertebrae and provide strength during bending and twisting. Inner lamellae contain predominantly type II collagen merging into endplates and provide circumferential stress to balance the swelling pressure of the nucleus. Tensile stiffness of the collagen fibrils increases radially from inner region to outer region.
2.1.3.4 The Endplates

As a component of cartilaginous joints of the spine, the endplates intimately related to both the nucleus pulposus and the anulus fibrosus. Some researchers consider the endplate to be part of the vertebral body, whereas others believe it to be part of the intervertebral disc. In childhood, the opposing vertebral bodies are completely covered by thin plates of cartilage. After puberty, the periphery of the cartilaginous plates ossifies and fuses with the primary bone. The central part remains cartilaginous. Thus, in adult, the bony endplates are covered centrally by thin cartilage endplates up to 1mm thick. The surrounding rim of bone, up to 1 cm in width forms the major site of attachment of AF to the bone.

The nutrient supply to the intervertebral disc may depend on diffusion of fluid from the marrow of the vertebral bodies across the subchondral bony endplate and cartilaginous endplate.

Fig 2.6 Organization of the collegan fibrils in the annulus lamella (courtesy from the Slide Library of the Othorpaedic Surgery Department of the National University Hospital)
2.1.3.5 Disc Degenerations

As the IVD ages, the NP becomes less elastic and more fibrous because of an increase in disorganized collagen and decrease in protein-polysaccharide complex (Fig 2.7). Because of these biochemical changes the NP loses its water content and gel characteristics and its ability to transform axial pressure evenly to all parts of the annulus is impaired (Fig 2.8). The annulus therefore is subjected to abnormal stress and shows evidence of degenerations. Although the aged disc is still capable to transmitting loads induced by routine daily activities it has become vulnerable to sudden and violent loads. Disc pathologies accelerate the pace of biochemical changes and the deterioration of is gel characteristics significantly (Anthony, 1970).

![Fig 2.7 The nucleus pulposus gradually loses its water content and gel-like property as the disc ages. Left picture shows an adolescent disc; the middle one is a disc in its 2nd to 3rd decade; the right one is a disc of seriously aged spine. (Courtesy from the Slide Library of the Orthopaedic Surgery Department of the National University Hospital)]
Chapter 2

2.2 Review on lumbar spine modeling

The basic component of the lumbar spine section is the lumbar motion segment or functional spinal unit. A lumbar motion segment consists of a vertebra, a disc, and another vertebra connected by the appropriate muscles, ligaments and other soft tissue. Many research efforts have been devoted into FE modeling of the lumbar motion segment.

2.2.1 Previous finite element models

The first finite element analysis of a spinal motion segment was conducted by Belytschko et al. in 1974 (Belytschko, 1974) (Fig 2.9). The modeling was based on the assumption of axial symmetry and linear orthotropic material properties for the disc. The same model
was later experimented with varied disc material properties in order to match the simulation results with the experimental measurements.

From another perspective, Spiker (Spiker, 1982) implemented a parametric study on a simplified motion segment model. They took into consideration a range of values for the geometrical and material properties to investigate the effects of different parameters on the disc’s response to loadings.

A major development in IV disc and motion segment modeling is Shirazi’s 3-dimensional non-linear finite element model of the L2-3 disc boy unit (Shirazi, 1984) (Fig 2.10). In this model, various anatomical structures, like cortical bone, cancellous bone, bony endplates and IV disc components were represented altogether. The disc’s nucleus was modeled as an incompressible fluid cavity, while the annulus was modeled as layers of fiber elements embedded in the ground substance. Because of large displacement and strain experienced by the model during loading, non-linear geometry and non-linear material solution was used in the analysis. The basic model was later further modified to include the facet joints which were modeled as contact problem (Shirazi, 1991).

Simon et al. reported a poroelastic model of an invertebral disc in 1985(Simon et al., 1985). The characteristic of this poroelatic model is that when a static load is applied the fluid phase can move with respect to the solid phase and therefore is squeezed out leaving the remaining solid phase to support the static load. On the other hand, when an instantaneous short-term load is applied the disc behaves as if it is incompressible. It was
reported that the predictions on the internal disc pressure given by this poroelastic model agreed well with experimental measurements (Duncan, 1997; Martinez, 1997).

Wang et al. investigated the time-dependant response of the lumbar disc. The model created by them exemplifies a typical construction of a motion segment finite element model including vertebrae, endplates and an intervertebral disc (Fig 2.11).

The spinal ligaments are normally included in the motion segment model. Non-compression type elements are usually adopted for the simulation of the ligaments. One example is the model of one motion segment by Lu et al. which included the anterior and posterior longitudinal ligaments, the intertransverse ligaments and interspinous ligaments, the capsular ligaments, the ligamentum flavum and the supraspinous ligament (Lu et al., 1996) (Fig 2.12). The first model which considers the effect of the spinal muscle forces was proposed by Goel et al. Their results indicated that the inclusion of the muscle forces affected the translation and rotation of the motion segments and decrease the intradiscal pressure while increasing the load taken by the facet joints (Goel et al., 1993) (Fig 2.13).

Smit (Smit, 1996) created a highly geometrically accurate finite element model for a section a human lumbar spine including 3 vertebrae and 2 intervertebral discs to study the remodeling behaviors of the trabecular bone structures (Fig 2.14). The model consists of well-shaped and well-organized elements which efficiently captured the major geometrical features of the human lumbar spine.
Besides axial compression load, the effects of flexion, extension, lateral bending and torsion were also investigated in finite element modeling. Lu et al. confirmed that disc prolapse was more likely to occur under combined compression and bending at the junction between posterior inner annulus and the endplates (Lu et al., 1996). Tan examined the response of the disc under various loading schemes and concluded that the nucleus plays an important role in resisting axial compression while the annulus collagen fibers are essential in providing resistance against bending loads, such as flexion, extension and torsion (Tan, 1998).

Fig 2.9 The first model of an intervertebral disc and adjacent vertebrae developed by Belytschko et al. in 1974. The model is axisymmetric with a horizontal plane of symmetry. (Belytschko et al, 1974)
Fig 2.10 The three-dimensional non-linear model of the L2–L3 disc body unit developed by Shirazi-Adl et al. in 1983. Because of symmetry, only a quarter of the joint was modelled, with symmetry about the sagittal plane and mid-horizontal plane. (Shirazi-Adl et al., 1984)

Fig 2.11 An exploded view of a typical motion segment model showing the vertebrae, end plates and intervertebral disc. (Wang et al., 1997)
Fig 2.12 A full three-dimensional finite element model of the L2–L3 motion segment showing typical ligament attachments included in many of today’s models (Lu et al., 1996).

Fig 2.13 Finite element model of two motion segments from the lumbar spine which includes both ligaments and muscles. (Goel et al., 1993)
Fig 2.14 Finite element model of human lumbar spine section from L2 to L4 with spinal ligaments (Smit, 1996)

2.2.2 Discussion & conclusion

The models presented in the previous section were developed to suit for different research objectives. The model by Belytschko (Belytschko et al., 1974) and the first Shirazi model (Shirazi, 1984) distinguished different vertebral bone structures, i.e. cortical bone and cancellous bone. However, the model was built with the assumption of axis-symmetry of the spine and did not include the posterior elements which are an integral part of the spine anatomy. Therefore these two models are not a complete reconstruction of the lumbar spine anatomy and cannot reveal the overall lumbar spine mechanical response under loadings. Shirazi’s second model (Shirazi, 1986) and models used by Wang (Wang et al., 1997), Lu (Lu et al., 1996) and Goel (Goel et al., 1993) all included the posterior elements and spinal ligaments which all plays significant roles in loading sharing and distribution of the lumbar spine. However, the posterior elements in
Shirazi’s second model and Wang’s model were represented with simplified geometry and limited number of elements.

Smit’s model was chosen as the template mesh for mesh mapping operation because the model possesses realistic geometry, well-organized elements and minimum amount of distorted elements. The model’s realistic geometry reduces the amount of the pre-mapping processing works. The distinctive organization of the elements in the Smit’s mesh, which are important in modeling different anatomical components of the spine, can be inherited in the children meshed generated from mesh-mapping. The minimum number of distorted elements in the Smit model ensures that effects of element distortion in the template mesh on element shape regularity of the children meshes are kept at a trivial level.
Chapter 3 Methods in building patient-specific FE models

3.1 Introduction

Finite Element Method (FEM) has been widely used in biomedical engineering researches, especially in the fields of orthopedics and injury mechanics. There have been extensive investigations utilizing FEM in studying the biomechanical behaviors, stress and strain distribution patterns of the various musculoskeletal structures such as spine motion segment (Shirazi, 1984, 1986) and mandible (Vollmer, 2000) and prostheses like bone implant (Skinner et al., 1994; Kuiper et al., 1996).

Unfortunately the studies utilizing FEM are usually limited by the number of meshes due to the daunting amount of effort required to build a FE mesh for each modeling subject. As a result, many previous researches adopted generic FE meshes based on averaged measurements of sampled subject geometries. The material properties used in these generic models are usually based on established standard values which have been adopted over years.

The concomitant drawback of these generic models is that they are unable to give accurate analysis results at a patient-specific level. Firstly, this is because the geometrical shape of an anatomical structure usually varies from individual to individual and from healthy to abnormal conditions of the same individual. Previous studies have demonstrated that the influence of the geometrical differences on the FEM output can be
significant (Candadai, 1992). As a result, the accuracies of the results obtained from these generic models were usually undermined by the lack of patient-specificity in the geometrical shapes of the anatomical structures. Secondly and probably more significantly, since the materials properties used in the generic models are only empirical values the modeling results cannot be interpreted as the exact behaviors of any individual subject even if the model has the exact geometrical appearance as the specific subject.

Because of these limitations of the generic models, there have been many previous researches aiming at improving the subject specificity of the FE modeling. In this paper these past works will be presented according to their methodologies. The advantages and disadvantages of each method will be discussed. Lastly, the issue of material property input in FE modeling of biological structures will also be discussed.

3.2 Meshing patient-specific geometry

Two classical approaches are employed to build volumetric FE mesh with patient-specific geometry, manual meshing and automatic meshing.

3.2.1 Manual meshing

Manually building a mesh is the best choice when time permits because one can create a mesh with the more desirable type of element, hexahedral elements. From the computational biomechanics point of view, hexahedral elements are preferred over tetrahedral elements because tetrahedral elements can lead to artificial anisotropy, i.e.
false stiffness in certain directions, inside the mesh and create overstressed areas. At the same time the mesh creator has a lot of control over the mesh organization and can easily classify elements into different groups corresponding to different anatomical components. However manual mesh is an extremely complex, tedious and time-consuming process hence normally limited to one specimen due to the prohibitive amount of manual works required to match each subject’s geometrical morphology.

