Modeling, Simulation and Determination of the State of Tension and Deformation of the Driver's Spine

Andrei Zoltan Farkaş, Veronica Argeşanu, Mihaela Jula, Raul Mikloș Kulcsar, Ionuț Silviu Borozan, Carmen Sticlaru, Inocențiu Maniu

Keywords
Modeling, Simulation, Wear, Driver’s Spine, FEM

Email address
farkasandreiz@gmail.com (A. Z. Farkaş)

Citation

Abstract
The objective is to determine when musculoskeletal pain and/or diseases occur in drivers running the car on different routes which leads to deformations due to friction an wear of the intervertebral disc. Determination made by analyzing the tension and strain of the spine using musculoskeletal modeling and FEM simulation. After simulating the driving conditions we obtained results that exceed values compared with data from the literature in the field due to friction and wear.

1. Introduction

As Zenk R. and his research team state in their research paper “in modern society the automobile is an essential companion in everyday life. Be it commuting to and from work or during our leisure time - every week most of us spend many hours sitting in their car” [19], statement that we all empathize in our daily life. The human body has not yet been adapted to being exposed to the G forces that occur while driving and to the prolonged sitting position, continue contraction of the spine’s muscle groups, induces an additional load on the spine’s vertical direction increasing the equivalent tension in the vertebrae and in particular in the intervertebral discs, due to this factors, findings in open literature report that drivers will encounter driving fatigue [8], [9] after long distance driving, fatigue that is directly linked to musculoskeletal pain and eventually diseases [14], [3] caused by deformations that exceed the limits in which musculoskeletal disorders can be avoided or treated by physiotherapy recovery. Considering the large number of active drivers it is important for us to determine the moment when musculoskeletal pain/diseases occurs, an analysis is required of the tension and strain state of the spine using musculoskeletal modeling.

The shape of the spine in the sagittal plane and inclination amplitudes in the coronal plane are determined directly from the driver's body reactions to the forces that occur when running the car on different routes. When the spine is in a non ergonomic position in the sagittal plane, the coronal amplitudes of the lateral inclinations lead to deformations of the intervertebral discs, which exceed the anatomical limits mentioned earlier [12].

Based on these statements we require a finite element analysis of a segment of the spine, consisting of two vertebrae and the intervertebral disc between them. [1], [2].
The segment is under compression and the vertebrae have a rotational movement of amplitude limited on the coronal plane corresponding with the spinal segment [2].

The study aims to determine the equivalent tensions and the deformations that drivers are subjected to, using a finite element method analysis. Analysis that will highlight the maximum equivalent stresses and the deformation that occur in the region of the spine that is the most predisposed to musculoskeletal pain and diseases.

2. Finit Element Method Analysis of the L4-L5 Segment Using the ANSYS Software

According to the literature in the field, the highest stresses occur in the lumbar spine area, particularly between the L4-L5 and L5-S1 segments. In this analysis we will consider the L4-L5 segment. [4], [5], [7], [8], [10], [11]

The L4 and L5 vertebrae were modeled in CATIA v5 CAD software [12], [13] the models were imported and assembled in Pro Engineer CAD software, after which we modeled the intervertebral disc that is located between the lower surface of the L4 vertebra and the top surface the L5 vertebra.

The whole assembly was then imported into ANSYS 11.0 software. In Figure 1 the L4-L5 segment is presented after it has been imported, and in Figure 2 we present the intervertebral disc modeled on the L4 and L5 vertebrae surfaces.

After importing the L4 - intervertebral disc - L5 assembly we introduced the mechanical characteristics of the vertebrae and the intervertebral disc. According to the literature in the field the mechanical characteristics of bone structure of the vertebrae and intervertebral disc structure are as given in Table 1.

The next step was meshing the parts. In Figure 3 is presented the L4 segment -intervertebral disc -L5 segment mesh.

Figure 1. The L4 - intervertebral disc - L5 assembly imported in ANSYS 11.0. software.

