journal homepage: www.intl.elsevierhealth.com/journals/cmpb

An interactive multiblock approach to meshing the spine

*Nicole A. Kallemeyn, Srinivas C. Tadepalli, Kiran H. Shivanna, Nicole M. Grosland**

The University of Iowa, Iowa City, IA, United States

ARTICLE INFO

Article history:
Received 30 September 2008
Received in revised form
24 February 2009
Accepted 18 March 2009

ABSTRACT

Finite element (FE) analysis is a useful tool to study spine biomechanics as a complement to laboratory-driven experimental studies. Although individualized models have the potential to yield clinically relevant results, the demands associated with modeling the geometric complexity of the spine often limit its utility. Existing spine FE models share similar characteristics and are often based on similar assumptions, but vary in geometric fidelity due to the mesh generation techniques that were used. Using existing multiblock techniques, we propose mesh generation methods that ease the effort and reduce the time required to create subject-specific allhexahedral finite element models of the spine. We have demonstrated the meshing techniques by creating a C4–C5 functional spinal unit and validated it by comparing the resultant motions and vertebral strains with data reported in the literature.

1. Introduction

The finite element (FE) method constitutes one of numerous analytical techniques used in conjunction with experimental procedures to solve problems in spinal biomechanics. The unique capability of the FE method to evaluate stresses in structures of complex geometry, loading, and material behavior makes the technique advantageous over many other mathematical alternatives [1]. The literature reveals that FE models of varying complexities have been developed to study the spine [1–6]. Modeling techniques often require a number of simplifications in terms of geometry, material properties, and loading that may adversely affect the validity of the results. Finite element modeling as applied to three-dimensional and/or nonlinear structures remains tedious and expensive. Moreover, the expense of an FE analysis is frequently underestimated (i.e., software-licensing fees, computational time, personnel time for model development, in addition to running and analyzing the results, etc.). The primary bottleneck currently remains the time devoted to model development.

Nevertheless, the past two decades have shown a growing interest in computational modeling of the spine. In 1987, Yoganandan et al. [7] reviewed mathematical modeling of the spine, including continuum models, discrete and lumped parameter models, in addition to models based on the advanced finite element method. In 1995, Goel and Gilbertson [1] summarized applications of the finite element method to the thoracolumbar spine. That same year, Gilbertson [3] reviewed ongoing developments in spine biomechanics research through the finite element method. Shortly thereafter, Yoganandan et al. [6] followed up with a critical review of cervical spine finite element modeling applications. More recently, Fagan [2] presented an overview of finite element analysis in spine research, and Natarajan et al. [4] provided a review of the most recent advances in the development of poroelastic analytical models. These key review articles have concentrated on developments in model construction, constitutive law (material properties) identification, loading and boundary condition details, and efforts toward validating the models. The advancements in imaging techniques and the vast improvements in computational speed

* *Corresponding author at*: 1418 Seamans Center for the Engineering Arts and Sciences, The University of Iowa, Iowa City, IA 52242, United States. Tel.: +1 319 335 6425; fax: +1 319 335 5631.
E-mail address: nicole-grosland@uiowa.edu (N.M. Grosland).
0169-2607/$ – see front matter © 2009 Elsevier Ireland Ltd. All rights reserved.
doi:10.1016/j.cmpb.2009.03.005

have permitted the complexity of the models to continue to increase.

Today, the majority of spine models [6,8–21] share the following characteristics. The geometry is based on image datasets, typically axial slice contours from CT image datasets from a cadaveric specimen (age: late 60s +). The models are multi-segment in nature, consisting of a given number of vertebrae, each separated by an intervertebral disc, bilateral articulating facet joints, and representations of the ligamentous structures at each segmental level. Nonlinear cable or truss elements are typically used to define the ligaments, while contact interactions at the facets are modeled via surface interactions, or more often using gap elements with no resistance in tension and a stiffening resistance in compression. The intervertebral disc is often represented by an isotropic material reinforced by criss-crossing fibers, while the nucleus is represented as an incompressible fluid filled cavity. The assumption of symmetry is oftentimes adopted to minimize the time devoted to mesh development as well as computational run time. This assumption also aids model validation in the sense that the number of loading conditions considered may be reduced. Lastly, commonly accepted material property assignments are taken from the literature [10,21,22]. Despite these similarities, the geometric fidelity of these models differs considerably. This dissimilarity is likely attributed to the challenges associated with meshing the geometric irregularities of the spine. Semi-automated mesh generation techniques exist for the spine [23–25]; however, spine FE models generated using these procedures seldom consist entirely of hexahedral elements, and are often comprised of a mixture of tetrahedral, wedge, and hexahedral elements. Hexahedral elements are generally accepted as being the preferred element type for 3D nonlinear analyses [24–28].