3.2.2 Automatic meshing

In view of the limitations of manual meshing a number of automatic meshing techniques have been developed. The geometrical information of the modeling subject is obtained either through the use of the coordinate-measuring machines or three-dimensional digitizer, or directly from medical scan images. The advantage of automatic meshing is its speed and geometry accuracy. Meshes which conform to different surface geometries can be quickly generated.

3.2.2.1 Mesh built from medical images

Most of the automatic meshing techniques use CT images to provide the geometry of the mesh. Various edge-detection algorithms have been developed to delineate the external contours of the meshing subject on
the CT images, a process called segmentation. The same CAD softwares and FEA meshing tools mentioned in the previous section can also be used to create meshes based on geometry data extracted from segmented CT images. The meshes generated can be hexahedral element based and can have regular mesh elements. However, defining the geometry entities in the “bottom-up” approach is usually very time-consuming.

There are now software packages which can directly convert the segmented CT data into FE meshes. One type of mesh generated in this way is made up of tiny tetrahedral elements, for instance a mesh of a vertebra (Fig 3.1). This type of meshes has highly accurate surface morphology because the tetrahedral elements are very adaptive and flexible in forming volumetric meshes with very complicated geometrical shapes. However, the tetrahedral meshes may have singular regions with extremely high density of elements. This not only increases the computational burden but also induces over-stiffening and locking behaviors (Hughes, 1987) and leads to overstressed area (Ansys, 1999).
One improved automatic meshing technique is automated generation of voxel meshes from a stack of medical images, such as CT images (Keyak, 1990). One example is a voxel mesh of a cervical vertebra (Fig 3.2, Bozic et al.). The meshes generated consist of a large quantity of tiny cubic hexahedral elements, or so-called voxels, with their size defined by the spatial resolution of the image stack. The advantage of this approach in modeling of hard tissues like bones is that material properties, such as the apparent bone density, Young’s modulus, of each voxel can be derived from the CT images (usually by empirical relationships of the type derived by Carter and Hayes, 1977). Nevertheless, due to the large quantity of the voxels, the computation is usually very time-consuming. Furthermore, voxel meshes have an abrupt step-like surface morphology which leads to inaccuracy in the computation of surface strains (Jacobs et al., 1993). Consequently voxel meshes are not suitable for analysis of contact problem without mesh surface smoothening. Attempts were made on smoothening the step-like surface of the voxel mesh by adjusting the position of the surface nodes so as to reduce the inaccuracy (Fig 3.3, Nicole, 2003, Fig 3.4 Daniel et al., 1997). However, smoothening all the internal interfaces in between different anatomical components, which usually possess different material properties, remain a difficult task because of complex multi-material boundary conditions. In addition like the tetrahedral meshes, the voxel meshes are usually poorly organized and do not allow anatomical structures to be differentiated from another within the mesh (Chabanas, 2003), especially for soft tissues like muscles and ligaments which are slender and curved.
Fig 3.2 Voxel mesh of a C4 vertebra obtained directly from CT data, where each voxel was converted directly into a finite element. (Bozic et al., 1994)

Fig 3.3 Smoothened voxel mesh of a femur head used for contact analysis (Nicole, 2003)

Fig 3.4 Finite element model of a human skull base, a) original voxel mesh with discontinuous surface, b) mesh surface smoothened using the centroid-based algorithm. (Daniel et al., 1997)
3.2.2.2 Mesh built through registration of medical images

Because of the limitations of the automatically generated meshes, efforts were directed into developing mathematical algorithms to transform a standard reference mesh into different subject geometries. The transformation is usually achieved by applying elastic mapping algorithms and hence an automated or semi-automated process. Elaborate works can be devoted into building an optimal reference mesh with well-shaped and organized hexahedral elements. One approach is to transform a reference mesh or template mesh using mapping functions derived from registering reference images to subject images based on MR image volumes (Fig 3.5 McCarthy, 2002). The reference mesh was built based on one manually segmented MRI dataset which is later used for registration with other patients’ images. This method may not work well if the difference between the reference mesh and the target mesh are significant because of difficulties in performing registration.

3.2.2.3 Octree-spline Non-rigid deformation mesh-match algorithm

Szeliski developed a free-form deformation algorithm with hierarchical multi-resolution representation of deformation spline (Lavallee et al., 1995, 1996; Szeliski et al., 1996). This algorithm was later adopted by Beatrice Couteau et al. to map a standard reference mesh of a femur head into 3D surface segmented from CT images (Fig 3.6, Beatrice Couteau et al., 2000). Another application of this algorithm is in generating patient-specific FE meshes of facial soft tissues (Fig 3.7, Chabanas, 2003). This mesh-matching algorithm employs adaptive octree spline resolution, which increases near the matching surface, to minimize the squared distances between two surfaces. Therefore the mapping
process takes only minutes to finish. However, as a surface-based rather feature-based approach this algorithm produces less satisfactory results when mapping elongated anatomical features, such as in the vertebrae. Furthermore, pre-mapping alignments and adjustments of the template and target surfaces are needed before mapping. Therefore it is not a fully automated procedure.

Fig 3.5 A reference MRI dataset was manually segmented first (upper left). The template segments (lower left picture) were then registered to a volunteer image (lower right) to generate a volunteer 3-D image. Differences can be seen in the shape of the tibial plateau and femoral condyle as indicated. The resultant mapping function is used to transform the template mesh into a patient-specific mesh (upper right).

Fig 3.6 Superimposition of the reference 3D mesh (grey) with the 3D surface target points (black) from one femur b) 3D mesh of the volunteer’s femur (shaded grey) generated from the reference 3D mesh (wire frame).
3.3 Material properties in FE modeling of biological structures

Another type of input information which is crucial to the accuracy of the FEM simulation is the material property. Biological structures, no matter hard tissue or soft tissue, usually have heterogeneous, anisotropic material properties. The same type of inter-individual and time-dependant variations in material properties also exist as those in the geometrical morphologies. These characteristics of the biological material properties make their acquisition, input into FEM and subsequent validation extremely difficult and complex.

Obtaining material properties for biological tissues, especially soft tissues are difficult and in vitro experiments only provide general attributes of the tissue materials based on limited number of samples. Empirical correlations have been derived in an attempt to estimate the material properties, like the bone density and Young’s modulus of hard tissues from CT images (Carter and Hayes, 1977) based on the pixel intensity of the images. However MRI which is a much better imaging tool for soft tissues has a
completely different imaging mechanism compared with CT. Therefore except water content in a qualitative level, other information on tissue material property is difficult to be ascertained from MR images.

Input of material property to FE model is another tricky task. Mesh built with relatively large elements like those meshes generated through elastic mapping cannot have different material properties assigned to different parts of one large element hence has a poor material property resolution. Apparently, this type of mesh is not suitable for studies, which require detailed material property input, like bone mechanics modeling (HoBaTho, 2003). The voxel mesh has a much higher material property resolution. However, this comes at the expense of extended computational time. Moreover, voxel mesh is not useful for modeling of soft tissues such as muscles and ligaments. This is because material properties of soft tissues are difficult to be defined and therefore obtained at voxel detail level.

As a result, while there are a number of successful methods which can efficiently generate meshes with patient-specific geometry, assigning patient-specific material properties is still an ill-defined and complicated problem.

3.4 Conclusion

Accurate FE modeling of biological structures requires incorporation of accurate patient-specific information on both the geometry and material property of the modeling subject. Rapid construction of the FE mesh with accurate representation of the modeling subject’s
geometrical shape proves to be very important considering the prevalent variations in geometrical shapes among different subjects. To summarize a table was created to compare the advantages and disadvantages of various mesh generation methods (Table 3.1). Medical imaging methods like MRI, CT integrated with image processing techniques are currently the most often used platform to extract geometry information of anatomical structures from patients. Based on this platform geometrically accurate FE meshes can be generated quickly through various computer algorithms. Among these meshes, voxel mesh is used extensively for bone structure modeling because it enables high resolution material property input into the FE model. Tetrahedral meshes are useful when accurate representation of the complicated subject geometry is important to the success of the FE modeling. Meshes generated from a reference mesh through elastic mapping algorithms are more useful in modeling of soft tissues. This is because of that their well organized mesh structures and desirable element shapes provide users with better control over tissue components definition. This type of mesh proves particularly useful when the modeling subject is a composite and relatively large anatomical structure containing many tissue components with distinct material property and biomechanical behavior, such as a section of spine, which contain not only hard tissue like bone, but also soft tissues like muscles, ligaments and intervertebral discs.

While many methods have been developed to generate FE models with patient-specific geometrical accuracy, acquisition, input and validation of patient-specific tissue, especially soft tissue, material properties is still lack of effective methods. Therefore appropriate assumptions need to be made on the material property input depending on the
objective of the FE analysis. Because of these assumptions supplementary validation of FE analysis results through experimental tests are extremely important. Nevertheless despite its limitations FEM is still currently the most useful computational prediction and analysis tool in various biomedical researches.

Table 3.1 Comparison of different meshing techniques

<table>
<thead>
<tr>
<th>Methods</th>
<th>Advantages</th>
<th>Disadvantages</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manual meshing</td>
<td>- geometrically accurate</td>
<td>- time-consuming</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- desirable element type</td>
<td>- not suitable to generate large number of meshes</td>
<td></td>
</tr>
<tr>
<td>Meshing using digitizer</td>
<td>- geometrically accurate</td>
<td>- not able to mesh object in vivo</td>
<td>Teo et al., 1994</td>
</tr>
<tr>
<td>Meshing using CAD software</td>
<td>- geometrically accurate</td>
<td>- tedious in generating mesh with desirable element type</td>
<td>HoBaTho, 2003</td>
</tr>
<tr>
<td>3D meshing</td>
<td>- very fast</td>
<td>- high element density</td>
<td>Fagan, 2002</td>
</tr>
<tr>
<td></td>
<td>- capable of representing complex geometry accurately</td>
<td>- extended computation hour</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- possible though huge effort is needed</td>
<td>- tetrahedral element induce false stiffness leading to less accurate results</td>
<td></td>
</tr>
<tr>
<td>Voxel meshing</td>
<td>- fast</td>
<td>- step-like surface</td>
<td>Bozic et al., 1994</td>
</tr>
<tr>
<td></td>
<td>- desirable hexahedral elements</td>
<td>- not suitable for surface stress analysis unless having the surface smoothened</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- because of uniformly shaped elements possible to reduce computation time by developing new FE formulation</td>
<td>- huge number of elements resulting in long computation time</td>
<td></td>
</tr>
<tr>
<td>Registration based</td>
<td>- high degree of automation</td>
<td>- mapping quality dependant on the quality of segmentation</td>
<td>McCarthy, 2002</td>
</tr>
<tr>
<td>Octree-spline deformation</td>
<td>- fast</td>
<td>- less satisfactory mapping quality at elongated geometry</td>
<td>Beatrice Couteau et al., 2000</td>
</tr>
<tr>
<td>TPSI</td>
<td>- easy implementation</td>
<td>- semi-automatic</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- able to map difficult geometry</td>
<td>- relatively long computation time</td>
<td></td>
</tr>
</tbody>
</table>
Chapter 4 Normal line searching TPSI method

