FINITE ELEMENT MODELING OF HUMAN CLAVICLE UNDER DYNAMIC LOADING

Razvan Rusovici, Ph.D., Megan Pendergast, M.S.
Florida Institute of Technology,
150 W. University Blvd. Melbourne, Florida, 32901 U.S.A.
rrusovici@yahoo.com
Jeffrey T. O’Brien, M.D.
650 S. Courtenay Pkwy., Merritt Island, Florida, 32952 U.S.A
Isabel Ghita, M.D., Ph.D.
Universitatea de Medicina si Farmacie “Carol Davila”
Blvd. Eroilor Sanitari 8, Bucharest, Romania

ABSTRACT
Clavicle fractures are common injuries that may from dynamic events such as falls or blunt-body trauma. Methods of repair may be non-operative, or surgical. Surgical repair is accomplished via fixation plates or intramedullary rods. The research investigated the use, and prediction accuracy, of advanced material models within a finite element framework to predict dynamic stresses and strains within the clavicle body which would be subjected to a dynamic load. The material models used were considered to be viscoelastic and orthotropic. The proposed finite element models were successfully verified against experimental results available in literature. The models were then used to predict the time-varying stresses and strains within the clavicle as a result of arbitrarily-chosen, short-impact dynamic loading. A comparison of predictions, corresponding to each material model, was performed.

KEY WORDS
Finite Element, Viscoelasticity, Clavicle, Fracture

1. Introduction
Clavicle fractures are a relatively common injury and account for 20% of all adult upper-body fractures, and 2.6-5% of all fractures [1,2,3]. These may be caused by dynamic loading that is a direct result of falls (mostly motorcycle-accident related) on shoulder or outstretched upper limbs, or blunt-body trauma. The prediction of the time-dependent strain and stress distribution within the clavicle during the accidental dynamic loading, with an accurate analytical model is important for design of prevention and safety systems [1].

There are many methods to heal the fracture, including both non-operative and surgical approaches. The surgical methods include either the insertion of an intramedullary rod or a prosthetic fixation plate. The most common surgical repair method is the insertion of a prosthetic plate, which is fixed to both segments of the fractured clavicle via screws. The type of the screws (locking vs. non-locking), number of screws and plate thickness are all surgery-related variables that must be considered and implemented by the surgeon based on her or his training and experience. An analytical approach that could indicate these optimal set of surgical variables would be a valuable tool to the surgeon. While common, clavicle fractures have not been researched as extensively and intensively as similar research performed for bones such as the femur and tibia. This fact applies to both experimental and analytical investigations [4]. A potential cause for the relative small of experimental studies is that the human shoulder anatomy (such as glenohumeral ball and socket joint) is specific to climbing and grasping mammals (including humans and apes) [4], thus clavicle experimental studies cannot be conducted readily on other type of animals.

In an attempt to answer the existing issues, the current work focused on developing an analytical predictive approach centered on a finite element model (FEM) of the clavicle. The clavicle FEM predictions used in this work were validated by comparison to experimental results [4]. Finite element modeling enables stress and strain analysis of the clavicle under various fracture scenarios without the need for extensive experimentation [6]. The FEM outlined by this research is anatomically accurate and employs advanced material models to represent cortical and trabecular bone properties, such as viscoelasticity. Since short-duration impact force could generally be characterized, in the frequency domain, by a bandwidth covering spanning several hundreds of hertz, it was important to represent the frequency-dependence of the bone material moduli by including viscoelasticity in the material representation. One of the goals of this paper was to ascertain the difference among displacement predictions caused by dynamic loading of clavicle, for FEM’s that alternatively did and did not incorporate bone
viscoelasticity, and based on published experimental results.

The presented research lays the initial groundwork for a clavicle and fixation-plate analytical model, one that would be able to predict the optimal surgical parameters (plate thickness, type and number of screws) for a specific patient. Such a model could prevent occasional occurrences of stress-shielding in repaired clavicles. The model could also predict the evolution of the maximum loading that the patient may place on the clavicle as the healing process evolves. This prediction capability would be important since fracture reoccurrence is often caused by patients who involve themselves in regular physical activities a short time after surgery.

2. Model Development

This section will describe the FEM development, including mesh development, material properties, boundary conditions (loading and constraints), and solver algorithm considerations.

2.1 Modeling

The FEM was developed within ANSYS™ software [7]. The clavicle finite element model was anatomically-accurate and consisted of a cortical shell, which surrounded the trabecular bone core of the clavicle (Figure 1a). Figure 1b shows a model of the human clavicle along with major muscles attached to it, as it was depicted within AnyBody™ software. The finite element model used in this paper was a 151 mm clavicle (average of 196 clavicles [8]. The clavicle was modeled using ANSYS Solid187 elements to represent the cortical and trabecular bone. The cortical thickness ranged from 1.5-2 mm throughout the model. The model consisted of 25,000 elements and 100,000 nodes. The computer aided model’s size, cortical thickness etc. may be varied parametrically.

