A Parametric Study of the Spinal Motion Segment Unit Fixation Using the Finite Element Method

Mohamed Fahmy, Ph.D., P.E.
Department of Industrial Engineering, University of Miami, U.S.A.

Shihab Asfour, Ph.D.
Department of Industrial Engineering, University of Miami, U.S.A.

Loren Latta, Ph.D., P.E.
Department of Orthopaedics & Rehabilitation, University of Miami, U.S.A.

Lutz Biedermann, CPO
Biedermann – Motech, GmBH, Schwenningen, Germany

Abstract

Metal screws and rods are used in intervertebral disk fixation. The most common in mechanical failure clinically is at the lower screw. In order to reduce or eliminate this mechanical failure, it is useful to study the mechanical behavior of the implant under different loading conditions. Experimental work has shown the effects of varying rod diameters on the bending moments and stresses in the rods, but stresses in the screws could not be addressed by experimental techniques. In the current paper, a 3-D finite element biomechanical model was developed using ANSYS software. The model included two vertebral bodies and the implant, which consists of two screws, a rod, and a nut on each side of the vertebrae. The intervertebral disc was completely removed and a metal cage of diameter of 16 mm was provided on each side. In this work an extension moment of 10 N.m with a compressive force of 1000 N was studied. In the model, a 10-node tetrahedral element was used to represent the vertebral body, screws, the rods, the nuts, and the graft bone, while an 8-node shell element was used to model the cage. Several parameters were investigated to study their effects on the normal stress distributions in the screws, rods, cage, and graft bone. These parameters include the rod diameter (4mm, 5mm, and 6.3 mm), the presence of a cage with and without bone graft, and the location of the cage (anterior versus posterior). A total of 12 cases for each parameter will be reported.

Introduction

Surgical fixation is used to treat patients with certain lumbar spinal conditions such as fractures, degenerative spondylolisthesis, tumors, degenerative scoliosis, and severe disc disease. There are several devices that are used for this purpose. The pedicle screw fixation device has become the standard procedure for this type of fixation [1 through 3]. The most significant complication with these devices is the breakage of the pedicle screw. Yahiro [4] showed that 7.1% of all patients had broken pedicle screws, while all other complications were less than 5.6% in the same group, based on clinical data. In a cohort study of pedicle screw fixation, Yuan et al [1] showed that 6.7 % of patients with fracture and 2.6% of patients with degenerative spondylolisthesis had pedicle failure. This failure is due to high mechanical stresses which are very difficult to measure experimentally. The finite element method is an ideal tool to study the stress distribution in all parts of the fixation device and hence design a better fixation device. There have been extensive research in modeling two or more vertebrae in the last 30 years. The first detailed finite element model of the disk-body unit was given by Belytschko et al [5]. Goel et al [6] presented the first work to include an implant in the finite element modeling. They developed an intact model for L4-L5 that was modified to simulate bilateral decompression surgery. It was further stabilized using an interbody bone graft and a set of Steffee Plates and screws. They showed that using a spinal fixation device induce stress shielding in the vertebral body. Goel et al [7] studied the effect of reducing the rigidity of the fixation devices on stress shielding by comparing among different posterior implants. Skalli et al [8] investigated the stability of three segment vertebral bodies when the middle vertebral body is fractured using anterior
bone graft with or without Cotrel-Dubousset-like posterior device. Zhang and Yang [9] investigated three types of anterior fusion. Their study emphasized the importance of a larger interface area to the success of the spinal fusion. Tororibe et al [10] evaluated the stability in posterolateral fusion using 3-D nonlinear finite element method. They also examined the effects of facet fusion and disc denucleation on the posterolateral fusion. Rohlmann et al [11] studied the impact of changing the rod diameter of an internal bisegmental fixator. Movements of threaded cages in posterior lumbar interbody fusion were studied by Pitzen et al [12]. Recently, Fahmy et al [13] studied the effect of the rod diameter and location of the cage on the stresses induced in the implant for a spinal fixation device subjected to flexion moment.

Procedure

The size and shape of the vertebral body was taken from an L5 vertebral body of an adult cadaver. The intervertebral disk and the posterior bodies with the attached ligaments were removed from the model. It has been assumed that a perfect bond at the screw/rod interface and also at the screw/bone interface. The model consisted of two vertebral bodies and a posterior fixation device. The fixation device included two screws, one rod, and one nut on each side of the vertebral bodies. A titanium mesh cage with or without a cancellous bone graft is provided either. There were two positions of the cage: (1) anterior location and (2) posterior location. The thickness of the vertebral body was 18.00 mm, while the average thickness of the intervertebral disk was 10.00 mm. The screw diameter was 4.00 mm and the cage diameter was 16.00 mm with 1.00 mm thickness. Three diameters for the rod were used: 4.00, 5.00 and 6.3 mm, respectively.