4.1 Introduction

Application of Finite Element Analysis (FEA) in studying stress and strain analysis in orthopaedic biomechanics has been limited by difficulties in quickly generating 3D meshes. This is because the geometrical shape of an anatomical structure varies from individual to individual. The author has developed a novel method using Thin-Plate Spline Interpolation (TPSI) algorithm to map a standard 3D hexahedral mesh to patient-specific geometries of anatomical structures using the lumbar spine as an example. The difficulties in performing the TPSI lie in locating correspondence points on the template mesh and the subject surface. This was previously done by manual matching the pre-selected markers (Evans et al, 1991). The method proposed in this dissertation aims at enhancing the degree of automation in locating the correspondence point as well as increasing the quantity of the correspondence points in order to improve the overall quality and efficiency of the elastic mapping. The capability of this method is demonstrated in the dissertation by mapping a template mesh of a human lumbar vertebra into subject-specific vertebra geometries obtained from Computed Tomography (CT) images. For the first time, such a mapping was performed on vertebral bodies which are considered a challenge because of the very complicated geometry.
4.2 Materials & methods

4.2.1 Lumbar Template FE mesh

The approach proposed in this dissertation requires at least one template mesh for subsequent generation of subject-specific meshes through elastic transformation. The template mesh used in the dissertation is a hexahedral FE mesh of a human lumbar vertebra and was derived from modification of a FE mesh originally developed by Smit (Smit, 1996). The lumbar template mesh is shown in Fig 4.1. It contains 1164 hexahedron elements and 1678 nodes. This mesh includes elements representing the cortical shell and vertebral bony endplates, which are not present in the original Smit mesh. The template mesh surface is referred to as the template surface.

![Template Lumbar Vertebra Mesh](image1)

**Fig 4.1 a) the templates lumbar vertebra mesh b) the template mesh with the surface normal vectors of each surface node represented as short truss elements**

4.2.2 The patient-specific data

The patient-specific lumbar vertebra surface geometry, referred to as target surface, is made up of a 3D point cloud, which was extracted from a segmented human lumbar spine CT dataset using the software Mimics V6.3 (Mimics, Leuven). The subject patient has a
deformed lumbar vertebra with significant osteophyte outgrowth at the lateral and anterior rim region of the vertebral body. Another set of data is a pig vertebra CT dataset. The pig vertebra has a very different overall geometry compared with the template mesh that was built on a human lumbar vertebra. These excessive geometrical irregularities or differences put the proposed method to a stringent test.

4.2.3 Thin-plate spline interpolation

Thin-plate spline interpolation algorithm is used in this study to map the surface of the template FE mesh onto the target surface. Bookstein et al. (Bookstein et al., 1989) first proposed the use of thin plate spline interpolation for point-based registration and applied this method to 2-D images. With an elegant algebraic formulation expressing the dependence of the physical bending energy of a thin metal plate on point constraints (Fig 4.2, a), this algorithm minimizes the bending energy (Eqn 4.1) of the thin plate splines. Therefore when used to solve 2-D or 3-D interpolation problems it yields a relatively smooth interpolated surface with local deformations (Fig 4.2, b). The mapping also ensures that the internal nodes of the FE mesh are moved at interpolated distances so that the shape of the elements remains FEA compliant.

Fig 4.2 (a) A thin steel plate tacked at 4 points. (b) Application of TPSI in a 2-D interpolation problem.
The extension of this method to the 3-D case with arbitrary sets of base points is as follows. Suppose in the original image, there is a set of \( n \) base points:

\[
P_1 = (x_1, y_1, z_1), ..., P_n = (x_n, y_n, z_n)
\]

In the reference image, there is the corresponding set:

\[
V_1 = (x'_1, y'_1, z'_1), ..., V_n = (x'_n, y'_n, z'_n)
\]

The interpolated shift for each base point in the original image is calculated using the following equation.

\[
f(x, y, z) = a_1 + a_x x + a_y y + a_z z + \sum_{i=1}^{n} w_i U(|P_i - (x, y, z)|)
\]  \hspace{1cm} (Eqn 4.2)

There will be one equation for each of the transformation along the \( x \), \( y \), \( z \) axes. In equation (Eqn 4.2), \( a_1 + a_x x + a_y y + a_z z \) is actually the affine transformation portion of the entire transformation. The thin-plate spline interpolation is represented by the remaining portion \( \sum_{i=1}^{n} w_i U(|P_i - (x, y, z)|) \). The function \( U(r) \) is the fundamental solution of the biharmonic equation which minimizes the “bending energy”. In the case of a 3D interpolation problem, it is in the form of \( U(r) = r(r = \sqrt{x^2 + y^2 + z^2}) \). The portion \( \sum_{i=1}^{n} w_i U(|P_i - (x, y, z)|) \) represents the weighted sum of the distances of the point \((x, y, z)\) to every base point \( P_i \) in the image. Coefficients \( a \) and \( w \) in equation are unknowns and to be determined. There are three sets of \( a \) and \( w \), one for each of the three coordinate transformations.
The matrix form of the above equation can be defined by following matrices.

\[ P = \begin{bmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ \vdots & \vdots & \vdots & \vdots \\ 1 & x_n & y_n & z_n \end{bmatrix}; \quad K = \begin{bmatrix} 0 & U(r_{12}) & \cdots & U(r_{1n}) \\ U(r_{21}) & 0 & \cdots & U(r_{2n}) \\ \vdots & \vdots & \ddots & \vdots \\ U(r_{n1}) & U(r_{n2}) & \cdots & 0 \end{bmatrix} \]

\[ V = \begin{bmatrix} x'_1 & x'_2 & \cdots & x'_n \\ y'_1 & y'_2 & \cdots & y'_n \\ z'_1 & z'_2 & \cdots & z'_n \end{bmatrix}^T; \quad W = \begin{bmatrix} w_{x1} & \cdots & w_{xn} \\ w_{y1} & \cdots & w_{yn} \\ w_{z1} & \cdots & w_{zn} \end{bmatrix}^T; \quad A = \begin{bmatrix} a_{x1} & a_{xx} & a_{xy} & a_{xz} \\ a_{y1} & a_{yx} & a_{yy} & a_{yz} \\ a_{z1} & a_{zx} & a_{zy} & a_{zz} \end{bmatrix} \]

Using above matrices Eqn 4.3 can be formulated. Eqn 4.4 represents the boundary conditions and constrains the elastic portion of the transformation to zero at infinity distance.

\[ KW + PA = V \quad \text{(Eqn 4.3)} \]

\[ P^T W = 0 \quad \text{(Eqn 4.4)} \]

Eqn 4.5 is derived by combining Eqn 4.3 and Eqn 4.4 into one matrix equation, in which \( O \) is a zero matrix. Based on Eqn 4.5 and Eqn 4.6, Eqn 4.7 can be derived to calculate the coefficient matrices \( W \) and \( A \).

\[ L = \begin{bmatrix} K & P \\ P^T & O \end{bmatrix} \quad \text{(Eqn 4.5)} \]

\[ Y = \begin{bmatrix} V \\ O \\ O \end{bmatrix}^T \quad \text{(Eqn 4.6)} \]

resulting in

\[ L^{-1} Y = \begin{bmatrix} W \\ A \end{bmatrix} \quad \text{(Eqn 4.7)} \]
After obtaining all the coefficients needed, Eqn 4.2 can be used to interpolate all the points in the image.

Previous applications of thin-plate spline interpolation used a set of manually selected reference points or so-called markers to derive the transformation matrix. As a result, the matching between the template surface and the target surface can only be achieved at the locations of the markers. Furthermore, manual selection and pairing of the reference points and their correspondence points is inefficient and might give rise to inaccuracies.

In this study the author used a normal line searching technique to automatically locate the correspondence points for each surface nodes of the template FE mesh.

### 4.2.4 Local Surface Normal

Firstly the local unit vector surface normal is found. Each consecutive pair of central vertex-neighboring vertex $A$, central vertex-neighbor vertex $B$ vectors is taken and used to form the vector product (Fig 4.3). The vector sum of these vectors is scaled to unit length to create. By initially taking the sum of normal vectors before rescaling to unity, the sum is made relatively robust; the smaller a particular central vertex-neighboring vertex $A$-neighboring vertex $B$ triangle is, the more poorly conditioned is the estimate of normal direction, but this normal will contribute less towards the sum of normal vectors.
4.2.5 Normal line searching method

In this method, the surface normal vector of each surface node (base point) in the template mesh is calculated. Because the surface contour of the template mesh and the patient-specific data is similar at most of the locations it is assumed that the correspondence point of each surface node on the template mesh lies in the vicinity of the straight line which coincides with its surface normal vector. This assumption is not valid at the locations where the template and target surfaces have very different contour profiles. Hence pre-mapping adjustments and alignments of the template mesh with respect to the target surface are needed. This is to reduce the total area of the template mesh surface which has dissimilar contours compared with the target surface to a minimal level. Normally there will still be some places with mismatching contours after pre-mapping processing. The surface nodes at these locations of the template mesh should be excluded from the nodes used for normal line searching.
For a surface node $A$ on the surface of the template mesh, the procedure to identify its correspondence point $A'$ from the target surface point cloud is illustrated in Fig 4.4a. In order to qualify the scope of the search for $A'$ and reduce the computational time, an initial global search is carried out: all the points on the target surface which are within a prescribed distance ($d_1$) from $A$ are selected out and named as point set $A_i$ (*both square and round dots*). The points in $A_i$ which are within a prescribed distance ($d_2$) from the surface normal line of $A$ are again selected out and named as point set $A_j$ (*the round dots*). The projection of the vector leading from $A$ to each point in $A_j$ ($V$) on the surface normal vector of $A$ is calculated and named as $P$. If there are positive values of $P$ (both negative and positive values exist or all are positive), the three $A_j$ points ($A_{1j}$, $A_{2j}$, $A_{3j}$) with the largest positive projection are identified and the geometrical center of these 3 points (mean of the coordinates of the three points) is set as the correspondence point $A'$, if there are no positive $P$ value (the target surface lies within the template mesh surface), the 3 $A_j$ points with smallest magnitude value of $P$ are identified and geometrical center of these 3 points is set as the correspondence point $A'$. The calculation and utilization of projection values here is to ensure that the template mesh surface is correctly expanded or shrunk to match the target surface. The value of $d_1$ and $d_2$ are set depending on the density of the target surface point cloud and the mapping accuracy required. Smaller value of $d_1$ means smaller 3D space searched for the correspondence point and hence shorter computational time. Consequently the use of small $d_1$ begets the possibility of omitting the correspondence point after the initial global search. On the other hand, smaller value of $d_2$ means that the $A_j$ points and the $A'$ derived are closer to the surface normal vector of $A$ resulting in more accurate mapping. However if $d_2$ is too small there may not be
sufficient quantity of $A_j$ points which are important to give a good estimation of the $A'$ position. Therefore the value of $d_1$ and $d_2$ need to be decided by trial and error so as to optimize the mapping quality while keeping the computational time short.