Figure 2. The intervertebral disc modeled in Pro Engineer CAD software according to the lower surface of L4 and the top surface of L5, and imported into ANSYS 11.0 software.
In the ANSYS software, Workbench Platform Simulation Module was chosen with Flexible Dynamics. The L5 vertebrae is considered as embedded in its lower surface, and to the L4 vertebra we imposed a rotation identical to the coronal plane Simulation in CATIA V5 CAD software in DMU Kinematics Module. [3]

| Table 1. The mechanical characteristics of the vertebrae and intervertebral disc used in the simulation. |
|--------------------------------------------------|-------------------------------------------------------------------------------------------------|
| Structure                        | Young's Elastic Modulus [MPa] | Poisson Coefficient [\(\nu\)] | Density [\(\rho\)] [kg/m³] |
| Vertebra                          | 4.10²                          | 0.3                             | 2000                         |
| Intervertebral disc               | 1                              | 0.5                             | 1250                         |

The loading was done by applying to the upper surface of the L4 vertebra the corresponding pressure force resulting from the gravitational acceleration product with the upper body mass of the driver. The top surface area of the L4 vertebra was approximated using CAD software Solid Edge V10. In Figure 4 it is shown how to approximate the surface area of the upper vertebra L4. In the figure we can see that the surface is divided into four different areas noted by A1, A2, A3 and A4. According to the values resulting from software CAD Solid Edge V10, the entire surface area is approximately:

\[
A_{sL4}=A_1+A_2+A_3+A_4 = \frac{\pi \times 25.22^2}{2} + 13.2 \times 24.04 + \frac{\pi \times 13.2^3}{4} + \frac{\pi \times 13.2^3}{4}
\]

\[
A_{sL4}=1590.126042 \text{mm}^2
\] (1)

The time for conducting the simulation coincides with the sinusoidal equation frequency which describes the time variation of the angle between vertebrae L4 and L5 [2]. This time is divided into 18 simulation steps (time steps).

3. FEM Analysis Results Using ANSYS Software

FEM analysis of the L4-disc-L5 assembly sought to determine the equivalent tensions in the vertebrae and disc, and in particular deformations of the intervertebral disc.

Figures 5 a), b) and c) shows the distribution of tensions in the vertebrae, from the 18 time steps mentioned earlier we present the maximum inclination limits left-right, and in neutral vertical position.
In Figure 6 the time variation of the maximum equivalent stress in the vertebrae is graphically presented for all the 18 time steps used in the simulation of the spine's movement.

On the time axis “0” represents the maximum left inclination of the spine and “1, 6337” represents the maximum right inclination of the spine, neutral vertical position finding itself at “0, 7688” on the time axis.

In Figure 8 the time variation of the maximum equivalent stresses in the intervertebral disc is graphically presented for all the 18 time steps used in the simulation of the spine's movement.

On the time axis “0” represents the maximum left inclination of the spine and “1, 6337” represents the maximum right inclination of the spine, neutral vertical position finding itself at “0, 7688” on the time axis.

Figure 9 a), b) and c) shows the distribution of tensions in the intervertebral disc’s lower side this time, left-right maximum inclination, and neutral vertical position taken out of the 18 time steps used in the simulation.
In Figure 10 the time variation of the maximum equivalent stress in the intervertebral disc is graphically presented for all the 18 time steps used in the simulation of the spine's movement.

On the time axis “0” represents the maximum left inclination of the spine and “1.6337” represents the maximum right inclination of the spine, neutral vertical position finding itself at “0, 7688” on the time axis.

4. Conclusions and Discussion

Analyzing the results obtained (Figure 5,..., 10), we can see that the maximum equivalent stress distribution in both vertebrae and the intervertebral disc, are focused in their circumference in every step of the dynamic FEM analysis.

In comparison with data from literature in the field [3, [6], [9], [14], [17], [19] equivalent stress in the intervertebral disc have limit values of 3MPa on the circumference and it has a thickness of about 6mm, and towards the intervertebral disc nucleus the allowed maximum stress limits are 2MPa.

From the resulting dynamic FEM analysis, the maximum equivalent stress of the intervertebral disc in the extreme left is 2.93MPa. According to the literature in the field [15], [18], [19] the height of the intervertebral discs in the lumbar spine area is approximately 10mm. The maximum deformation resulting from FEM dynamic analysis is 7.99mm.

References