A three-dimensional finite element was developed using ANSYS software. Two elements were selected from the ANSYS element’s library: three-dimensional 10-Node (Solid 92) tetrahedral structural solid element and 8-node structural shell element. The tetrahedral element was used to represent the vertebral bodies, the bone graft, the screws, the rods, and the nuts. The shell element was used to model the titanium cage. Only half of the model was analyzed due to the geometrical and loading symmetry about the sagittal plane. This was achieved by restraining all nodes on the sagittal plane in the X-direction. All the nodes on the lower surface of the inferior vertebral body were restrained in all three directions to simulate a complete fixation at this level. The finite element models for the anterior cage and the posterior cage are illustrated in Figure 1. All the materials in the model were considered elastic and isotropic which require two parameters to describe their properties: E (elastic modulus) and ν (Poisson’s ratio). The cage was approximated by a solid tube that has 1.00 mm thickness with E of 4360 (Mpa) which provides a structural stiffness equivalent to the cage with its perforations. This value was calculated from an experiment conducted on HARMSr titanium mesh cage in compression in our biomechanical laboratory. The materials properties parameters are listed in Table 1.

![Figure 1 - Finite Element Model](image-url)
The applied load consisted of a force of 1000 N that was equally distributed normal to the posterior one-fifth of upper surface of the superior vertebral body to generate an extension moment of 10 N.m. The model was validated as described in [13].

Table 1. Materials Properties for the Model:

<table>
<thead>
<tr>
<th></th>
<th>(Elastic modulus) (MPa)</th>
<th>$\nu$ (Poisson’s ratio)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cancellous Bone</td>
<td>100</td>
<td>0.2</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>200000</td>
<td>0.3</td>
</tr>
<tr>
<td>Cage</td>
<td>4360</td>
<td>0.3</td>
</tr>
</tbody>
</table>

**Analysis Results & Discussion**

The results reported here included maximum normal tensile and compressive stresses in the rod, screws, cage, and graft bone. There were 12 cases for each parameter. Figures 2 and 3 show the maximum normal tensile and compressive stresses in the rod. The rod diameter was the most important factor in affecting the maximum stress values, as one expects. The maximum stress values decreased as the diameters increased. Higher stress values resulted in case of anterior cage than those resulted in case of posterior cage. The axial compressive force increased as the rod diameter increased from Figure 12. Figures 4 and 5 depict the maximum normal tensile and compressive stresses in the distal screw. As the diameter of the rod increased, the maximum normal tensile and compressive stresses increased in case of anterior cage. The same trend was found for the proximal screw as can be seen from Figures 6 and 7. The stress levels in the distal screw were always higher than those in the proximal screw, which is in agreement with failure that has higher degree of occurrence in the distal screw than in the proximal screw. Maximum tensile stresses in the anterior cage were higher compared with posterior cage. The opposite was true in case of maximum compressive stresses, where they were associated with posterior cage compared with anterior cage as shown in Figures 8 and 9. The axial compressive forces were higher in posterior cage than in case of anterior case. In both cages, the compressive force decreased with the increase of the rod diameter as in Figure 13. The stress values in the cage evaluated from the current model are low estimate of the real peak stresses in the cage, since the cage was modeled as a solid tube of the same axial structural stiffness. The maximum tensile and compressive stresses in the graft bone are shown in Figures 10 and 11. The tensile stresses in both the cages and the graft bone can be used to determine the minimum pre-compresses force needed to keep the graft bone and the cage in contact with both vertebrae. Similar to the cage, the axial force in the graft bone decreased with increasing the rod diameter and had higher values in case of posterior cage than in case of anterior cage as shown in Figure 14. The presence of the graft bone had resulted in lower stresses values in all cases except for maximum tensile stresses in the posterior cage. The range of stresses between the presence and the absence of graft bone can be viewed as the change of the stresses in each element just after the surgery and the complete healing is achieved.
Figure 2 - Maximum Normal Tensile Stresses in the Rod

Figure 3 - Maximum Normal Compressive Stresses in the Rod
Figure 4 - Maximum Normal Tensile Stresses in the Distal Screw

Figure 5 - Maximum Normal Compressive Stresses in the Distal Screw
Figure 6 - Maximum Normal Tensile Stresses in the Proximal Screw

Figure 7 - Maximum Normal Compressive Stresses in the Proximal Screw
Figure 8 - Maximum Normal Tensile Stresses in the Cage

Figure 9 - Maximum Normal Compressive Stresses in the Cage
Figure 10 - Maximum Normal Tensile Stresses in the Graft Bone

Figure 11 - Maximum Normal Compressive Stresses in the Graft Bone
Figure 12 - Axial Compressive Force in the Rod

Figure 13 - Axial Compressive Force in the Cage
Conclusion

In the current paper, a parametric study to investigate the effect of the rod diameters and location of the cage on the stress level of a spinal fixation implant has been conducted. The load in the current study was a compressive force of 1000 N with an extension moment of 10 N.m. The results have indicated that the increase in rod diameter resulted in decrease in the maximum stress levels in the rod and the cage and increase in the maximum stress levels in both screws. The effect was mixed in the maximum stress levels in the graft bone. Results also have shown that it is preferable to place the cage in a posterior location as opposed to place it anteriorly due to the lower stresses occurred in every component of the hardware under the extension moment. In a recent study by Fahmy et al [13], the anterior cage was at an advantage over the posterior cage for flexion moment. The optimal location of the cage should consider all types of loading and the frequency of their occurrence during a typical day of the patient’s activity.

References