However, the above normal line searching procedure may fail to locate the correct corresponding point in some special cases where there is an abrupt change in the surface curvature, for example, the vertebral body rims which demarcate the horizontal and side surface of the vertebral body. As shown in Fig 4.4b for a node $E$ located at the vertebral rim the correct corresponding point $E'$ may be omitted either from both point set $A_i$ and $A_j$ or from point set $A_j$ because $d_1$ or $d_2$ may be too small. Consequently, a wrong correspondence point $E_0$ is erroneously identified. In order to avoid this type of errors in locating the correspondence point, a different normal line search procedure is established for the nodes located at sharp edges of the template mesh surface as shown in Fig 4.4b for these nodes, a sufficiently large $d_1$ value is used to define the point set $A_j$ which now contains more points on the target surface further away from the normal vector. Then similarly as done in the previous procedure the projection of the vector leading from $E$ to each point in $A_j$ on the surface normal vector of $E$ is calculated. The $A_j$ point with largest positive project is identified as the correct corresponding point $E'$. This procedure works because the correct corresponding point of a template surface node at a sharp edge normally has the largest positive projection on the surface normal vector as evidenced in Fig 4.4b. Although this procedure requires more computational time to locate the corresponding point for one template surface node because of a bigger point set $A_j$ the
The overall computational burden inflicted on the entire mesh mapping process is not significant because the number of template surface nodes is limited.

The normal line searching method was executed by a self-developed Matlab coded program (Appendix A1).

Fig 4.4 Normal line searching
4.2.6 Pre-mapping adjustments

To a large extent the quality of pre-mapping adjustments and alignments of the template mesh determines the quality of the final mapping. The adjustments include initial positioning with strategically located control points, local shrinking, and iterative alignment with averaged transformation vector. The first adjustment is to position the template mesh according to the target surface. This is done by a small-scale thin-plate spline interpolation with a few strategically located control points, such as the tips of the posterior processes. If the template mesh has a much larger size in certain location as compared with the target vertebrae surface, the template mesh is scaled down locally to be slightly smaller than the target. At those elongated locations of the vertebra, such as the lateral processes, the template mesh is locally aligned iteratively by shifting the processes by an averaged transformation vector. The averaged transformation vector is the vector sum of all transformation vectors calculated for each surface node at the
processes region (Fig 4.5) (executed by Matlab coded program, Appendix A2). This is to ensure that when the template mesh is later expanded to map the target surface the template mesh elements are equally enlarged in all directions so as to maintain the regularity of the elements. Because all the details on the nodes and elements of the template mesh are pre-defined and not changing throughout the pre-mapping processing, these pre-mapping processing can be done in a standardized manner and hence be finished in a short period.

5.2.7 Mesh element correction

In order to check for distortion we used a commercial software FE package, ABAQUS V6.3 (ABAQUS). Abaqus checks the distortion of each element through the angle between isoparametric lines of the element. Classically, the criterion of this test consists in verifying whether the angle is greater than 45° or less than 135° in order to reduce the influence of the element distortion on the accuracy of the numerical integration (Zienkiewicz, 1989). If the angle is found outside this range of values a warning message is declared for the element. Despite many previous attempts, perfect and thorough correction of a set of elements inside a 3D mesh is a complex, ill-posed problem without any straightforward solution (Cannan et al., 1993; Freitag and Plassmann, 1999).

In this study we used a simple but effective method of reducing the number of distorted elements in the transformed mesh. The surface nodes of the mesh were kept unmoved while the internal nodes were shifted to a position that minimizes the sum of the Euclidean distances ($E$, Equ 5.8) from the node to its 6 neighboring nodes ($p_i$). The
coordinate of this new position \( (m) \) has been proven to be equal to the mean of the coordinates of all the neighboring nodes. The shifting of the internal nodes was done in an iterative manner until the positions of the nodes stabilize.

\[
E = \sum_{i=1}^{n} \| p_i - m \|^2, \quad (x_{m}, y_{m}, z_{m}) = \frac{1}{n} \sum_{i=1}^{n} (x_i, y_i, z_i) \quad \text{(Equ 4.8)}
\]

### 4.3 Results of mesh mapping

Application of the proposed method to a human lumbar vertebra and a pig vertebra showed successful capture of the complicated bone surface morphology (Fig 4.6-7). The external surfaces of the transformed meshes match closely with those of the segmented vertebrae. Although large portion of the elements in the transformed meshes are regular, the percentages of distorted elements are higher than those of the template mesh. The distorted elements are defined by the default criteria of Verify Mesh function of Abaqus/CAE 6.3-1. Here, the criteria include: face corner angle less than 30 degree, face corner angle greater than 140 degree and aspect ratio greater than 3. The percentages of elements in the template mesh which do not satisfy the above three criteria are 0%, 9% and 25% respectively. After the element shape correction procedure, the percentages of the distorted elements defined by all three criteria dropped significantly (Fig 4.8).
Fig 4.6 mesh generate for a pig vertebra. Left, central and right column are the top, side and perspective view of the target pig vertebra geometry, the template or template mesh; and the transformed mesh, respectively.
Fig 4.7 Mesh generate for a human L3 vertebra. Left, central and right column are the top, side and perspective view of the target L3 vertebra geometry, template mesh and the transformed mesh, respectively.

Fig 4.8 The results of the element shape correction procedure
4.4 Discussion

The proposed method essentially contains elements from both the feature-based and surface-base approach which are the two major methodologies in mapping 3D surfaces. Because the proposed method also enables volumetric deformation while achieving 3D surface matching it proves to be an effective semi-automatic FE mesh generation tool. The geometry of the generated meshes matches their targets’ geometry closely. The mesh surfaces are relatively smooth and suitable for surface stress and strain study. As compared with the Octree-spline mesh-match method, presently the benchmark algorithm in performing non-rigid mapping of 3D surfaces, the proposed method showed improved mesh-mapping quality with elongated geometrical features which are prevalent in lumbar spine. There was so far no previous published study using vertebrae that have a very complicated geometry as the mapping target for FE mesh generation. Adoption of pig and human vertebrae as the mapping target in this study demonstrated the capability of the proposed method.

The time required for the mapping is dependant on the density of the points on the template mesh surface (base points) used for the normal line searching. Higher density of base points means more surface normal vectors are calculated for the mapping and hence longer computation time. Higher density of base points can be achieved by including extra intermediate nodes (points located in between existing FE mesh surface nodes) as base points for normal line searching. Despite prolonged computation time increased density of base points does not further improve the mapping accuracy significantly. For this reason, the selection of base points is confined to the existing surface nodes on the
template FE mesh with no extra intermediate nodes included. With this base point density, the computation time required for the mapping of a vertebra is about 15-20 minutes on a 3GHz Dell PWS650 workstation. Because of different mapping objects and computation facilities it is difficult to compare the proposed method with previously reported mesh mapping methods in terms of computational time. Nevertheless, the short computation time needed by the proposed method makes rapid construction of the patient-specific FE mesh possible.

It was observed that the element density of the template mesh affects the accuracy of the mapping. At the location where the target surface has complex morphology, such as the posterior processes, pedicles and vertebral osteophytes in the case of the vertebra, high density of elements in the template mesh is needed to achieve a close match of the mapped mesh surface with the target surface. The template mesh used in the dissertation is a relatively simplified mesh with limited number of elements (around 1100 elements). Hence the mapping accuracy in region of the vertebral posterior elements is as not as good as that in the vertebral body region that has relatively simple surface morphology.

One limitation of this method is that it is not fully automatic because pre-mapping adjustments and alignments of the template mesh are needed. Nevertheless, high quality pre-mapping adjustments and alignments of the template mesh enable its successful mapping into very complicated geometry. Another limitation is that certain elements located at the regions with complicated geometry may become distorted after the
mapping. Therefore element shape correction is an important post-mapping procedure in generating FE-compliant meshes.

4.5 Conclusion

This chapter has introduced a new method employing thin-plate spline interpolation and normal line searching to rapidly generate geometrically accurate FE meshes from patient-specific geometry. The capability of this method has been demonstrated by mapping the template mesh into very different target geometries. Further works will involve experimental validation of the generated meshes in predicting mechanical behaviors of the modeled subjects.
Chapter 5  FE simulation of a lumbar motion segment

5.1 Finite element model

The purpose of the works presented in this chapter is to demonstrate the spectrum of information which can be obtained by modeling on patient-specific geometry developed using the proposed mapping method. It is not meant as a systematic investigation on biomechanics of any spinal pathology.

Based on a segmented human lumbar spine CT dataset, the FE mesh of a lumbar motion segment (MS) (including L2-3 vertebra) was created using proposed method. The spinal ligaments and the intervertebral disc were also included in the model (Fig 5.1). The creation of this model took around 2 working days to complete. The lumbar spine segment has a lateral rightwards-convex curvature resulting in a $31.9^\circ$ angle between the top surface of the L2 and bottom surface of the L3 vertebra. This abnormal lateral curvature is the manifestation of scoliosis conditions suffered by the patient. The two vertebrae also show extensive osteophyte outgrowth at the lateral and anterior sides. The disc height is apparently shorter than normal, especially at the left side due to the scoliotic curvature and possibly disc degenerations as well.

The cortical shell thickness as measured in CT images at different locations of the lumbar vertebra using Photoshop (Photoshop, San Jose) is $1.41\pm0.13$mm. Therefore the cortical shell was defined as $1.41$mm thick in the finite element model. The bony vertebral endplates were assumed to have the same thickness as the cortical shell.
In the finite element model, the vertebra body consists of a cancellous bone core and a cortical bone shell. The cortical shell thickness was measured in 3 CT images taken at different heights of the lumbar vertebra using Photoshop. In each image 3 measurements were conducted and the average of the total 9 sample readings is 1.41mm. Therefore the cortical shell was defined as 1.41mm thick in the finite element model. The bony vertebral endplates were assumed to have the same thickness as the cortical shell. A specific program has been written to define the thickness of the cortical shell elements in the mesh generated from mesh mapping without changing the external surface of the mapped vertebrae.