In an effort to ease hexahedral mesh development for anatomic modeling we recently released an open-source software package called IA-FEMesh (Iowa FE Mesh; www.ccad.uiowa.edu/mimx/IA-FEMesh) [29]. The meshing techniques employed by the software afford the user an interactive approach to multiblock meshing. As the complexity of the structure under consideration increases, however, so does the nature of the required block structure. Consequently, the objective of this study was to develop an enhanced set of tools aimed at modeling the irregular structures of the spine. The goal being that these proof-of-concept modeling techniques will ultimately be incorporated into the software to ease modeling of such structures; with the hopes of making subject-specific modeling a reality. Herein, the meshing techniques are demonstrated on the cervical spine. Moreover, a C4–C5 model was validated by comparing the resultant motions and vertebral strains with data reported in the literature.

2. Methods

2.1. *General building block technique*

For ease of illustration, the basic modeling practices behind the multiblock technique are demonstrated by meshing a simple sphere (Fig. 1). Each model initiates with a triangulated surface (STL or VTK format) of the structure(s) of interest. Thereafter, a block, or series of blocks are interactively positioned about the surface. As illustrated, a single block may be sufficient (Fig. 1A). As the complexity of the structure increases, however, so does that of the required block structure. Each building block (BB) is composed of mesh seeding arranged in rows, columns, and layers; the corresponding level of seed refinement is specified by the user (Fig. 1B). The mesh seeds of the building block are projected, via closest point projection, onto the surface of interest (Fig. 1C). As a result, the mesh seeds are morphed to the bony surface as nodes to lay the foundation for the FE mesh. Thereafter, Laplacian smoothing [30] is performed on the surface nodes, followed by elliptic [31] or transfinite [32] interpolation to compute the locations of the interior nodes (Fig. 1D). Once the nodal definitions are established throughout, the volume is filled with hexahedral elements. Thereafter, the material properties and loading/boundary conditions are assigned to the mesh and the model is exported in ABAQUS format. Please refer to Grosland et al. [29] for a more detailed overview of the meshing technique.

2.2. *Cervical spine meshing methods*

The cervical spine meshing techniques employ methods similar to those described for the sphere with modifications to accommodate the complex geometry. Multiple BBs were used to capture the vertebral features, and novel meshing tools were developed to connect the posterior and vertebral body mesh definitions.

CT data was acquired from a 74-year-old male cadaveric specimen (Siemens Sensation 64 CT scanner, slice thickness 0.6 mm, 0.5 mm in-plane resolution). Each vertebra was segmented to produce surface representations in STL format (Fig. 2A). The segmentation process was similar to that described by DeVries et al. [33], which has been shown to accurately represent the true bony geometry. Each vertebra was meshed in two stages (the vertebral body, followed by the posterior region) using the process described below. The amount of time to generate each mesh was of interest for comparison to other methods.

2.2.1. *Vertebral body meshing*

An imposed constraint of the spine model is the mesh pattern defining the vertebral body (Fig. 2E). This pattern was chosen based on experience, namely modeling the connectivity of the intervertebral disc (i.e., distinguishing the annulus fibrosis and the nucleus). Most notably, the pattern simulates the concentric layers of the annulus and accommodates the annular fiber definitions in each element. Again, the vertebral mesh initiated with the surface defined from the image dataset. Using ParaView (www.paraview.org), the STL surface definition was initially clipped at the pedicles enabling the posterior region to be removed. Delaunay triangulation was used to patch the hole introduced at the pedicles, thereby yielding a closed, continuous surface (Fig. 2B).

By coupling a novel interactive tracing technique with our projection method, it enabled the vertebral centrum to be meshed with relative ease. The first step consisted of delin-