The intervertebral disc is made of a nucleus core contained in a peripheral annulus wall. The annulus ground substance was modeled as solid elements embedded with 4 layers of truss elements representing the collagen fibrils. Both the collagen fibrils and the spinal ligaments were represented as tension-only truss elements which are only capable of resisting tension force. The posterior facets of the two vertebrae were modeled as contact problem with an initial gap distance of 0.5mm. It was further assumed that the joint surfaces can move along each other without friction.

The material properties of different spinal components were based on values reported in past literatures (Table 5.1, Kim et al., 1989). Considering the advanced stage of degenerations evidenced by all the abnormal conditions of this lumbar motion segment discussed above, the element type and material property of the nucleus pulposus was
chosen to reflect the characteristics of highly degenerated nucleus. Instead of incompressible hydrostatic elements, three-dimensional solid elements were adopted to represent the nucleus (Kim et al., 1989). The Young’s modulus of the nucleus was assumed to have a low value as 1MPa. The Poisson’s ratio was also assigned a relatively low value as 0.4 compared with 0.499 which many previous researches adopted to simulate incompressible fluid behavior of the nucleus (Lavaste, 1992).

![Fig 5.1 First and second row are different views of segmented lumbar motion segment and its FE model constructed, respectively.](image)

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus (MPa)</th>
<th>Poisson’s Ratio</th>
<th>Cross-section area (mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cortical bone</td>
<td>12000</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>100</td>
<td>0.2</td>
<td>-</td>
</tr>
<tr>
<td>Bony endplate</td>
<td>500</td>
<td>0.4</td>
<td>-</td>
</tr>
<tr>
<td>Posterior elements</td>
<td>3500</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>Annulus ground substance</td>
<td>4.2</td>
<td>0.45</td>
<td>-</td>
</tr>
</tbody>
</table>
### Ligaments

<table>
<thead>
<tr>
<th>Ligament</th>
<th>ALL (&lt;12%)</th>
<th>20 (&gt;12%)</th>
<th>PLL (&lt;11%)</th>
<th>20 (&gt;11%)</th>
<th>LF (&lt;6.2%)</th>
<th>19.5 (&gt;6.2%)</th>
<th>TL (&lt;18%)</th>
<th>58.7 (&gt;18%)</th>
<th>CL (&lt;25%)</th>
<th>32.9 (&gt;25%)</th>
<th>IS (&lt;14%)</th>
<th>11.6 (&gt;14%)</th>
<th>SS (&lt;20%)</th>
<th>15 (&gt;20%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALL</td>
<td>7.8</td>
<td>20</td>
<td>63.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PLL</td>
<td>10</td>
<td>20</td>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LF</td>
<td>15</td>
<td>19.5</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TL</td>
<td>10</td>
<td>58.7</td>
<td>1.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CL</td>
<td>7.5</td>
<td>32.9</td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IS</td>
<td>10</td>
<td>11.6</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SS</td>
<td>8</td>
<td>15</td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

ALL=anterior longitudinal ligament; CL= capsular ligament; IS = interspinous ligament; LF=ligamentum flavum; PLL=posterior longitudinal ligament; SS=supraspinous ligament; TL=intertransverse ligament

### 5.2 Biomechanical responses of the lumbar MS under pure loadings

#### 5.2.1 Axial compression

##### 5.2.1.1 Parametric study on the axial stiffness of the MS

Axial compressive load was applied as uniform axial displacement on the top surface of the superior vertebra with the bottom surface of the inferior vertebra fixed at all degrees of freedom. The reaction force at the bottom surface of the inferior vertebra was recorded and plotted with respect to the axial displacement applied during each increment of the simulation.

Because the geometry and material property of the intervertebral disc (IVD) is still difficult to be accurately ascertained from MR images, the only means to visualize IVD in vivo, it is currently not possible to assign patient-specific information to the IVD in the FE model. Nevertheless, certain level of patient-specificity in the FE modeling of the IVD can be achieved by varying some important IVD property parameters, such as the size of the disc nucleus, material property of the disc nucleus and disc collagen fiber content, etc. according to patients’ disc conditions. In order to simulation IV disc with
different degree of degeneration, the effects of the different parameters of the nucleus property on the compressive stiffness of the motion segment were investigated.

First of all the disc nucleus was modeled as an incompressible fluid cavity using hydrostatic elements to simulate a healthy disc with fully functional gel characteristics. Let’s call this model of the lumbar motion segment with hydrostatic elements representing the nucleus as *Normal model*. To simulate a degenerated disc with dehydration in its nucleus three-dimensional solid elements instead of incompressible hydrostatic elements were adopted to represent the nucleus (Kim *et al.*, 1989). The poisson’s ratio is an important attribute of solid materials. Many previous researches have adopted a value of 0.499 for the poisson’s ratio for the nucleus elements to simulate incompressible fluid (Lavaste, 1992; Smit, 1996). Therefore while the poisson’s ratio was fixed at 0.499 the Young’s modulus of the solid elements representing the nucleus was varied and the resulting axial compressive stiffness curves were compared with that of the *Normal model*. It was found that when Young’s modulus was given a value of 2MPa, the resulting axial compressive stiffness curve is almost identical to that of the *hydrostatic model*. This means that solid elements with Young’s modulus of 2MPa and Poisson’s ratio of 0.499 can be used to simulate a healthy disc nucleus under axial compression.

With the Young’s modulus fixed as 2MPa, the effect of differing Poisson’s ratio on the axial stiffness of the motion segment was studied. Decreasing Poisson’s ratio from 0.499 was found to reduce the axial compressive stiffness of the lumbar motion segment (Fig
5.2). This is because decreasing the Poisson’s ratio reduces the ability of the nucleus to expand laterally when subjected to axial compression, i.e. its gel characteristic.

The effect of differing Young’s modulus on the axial compressive stiffness of the motion segment was also studied. Reduced axial stiffness was observed with reduced Young’s modulus of the nucleus (Fig 5.3). The reason is apparently that nucleus with lower Young’s modulus value can take on less axial compression with same amount of displacement. It is interesting to note that removal of the nucleus did not completely abolish the axial stiffness of the motion segment. This is because of that the annulus ground substance which was assigned healthy material property is still capable to resisting axial compression.

![Graph showing the effect of Poisson's ratio on axial stiffness](image)

Fig 5.2 The effect of Poisson’s ratio of the disc nucleus on the axial stiffness of the lumbar motion segment
Chapter 5

5.2.1.2 Comparison with experimental results

Considering the advanced stage of degenerations evidenced by all the abnormal conditions of this lumbar motion segment discussed above, the material property of the nucleus pulposus was assumed to reflect the characteristics of highly degenerated nucleus. The Young’s modulus of the nucleus was assumed to have a low value as 1MPa. The Poisson’s ratio was also assigned a relatively low value as 0.4 compared with 0.499 which many previous researches adopted to simulate incompressible fluid behavior of the nucleus (Lavaste, 1992). The compressive stiffness curve was plotted and compared with experimental data reported in previous literatures (Fig 5.4a). The compressive stiffness curve computed is located at the lower spectrum within the envelope of the experimental data due to the degenerated disc nucleus simulated.

Fig 5.3 The effect of Young’s Modulus of the disc nucleus on the axial stiffness of the lumbar motion segment
5.2.1.3  Analysis of stress distribution pattern under axial compression

The Von Mises stress distribution pattern of the vertebral cortical shell when axial compression was applied was analyzed. To both the model with healthy disc nucleus simulated as incompressible hydrostatic cavity (Normal model) and the model with degenerated disc nucleus simulated as solid elements (Degenerated model) a high Von Mises stress “hot spot” (red region) was observed at the left side and mid-height of the superior vertebral cortex (Fig 5.5). This is probably due to firstly, the concave scoliotic curvature at the left side of the motion segment, and secondly, a pronounced concavity at the left side of the superior vertebra cortex as a result of osteophyte formation. It was noticed that under equal axial compression the value of Von Mises stress at the hot spot is significantly higher in the degenerated model than that in the normal model (Fig 5.6). In addition the high stress area is more concentrated in the left side of the cortical shell in the degenerated model than in the intact model. This might be explained by the impaired gel characteristics of the degenerated disc nucleus which not only reduced the overall axial stiffness of the motion segment but also compromised the disc’s load transfer and distribution functions.

From the medial sagittal view of both the normal and degenerated model it can be observed that the Von Mises stress in the cortical shell is much higher than that of the cancellous core (Fig 5.7). This is because that the Young’s modulus of the cortical shell (12,000MPa) is much higher than the Young’s modulus of the cancellous bone (100MPa). It was also noticed that in the superior vertebra the stress in the anterior cortical shell is higher than that in the posterior shell. Whereas, this difference in stress distribution in
cortical shell was in a reverse manner in the inferior vertebra, i.e. the stress in the posterior cortical shell is higher than that in the anterior cortical shell. A probable explanation for this phenomenon is that the superior vertebra is positioned slightly towards the posterior side over the inferior vertebra. This results in both the anterior portion of the cortical shell of the superior vertebra and the posterior portion of the cortical shell of the inferior vertebra being positioned closer to the centroid of the motion segment and therefore taking on more axial compression during load transfer.

Because the stress in the cortical bone is much higher than the stress in the cancellous bone the entire cancellous core shows up in blue color which represents the lowest stress value. In order to see the Von Mises stress distribution pattern of the cancellous core the maximum displaying stress value represented by the red color is lowered to a level close to the stress level experienced by the cancellous bone. The cortical region which now has higher stress value than the newly set maximum displaying stress is colored as grey.

The Von Mises stress contour plots of the cancellous core of both the Normal and Degenerated model are shown in Fig 5.8. In the cancellous core of the Normal model both inside and outside surface of the annulus bulge outwards because of the incompressible nature of the fluid nucleus simulated. This is consistent with the observations made by Meakin (Meakin, 2000). The central portion of the endplate is subjected to high stress. The stress in the cancellous bone adjacent to the endplates is evenly over the endplate. This is because that the hydrostatic characteristic of the nucleus enables the compression sustained by the nucleus to be transformed into equal hydrostatic
pressure exerted on the inside wall of the annulus and endplates. On the other hand, in the Degenerated model the inside surface of the annulus bulges inwards while the outside surface bulges outwards as reported by Meakin (Meakin, 2000). The Von Mises stress in the endplates is lower compared with that in the Normal model. The high stress area in the cancellous core is concentrated near the attachment site of the annulus ground substance. This shows that the degenerated disc nucleus is no longer capable of transferring axial compression force efficiently because of the loss of its gel characteristic. Therefore more axial load is transmitted through the annulus ground substance resulting in higher stress in nearby cancellous bone and reduced stress in the central portion of the endplates. This might explain the decreased incidence of the Schmorl’s Node in spines with seriously degenerated disc nucleus.

It is known that bone tissue undergoes remodeling process in response to chronic stress and strain according to Wolff’s law. Brown et al. (Brown et al., 1990) reported that tensile principal stress is a probable parameter associated with bone remodeling process in the form of shape optimization or internal variation in elastic moduli. Therefore the tensile principal stress distribution of bony endplates where osteophyte usually outgrowth occurs was studied. An interesting observation is that he high tensile principal region (red region) seems to coincide with the region of osteophyte formation (Fig 5.9). This suggests a possible correlation between osteophyte formation and tensile principal stress. Nevertheless, further investigations are needed to fully elucidate the exact role of tensile principal stress in the mechanism of osteophyte formation.
Fig 5.5 Von Mises stress distribution in the superior vertebral cortical shell of the motion segment model with a) a healthy disc nucleus or b) a degenerated disc nucleus (red, yellow, green and blue colors in order represent stress of highest value to lowest value, following figures follow the same convention).

Fig 5.6 Comparison of maximum Von Mises stress observed in the superior vertebral cortical shell under axial compression.
Chapter 5

Fig 5.7 Medial sagittal section view of the Von Mises stress distribution in the cortical shell of the Degenerated model under axial compression load of 3000N.

Fig 5.8 Medial sagittal section view of the Von Mises stress distribution in the cancellous core of a) the Normal model (the disc nucleus is not shown) and b) the Degenerated model under axial compression load of 3000N. (the grey color is used to represent the stress values which are higher than the stress value represented by the red color)
5.2.2 Flexion

5.2.2.1 Flexural stiffness of the lumbar MS

Flexural bending moment was applied to the motion segment model by adding a force couple onto the top surface of the superior vertebra. The bottom surface of the inferior vertebra was fixed at all degrees of freedom. The angular rotation of the superior vertebra was plotted against the flexural moment required to produce the rotation.

A similar parametric study as done in the compression simulation showed that the material properties of the disc nucleus did not affect the flexural stiffness of the lumbar motion segment significantly. The annulus collagen fibrils contribute to the flexural stiffness of the lumbar motion segment. The removal of the outer 2 layers of collagen fibrils reduced the flexural stiffness markedly (Fig 5.10). However, the removal of inner 2 layers of showed no significant reduction in the flexural stiffness. This might be due to
the fact that the outer layers of collagen fibrils are able to produce larger leverages under flexural bending because of larger anterior-posterior spacing between fibrils.

**Fig 5.10** The effects of dysfunctional collagen fibrils of the disc annulus on the motion segment flexural stiffness.

### 5.2.2.2 Comparison with the experimental results

For the flexion loading, simulation was also carried out without including spinal ligaments in the model. The flexural stiffness of the simulated motion segment is higher than previously reported experimental results that were mostly derived from normal motion segments without considering the spinal ligaments (Fig 5.4b). This high flexural stiffness of the motion segment can be attributed to the reduced disc height and extensive osteophyte formation which actually stabilized the motion segment (Tanaka, 2001). It...
was also reported that the motion flexibility decreases in lumbar motion segments with severely degenerated disc (Brown, 2002).

5.2.2.3 Analysis of the stress distribution pattern

The Von Mises stress distribution pattern in the cancellous core of the motion segment models with and without inclusion of the spinal ligaments are similar even though the model with spinal ligaments exhibits much higher flexural stiffness than the model without spinal ligaments (Fig. 5.11). However, the posterior elements, especially the pedicles and the spinous processes of the lumbar vertebrae in the ligamentous model experience much higher stress than those in the non-ligamentous model. This verified the notion that the spinal ligaments and posterior elements play an important role in resisting flexural bending. The supraspinous ligaments and the interspinous ligaments restrict the flexural bending motion of the lumbar motion segment by developing tensile force acting through their anchorage on the spinous processes. This leads to pronounced increase in the Von Mises stress in the spinous processes and the pedicles, which together resist bending moment induced by the ligaments like hanging cantilevers. Moreover, it is known that cantilevers with larger depth are more efficient in resisting bending moment than cantilevers with smaller depth. Hence it can be inferred that the geometrical dimension of the posterior elements can affect the flexural bending resistance of the motion segment significantly.
5.2.3 Lateral bending

Rightwards and leftwards lateral bending action of the motion segment model was induced by applying a force couple which stresses downwards either right or left side of the superior vertebra top surface while pulling upwards the other side. The boundary conditions are the same as those specified in other type of loading.

The lateral angular rotation at each simulation increment was plotted against the applied lateral bending moment. Simulations with and without including the spinal ligaments in the model were conducted. It was found that rightwards lateral bending stiffness is higher than leftwards lateral bending stiffness in both with ligament and without ligaments models (Fig 5.12). This unbalanced stiffness in lateral bending can be attributed to the rightwards scoliotic convex curvature of the motion segment. To both leftwards and rightwards bending inclusion of the spinal ligaments increases the stiffness.
Chapter 5

5.3 Conclusion

The construction and simulation of the lumbar motion segment finite element model demonstrates the potential applications of the proposed mesh mapping method in patient-specific finite element simulations. In a specific case of human spine modeling, as we know, many forms of lumbar spinal pathologies, such as osteophyte outgrowth, scoliosis, spondylolisthesis and lordosis are manifested by abnormal geometrical morphologies of the lumbar spine which contribute to the disruption of normal load-bearing and load-transferring functions of the lumbar spine and in some cases further exacerbation of certain spinal pathologies. The proposed method enables accurate representation of individualized pathological geometry of patients' lumbar spine in a finite element model.
Therefore the constructed FE model is capable of revealing the repercussions of the pathological spine geometry on the biomechanical characteristics of the patients’ lumbar spine. This kind of knowledge will definitely help orthopedic surgeons make decisions on the administration of corresponding treatments to restore or repair the impaired lumbar spine mechanical functions.
Chapter 6  Recommendations & Conclusions

6.1  Recommendations

6.1.1  Patient-specific geometry

Two important aspects of finite element simulation of anatomical structures like spine are accurate representation in the finite element model of the geometry and material property of the modeled subjects, respectively. Furthermore, for patient-specific finite element modeling the speed of the model construction is vital as well.

This dissertation proposed a novel method called Normal-line-searching TPSI method aimed at rapid and accurate representation of the patient-specific geometry of the lumbar spine by mapping a template mesh into patient-specific geometry. However, there are limitations of this method which should be addressed in future. Following paragraphs discuss future works which will possibly enhance the capability of the proposed method.

The mapping quality is compromised at the posterior elements of the vertebra because the low element density at this region in the template mesh is not adequate to reflect its complex geometry. Modifications on the template mesh can be made to increase the element density in the posterior element region of the template mesh. In the case of a vertebra with significant geometrical irregularities at its vertebral body region, the element density at the vertebral body region in the template mesh should also be increased to enable close matching of the mapped mesh with patient-specific geometry.
To ensure accurate mapping of the template mesh onto target geometry pre-mapping alignments and local adjustments of the template mesh are needed. Presently, these pre-mapping processing works of the template are done with Matlab coded programs. The effects of the pre-mapping alignments and adjustments on the modified template mesh can only be visualized via Abaqus CAE which is a separate modality from Matlab. Future works can be devoted to develop a software platform which enables interactive pre-mapping processing of the template mesh by integrating mesh modification and mesh visualization into one single piece of software with pre-mapping processing executed via icon-based user interface. This will speed up the pre-mapping alignments and adjustments procedures and reduce the overall time duration needed for new mesh generation.

6.1.2 Patient specific material property

The proposed mesh mapping method enables accurate capturing of the modeled subjects’ geometry. However; because of the anisotropic, inhomogeneous time-dependant nature of the material property of the biological structures assigning appropriate material property to the finite element model is also crucial to the accuracy of the modeling results besides accurate representation of the modeled subject’s geometry. In this dissertation the finite element model of the lumbar motion segment assumed generic material property values adopted by previous researches. A priority of future works should be developing in vivo material property acquisition methods to enhance the material specificity of the finite element modeling.
For determination of the vertebra bone tissue material properties in vivo efforts can be invested in finding and calibrating correlation formulas based on CT number like those developed by Carter and Hayes (Carter and Hayes, 1977). On the other hand, the currently most commonly used tool for soft tissue imaging, MRI, is presently used almost solely for the purpose of soft tissue visualization rather than providing quantitative information about the material properties of the soft tissues such as the intervertebral disc. Therefore novel techniques like administration of contrast agent, high resolution MR imaging should be explored for their possible application in helping extract more quantifiable information from the MR images.

### 6.2 Conclusion

This dissertation gives a comprehensive literature review on the finite element modeling of the lumbar motion segment and mesh generation techniques. Following that is the presentation of the developed Normal Line Searching TPSI mesh mapping method which achieved the first of the two objectives of this dissertation listed in the introductory chapter. The proposed method is capable of generating geometrically accurate finite element meshes in a short time period. Compared with other techniques discussed in the literature review section the advantages of the proposed method are that it can be used to map complex geometry like that of a vertebra and the element organization of the template mesh meant to represent different anatomical components is inherited in the generated mesh. Therefore the generated meshes have limited number of elements which are shaped and organized to efficiently represent the major geometrical features of the
anatomical structures modeled. Hence, the meshes created using the proposed method are particularly useful in modeling composite musculoskeletal structures like human spine.

The finite element simulation of a human lumbar motion segment accomplished the second objective of this dissertation. The simulation not only revealed some unique biomechanical characteristics of the motion segment with its unique geometry accurately captured in the finite element model but also demonstrated the potentials of the proposed mesh mapping method in patient-specific finite element modeling.

To conclude this study has achieved the two objectives set in the introduction chapter. Future works should be directed at improvement of the geometry mapping speed and quality and development of effective methods for tissue material property determination in vivo.
References

ABAQUS/Standard/CAE V6.3 (ABAQUS, Inc., Pawtucket)


MIMICS (Materialise NV, Leuven)


Photoshop (Adobe Systems, Inc., San Jose)
References


Tan, J.S., 1998. Finite element modelling of an intervertebral disc under pure and mixed loadings, dissertation (M.Eng.), Dept. of Mechanical and Production Engineering, Faculty of Engineering, National University of Singapore.


Appendix

A.1 Matlab coded program for normal line searching procedure

function nor=nor7
s1=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s1.txt',',',');
s2=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s2.txt',',',');
s3=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s3.txt',',',');
s4=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s4.txt',',',');
s5=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s5.txt',',',');
s6=load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s6.txt',',',');
element=load('C:\GY\Matlab6p5\work\bian\we\constant\element.txt',',',');
% N by 9 matrix
node=load
('C:\GY\Matlab6p5\work\bian\we\1premapping\alignout.out',',',');%input
ctnode=load('C:\GY\Matlab6p5\work\bian\we\constant\ctnode.txt',',',');
edgenode=load('C:\GY\Matlab6p5\work\bian\we\constant\edgenode.txt',',',');
% N by 4 CT nodes
edgenode=unique(edgenode);
Els1=unique(s1);
Els2=unique(s2);
Els3=unique(s3);
Els4=unique(s4);
Els5=unique(s5);
Els6=unique(s6);

[re,ce] = size(element);
elnode=element(1:re,2:ce);
eleID=element(1:re,1);

for s=1:length(Els1)
    [row, col] = find(element(:,1)==Els1(s));
    facet1(s,:)=element(row,[2 5 4 3]);
end

for s=1:length(Els2)
    [row, col] = find(element(:,1)==Els2(s));
    facet2(s,:)=element(row,[6 7 8 9]);
end

for s=1:length(Els3)
    [row, col] = find(element(:,1)==Els3(s));
    facet3(s,:)=element(row,[2 3 7 6]);
end

for s=1:length(Els4)
    [row, col] = find(element(:,1)==Els4(s));
    facet4(s,:)=element(row,[3 4 8 7]);
end

for s=1:length(Els5)
    [row, col] = find(element(:,1)==Els5(s));
    facet5(s,:)=element(row,[4 5 9 8]);
end

for s=1:length(Els6)
    [row, col] = find(element(:,1)==Els6(s));
    facet6(s,:)=element(row,[5 2 6 9]);
end
Appendix

facet=[facet1; facet2; facet3; facet4; facet5; facet6]; % all surface facets defined by 4 nodes, N by 4

[rf, cf] = size(facet);
[rn, cn] = size(node);
jj=1;

for s=1:rn
    [row, col] = find(facet==node(s,1));
    if and(length(row)<=6, length(row)>=1)
        surfnode(jj,:)=node(s,:); % surface nodes shared by
        jj=jj+1;
    end
end

[r1, c1] = size(surfnode);

for s=1:r1
    [row, col] = find(facet==surfnode(s,1));
    nodefacet(s,1)=surfnode(s,1); % surfnode with N(1-5) pairs of adjacent nodes in anti-closewise order (each adj node appears two times)
    for j=1:length(row)
        if col(j)==4
            nodefacet(s,2*j:2*j+1)=facet(row(j),[3,1]);
        elseif col(j)==1
            nodefacet(s,2*j:2*j+1)=facet(row(j),[4,2]);
        else
            nodefacet(s,2*j:2*j+1)=facet(row(j),[col(j)-1, col(j)+1]);
        end
    end
    if length(nodefacet(s,:))==1
        nodefacet(s,:)
    end
end

ndnormal=zeros(r1,4);

for s=1:r1
    [row, col] = find(nodefacet(s,:)>0);
    for j=1:(length(col)-1)/2
        ndnormal(s,1)=nodefacet(s,1); % surface normal vector at each surfnode
        ndnormal(s,1)=ndnormal(s,1);
    end
    if length(nodefacet(s,:))==1
        nodefacet(s,:)
    end
end

ndnormal=ndnormal/length(r1,4);

for s=1:r1
    [row, col] = find(nodefacet(s,:)>0);
    for j=1:(length(col)-1)/2
        ndnormal(s,1)=nodefacet(s,1); % surface normal vector at each surfnode
    end
end

% surfnode4 node(id, vector) N by 4
[row1, col1] = find(node(:,1)==nodefacet(s,1));
[row2, col2] = find(node(:,1)==nodefacet(s,2*j));
normal=cross(node(row2,2:4)-node(row,2:4), node(row1,2:4)-node(row2,2:4));
ndnormal(s,2:4)=ndnormal(s,2:4)+normal;
end
if norm(ndnormal(s,2:4))==0
    ndnormal(s,:
end
ndnormal(s,2:4)=ndnormal(s,2:4)/norm(ndnormal(s,2:4)); % normalize the normal vector
end
for j=1:rf
    side(4*j-3,:)=facet(j,[1 2]);
    side(4*j-2,:)=facet(j,[2 3]);
    side(4*j-1,:)=facet(j,[3 4]);
    side(4*j,:)=facet(j,[4 1]);
end

[side,IX] = sort(side,2);
side=unique(side,'rows');
[rs,cs] = size(side);

for j=1:rs
    ro1=find(edgenode==side(j,1));
    ro2=find(edgenode==side(j,2));
    if and(length(ro1)==1,length(ro2)==1)
        sidenode(j,8)=1;
    end
    [row2, col2] = find(node(:,1)==side(j,2));
    [row1, col1] = find(node(:,1)==side(j,1));
    sidenode(j,2:4)=0.5*(node(row2,2:4)+node(row1,2:4));
    [row2, col2] = find(ndnormal(:,1)==side(j,2));
    [row1, col1] = find(ndnormal(:,1)==side(j,1));
    sidenode(j,5:7)=(ndnormal(row2,2:4)+ndnormal(row1,2:4))/norm(ndnormal(row2,2:4)+ndnormal(row1,2:4));
end

for j=1:rf
    [row1, col1] = find(node(:,1)==facet(j,1));
    [row2, col2] = find(node(:,1)==facet(j,2));
    [row3, col3] = find(node(:,1)==facet(j,3));
    [row4, col4] = find(node(:,1)==facet(j,4));
    centernode(j,2:4)=0.25*(node(row1,2:4)+node(row2,2:4)+node(row3,2:4)+node(row4,2:4));
    [row1, col1] = find(ndnormal(:,1)==facet(j,1));
    [row2, col2] = find(ndnormal(:,1)==facet(j,2));
    [row3, col3] = find(ndnormal(:,1)==facet(j,3));
    [row4, col4] = find(ndnormal(:,1)==facet(j,4));
    centernode(j,5:7)=ndnormal(row1,2:4)+ndnormal(row2,2:4)+ndnormal(row3,2:4)+ndnormal(row4,2:4);
    centernode(j,5:7)=centernode(j,5:7)/norm(centernode(j,5:7));
end

centernode(:,8)=0;
extranode(:,2:8)=[sidenode(:,2:8);centernode(:,2:8)];%N by 7 node
(extranNd1d,x,y,z,normal vector)
[rex,cex] = size(extranode);
extranode(:,1)=[12001:12000+rex]';
surfnode=extranode(:,1:4);
ndnormal=extranode(:,[1 5 6 7 8]);
clear centernode sidenode extranode node facet side
[r1,c1] = size(surfnode);
[rct,cct] = size(centernode);
for s=1:r1
    if ndnormal(s,5)==0 %%%%%%%%%%%%%%%%%%%%
        ii=1;
        for j=1:rct
            if norm(ctnode(j,2:4)-surfnode(s,2:4))<=3.5
                %norm(dot(ctnode(j,2:4)-surfnode(s,2:4),ndnormal(s,2:4)))<=8
                ndline(ii,1)= ctnode(j,1);
                ndline(ii,2)=norm(cross(ndnormal(s,2:4),surfnode(s,2:4)-ctnode(j,2:4))); %ndline: distance of each ct node to the normal line of each surface node
                ndline(ii,3)=dot(ctnode(j,2:4)-surfnode(s,2:4),ndnormal(s,2:4)); %ndline(ii,3)=norm(ctnode(j,2:4)-surfnode(s,2:4));%ndline: distance of each ct node to each surface node
                ii=ii+1;
            end
        end
        if ii==1
            for j=1:rct
                if norm(ctnode(j,2:4)-surfnode(s,2:4))<=5
                    %norm(dot(ctnode(j,2:4)-surfnode(s,2:4),ndnormal(s,2:4)))<=8
                    ndline(ii,1)= ctnode(j,1);
                    ndline(ii,2)=norm(cross(ndnormal(s,2:4),surfnode(s,2:4)-ctnode(j,2:4))); %ndline: distance of each ct node to the normal line of each surface node
                    ndline(ii,3)=dot(ctnode(j,2:4)-surfnode(s,2:4),ndnormal(s,2:4)); %ndline(ii,3)=norm(ctnode(j,2:4)-surfnode(s,2:4));%ndline: distance of each ct node to each surface node
                    ii=ii+1;
                end
            end
            if ii==1
                tran(s,:)=[10000 0 0];
            else
                mini=sortrows(ndline(1:ii-1,:),2); % sequence the ct nodes according to their distance to the normal line first
                [rmini,cmini] = size(mini);
                for k=1:rmini
                    if mini(k,2)>1.4
                        mini(k:rmini,:)=[];
                        break
                    end
                end
                [rmini,cmini] = size(mini);
                mini=sortrows(mini,3);
                if rmini==0
                    tran(s,:)=[10000 0 0];
                else
                    [row, col] = find(mini(:,3)>0);
                    if length(row)>0
                        posNor=mini(rmini-length(row)+1:rmini,:); % if length(row)>=3
move = mean(ctnode([posNor(length(row), 1), posNor(length(row-1), 1), posNor(length(row-2), 1)])
        tran(s,:) = surfnode(s, 2:4) + dot(move - surfnode(s, 2:4), ndnormal(s, 2:4)) * ndnormal(s, 2:4);
    else
    negNor = mini;
    negNor = sortrows(negNor, 3); if rmini >= 3
        move = mean(ctnode([negNor(rmini, 1), negNor(rmini-1, 1), negNor(rmini-2, 1)])
        tran(s,:) = surfnode(s, 2:4) + dot(move - surfnode(s, 2:4), ndnormal(s, 2:4)) * ndnormal(s, 2:4);
    else
        tran(s,:) = surfnode(s, 2:4) + dot(ctnode(negNor(rmini, 1), 2:4) - surfnode(s, 2:4), ndnormal(s, 2:4)) * ndnormal(s, 2:4);
    end
    end
    end
    end
    else
        cont(s,:) = surfnode(s, 2:4);
    elseif ndnormal(s, 5) == 1
        jj = 1;
        for j = 1:rct
            if norm(ctnode(j, 2:4) - surfnode(s, 2:4)) <= 8
                ndline(jj, 1) = ctnode(j, 1);
                ndline(jj, 2) = norm(cross(ndnormal(s, 2:4), surfnode(s, 2:4) - ctnode(j, 2:4)));
            end
            jj = jj + 1;
        end
        mini = sortrows(ndline(1:jj-1,:), 2); % sequence the ct nodes
        [rmini, cmini] = size(mini);
        for k = 1:rmini
            if mini(k, 2) > 5
                mini(k:rmini,:) = [];
                break
            end
        end
        mini = sortrows(mini, 3); % then sequence the 100 ct nodes with smallest distance to the normal line according to their projection of the surface node normal
        [rmini, cmini] = size(mini);
        [row, col] = find(mini(:, 3) >= 0);
if length(row)>0
    negNor=mini(1:rmini-length(row),:);
    posNor=mini(rmini-length(row)+1:rmini,:);
    negNor=sortrows(negNor,3);
    posNor=sortrows(posNor,3);
    tran(s,:)=ctnode(posNor(length(row),1),2:4);

    tran(s,:) = surfnode4(s,2:4) + dot(ctnode(posNor(length(row),1),2:4)) -
    surfnode4(s,2:4), ndnormal(s,2:4)) * ndnormal(s,2:4);
else
    negNor=mini;
    negNor=sortrows(negNor,3);
    tran(s,:)=ctnode(negNor(rmini,1),2:4);

    tran(s,:) = surfnode4(s,2:4) + dot(ctnode(negNor(rmini,1),2:4)) -
    surfnode4(s,2:4), ndnormal(s,2:4)) * ndnormal(s,2:4);
end

cont(s,:)=surfnode(s,2:4);
end

clear mini ndline negNor posNor
end
[row, col] = find(tran(:,1)==10000);
tran(row,:)=[ ];
cont(row,:)=[];
node=plate(node,cont,tran);
dlmwrite('Final.out',node,','
A.2 Matlab coded program for iterative alignment procedure

```matlab
function align = align

s1 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s1.txt',',');
s2 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s2.txt',',');
s3 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s3.txt',',');
s4 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s4.txt',',');
s5 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s5.txt',',');
s6 = load('C:\GY\Matlab6p5\work\bian\we\constant\allfacet\s6.txt',',');

element = load('C:\GY\Matlab6p5\work\bian\we\constant\element.txt',','); % N by 9 matrix
node = load ('C:\GY\Matlab6p5\work\bian\we\lpremapping\mapa.out',','); % N by 4 matrix std FE nodes
ctnode = load('C:\GY\Matlab6p5\work\bian\we\constant\ctnode.txt',','); % mimics nodes
allsurfnd = load('C:\GY\Matlab6p5\work\bian\we\constant\allsurfnd.txt',',');

Els1 = unique(s1);
Els2 = unique(s2);
Els3 = unique(s3);
Els4 = unique(s4);
Els5 = unique(s5);
Els6 = unique(s6);

LLprocess = load('C:\GY\Matlab6p5\work\bian\we\constant\LLprocess.txt',',');
LLprocess = unique(LLprocess);
RLprocess = load('C:\GY\Matlab6p5\work\bian\we\constant\RLprocess.txt',',');
RLprocess = unique(RLprocess);
LsupN = load('C:\GY\Matlab6p5\work\bian\we\constant\LsupN.txt',',');
LsupN = unique(LsupN);
RsupN = load('C:\GY\Matlab6p5\work\bian\we\constant\RsupN.txt',',');
RsupN = unique(RsupN);
LinfN = load('C:\GY\Matlab6p5\work\bian\we\constant\LinfN.txt',',');
LinfN = unique(LinfN);
RinfN = load('C:\GY\Matlab6p5\work\bian\we\constant\RinfN.txt',',');
RinfN = unique(RinfN);
SpProcess = load('C:\GY\Matlab6p5\work\bian\we\constant\SpProcess.txt',',');
SpProcess = unique(SpProcess);

[re, ce] = size(element);

for s = 1:length(Els1)
    [row, col] = find(element(:,1) == Els1(s));
    facet1(s,:) = element(row,[2 5 4 3]);
end
for s = 1:length(Els2)
    [row, col] = find(element(:,1) == Els2(s));
    facet2(s,:) = element(row,[6 7 8 9]);
end
for s = 1:length(Els3)
    [row, col] = find(element(:,1) == Els3(s));
    facet3(s,:) = element(row,[2 3 7 6]);
```

Appendix

end
for s=1:length(Els4)
    [row, col] = find(element(:,1)==Els4(s));
    facet4(s,:)=element(row,[3 4 8 7]);
end
for s=1:length(Els5)
    [row, col] = find(element(:,1)==Els5(s));
    facet5(s,:)=element(row,[4 5 9 8]);
end
for s=1:length(Els6)
    [row, col] = find(element(:,1)==Els6(s));
    facet6(s,:)=element(row,[5 2 6 9]);
end
facet=[facet1;facet2; facet3; facet4; facet5; facet6]; %all surface facets defined by 4 nodes, N by 4

[rf,cf] = size(facet);
[rn,cn] = size(node);

[ra,ca] = size(allsurfnd);

for i=1:5
    [rIL,cIL] = size(LinfN);
    jj=1;
    ii=1;
    for s=1:rIL
        [row, col] = find(facet==LinfN(s,1));
        [rown, coln] = find(node(:,1)==LinfN(s,1));
        if and(length(row)<=6,length(row)>=1)
            surfnodeIL(jj,:)=node(rown,:);
            jj=jj+1;
        end
    end
    end
    [tranIL,contIL]=nor(surfnodeIL,node);

    [r,c] = size(tranIL);
diff=tranIL-contIL;
vector=mean(diff,1);
for j=1:r
    tranIL(j,:)=contIL(j,:)+vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

[rIR,cIR] = size(RinfN);
jj=1;
ii=1;
for s=1:rIR
    [row, col] = find(facet==RinfN(s,1));
    [rown, coln] = find(node(:,1)==RinfN(s,1));
    if and(length(row)<=6,length(row)>=1)
        surfnodeIR(jj,:)=node(rown,:);
        jj=jj+1;
    end
end
end
[tranIR,contIR]=nor(surfnodeIR,node);
\[ r, c \] = size(tranIR);
diff = tranIR - contIR;
vector = mean(diff, 1);
for j = 1:r
    tranIR(j,:) = contIR(j,:) + vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
\[ rL, cL \] = size(LLprocess);
jj = 1;
ii = 1;
for s = 1:rL
    [row, col] = find(facet == LLprocess(s, 1));
    [rown, coln] = find(node(:,1) == LLprocess(s, 1));
    if and(length(row) <= 6, length(row) >= 1)
        surfnodeL(jj,:) = node(rown,:);
        jj = jj + 1;
    end
end
[tranL, contL] = nor(surfnodeL, node);

\[ r, c \] = size(tranL);
diff = tranL - contL;
vector = mean(diff, 1);
for j = 1:r
    tranL(j,:) = contL(j,:) + vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
\[ rR, cR \] = size(RLprocess);
jj = 1;
ii = 1;
for s = 1:rR
    [row, col] = find(facet == RLprocess(s, 1));
    [rown, coln] = find(node(:,1) == RLprocess(s, 1));
    if and(length(row) <= 6, length(row) >= 1)
        surfnodeR(jj,:) = node(rown,:);
        jj = jj + 1;
    end
end
[tranR, contR] = nor(surfnodeR, node);

\[ r, c \] = size(tranR);
diff = tranR - contR;
vector = mean(diff, 1);
for j = 1:r
    tranR(j,:) = contR(j,:) + vector;
end

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
\[ rsL, csL \] = size(LsupN);
jj = 1;
ii=1;
for s=1:rsL
    [row, col] = find(facet==LsupN(s,1));
    [rown, coln] = find(node(:,1)==LsupN(s,1));
    if and(length(row)<=6,length(row)>=1)
        surfnodeSL(jj,:)=node(rown,:);
        jj=jj+1;
    end
end
[r1,c1] = size(surfnodeSL);
for s=1:r1
    [row, col] = find(surfnodeL(:,1)==surfnodeSL(s,1));
    if length(row)>0
        surfnodeSL(s,1)=0;
    end
end
[row, col] = find(surfnodeSL(:,1)==0);
surfnodeSL(row,:)=[];
[tranSL, contSL]=nor(surfnodeSL,node);
[r,c] = size(tranSL);
diff=tranSL-contSL;
vector=mean(diff,1);
for j=1:r
    tranSL(j,:)=contSL(j,:)+vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
[rsR,csR] = size(RsupN);
jj=1;
ii=1;
for s=1:rsR
    [row, col] = find(facet==RsupN(s,1));
    [rown, coln] = find(node(:,1)==RsupN(s,1));
    if and(length(row)<=6,length(row)>=1)
        surfnodeSR(jj,:)=node(rown,:);
        jj=jj+1;
    end
end
[r1,c1] = size(surfnodeSR);
for s=1:r1
    [row, col] = find(surfnodeR(:,1)==surfnodeSR(s,1));
    if length(row)>0
        surfnodeSR(s,1)=0;
    end
end
[row, col] = find(surfnodeSR(:,1)==0);
surfnodeSR(row,:)=[];
[tranSR, contSR]=nor(surfnodeSR,node);
[r,c] = size(tranSR);
diff=tranSR-contSR;
vector=mean(diff,1);
for j=1:r
tranSR(j,:) = contSR(j,:) + vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
[rsP, csP] = size(SpProcess);
jj = 1;
ii = 1;
for s = 1:rsP
    [row, col] = find(facet == SpProcess(s, 1));
    [rown, coln] = find(node(:, 1) == SpProcess(s, 1));
    if and(length(row) <= 6, length(row) >= 1)
        surfnodeSP(jj,:) = node(rown,:);
        jj = jj + 1;
    end
end
[tranSP, contSP] = nor(surfnodeSP, node);
[r, c] = size(tranSP);
diff = tranSP - contSP;
vector = mean(diff, 1);
for j = 1:r
    tranSP(j,:) = contSP(j,:) + vector;
end
clear diff vector

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

surfnode = [surfnodeL; surfnodeR; surfnodeSL; surfnodeSR; surfnodeIL; surfnodeIR; surfnodeSP];
[rs, cs] = size(surfnode);
for s = 1:ra
    [row, col] = find(surfnode(:, 1) == allsurfnd(s, 1));
    if length(row) == 0
        [rown, coln] = find(node(:, 1) == allsurfnd(s, 1));
        other(ii,:) = node(rown,:);
        ii = ii + 1;
    end
end
tran = [tranL; tranR; tranSL; tranSR; tranIL; tranIR; tranSP; other(:, 2:4)];
cont = [contL; contR; contSL; contSR; contIL; contIR; contSP; other(:, 2:4)];
node = plate(node, cont, tran);
for s = 1:rs
    [rown, coln] = find(node(:, 1) == surfnode(s, 1));
    if length(rown) == 0
        surfnode(s, 1)
    end
    Newsurfnode(s,:) = node(rown,:);
    error(s, 1) = norm(Newsurfnode(2:4) - surfnode(2:4));
end
if max(error) <= 0.02
i
break
end
clear tran cont other tranL tranR contL contR surfnode
surfnodeL surfnodeR surfnodeSL surfnodeSR surfnodeSP surfnodeIL
surfnodeIR
clear tranSL tranSR tranSP contSL contSR contSP diff vector
contIL contIR tranIL tranIR
dlmwrite('alignout1.out',node,'')