![Finite Element Mesh of Clavicle and Model of Clavicle in AnyBody™ software](image)

2.2 Material Models

Elastic bone properties and orthotropic bone properties were obtained from [9-12]. The cortical and trabecular bone were considered to be either isotropic, orthotropic and viscoelastic in respective analyses. The orthotropic material properties for bone are shown in Table 1. Viscoelastic properties were implemented via a Prony-series expansion of the cortical bone shear-modulus (Table 2).

| Table 1. Bone Material Properties Used for the Linear Elastic Loading Case |
|------------------------|-----------------|-----------------|
| Linear Material Properties of Bone | Cortical | Trabecular |
| E (MPa) | 14,200 | 445 |
| \(v_{xy}\) | .46 | .4 |
| \(G_{xy}\) (MPa) | 6,700 | 5 |
| Density (kg/m\(^3\)) | 1850 | 290 |
| Damping Ratio | .146 | .058 |

The resulting complex shear modulus \(G^*\) is presented in Figure 2.

![Complex Viscoelastic Shear Modulus](image)

2.3 Boundary Conditions

Two clavicle loading conditions were studied: the first (loading case 1) was used for model validation and the second (loading case 2) was used to compare displacement-prediction sensitivity to material model choice (i.e. elastic vs. viscoelastic). The first loading
condition was transient, axially-oriented (along clavicle longitudinal axis) and was obtained from literature [3] (see Figure 3 and 4). In load case 1, in order to simulate a compressive impact, the sternoclavicular end of the clavicle was restrained in all degrees of freedom and the acromioclavicular (lateral) end of the clavicle was restrained from Y,Z, motion but allowed to displace in X (the long axis of the clavicle). The same boundary condition was used for loading case 2. The first loading condition was applied with a 1.5 m/s speed, which represents a strain rate of $5s^{-1}$, which could be construed as a short fall on the shoulder [14]. The second loading case simulated a very short duration (0.008 s), half-sine, blunt-body impact, and was applied similar to the first force (Figure 4). Separate analyses, under the two loading conditions, were conducted for FEM which considered separately viscoelastic, orthotropic and isotropic elastic bone material behavior, respectively.

![Figure 3 Finite Element Mesh of with Constraints and Axial Force: 1st Loading Case](image)

The results obtained in the analyses were compared in order to determine the influence of viscoelastic properties on the resulting stress/strain distribution in the bone. Modal analysis was performed to ensure that the dynamic displacement response of the model was captured with an appropriately sized time step.

2.4 Numerical Solver Considerations

In order to capture the effects of both frequency-dependent viscoelastic material behavior and frequency-dependence of dynamic forces within the predicted stress response, the finite element solver had to have a sufficiently small time step. A time step that would be less than 1/20 of the natural period desired to be captured in the response is generally recommended for implicit solvers, such as the Newmark algorithm used in this analysis. For this purpose, a preliminary modal analysis was performed with the same FEM in order to obtain a quick analysis of the modes that would be excited by the dynamic forces (Figure 5).

The modal analysis was performed first for a free (unconstrained) model and secondly for model with both ends of the clavicle restrained. This analysis was performed to verify validity of modal analysis results, as more constrained models generally have higher natural frequencies than unconstrained ones for bending modes. The solver used with the ANSYS software was the Block Lanczos method. These models produced the three mode shapes seen in Figure 5, and the corresponding natural frequencies values and periods are shown in Table 3.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Unconstrained Frequency (Hz)</th>
<th>Constrained Frequency (Hz)</th>
<th>Constrained Period $T_c$ (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>907</td>
<td>1199</td>
<td>0.00083</td>
</tr>
<tr>
<td>2</td>
<td>1135</td>
<td>1908</td>
<td>0.00052</td>
</tr>
<tr>
<td>3</td>
<td>2183</td>
<td>2289</td>
<td>0.00044</td>
</tr>
</tbody>
</table>

Spectra of the two forces (loading case one and loading case 2) in the frequency domain were also obtained (Figure 6). Figure 6 and Table 3 shows that the dynamic loads would put energy in the first bending mode only, since the main energy of the loads was found to be mainly in a bandwidth of less than 200 and 400 Hz, respectively. Of course, this observation would be generalized to the bone material considered in the studies (i.e. a less dense bone may have modes that would have lower natural frequencies than those presented in Table 3). The results of the modal analysis, coupled to the spectra of the loading conditions indicated that it would be the first bending mode shape of the clavicle that would be mainly be seen in the clavicle response to both loading conditions. While a time step of 0.0004 s could have been chosen based on the above considerations, a time step of 0.0001 s was conservatively used in order to assure capturing of the viscoelastic nature of the bone material, when it was used in the material model.
3. Results and Discussion

3.1 Validation: Loading Case 1

The validation case corresponded to the 1st loading case, described in the previous section. The developed FEM model was validated by comparing the deflection (measured at the middle of the clavicle) in the longitudinal direction of the clavicle (considered to be the X axis) to experimental results obtained by other researchers [4]. There was good agreement between experiment (which stated deflections of 5.4 ±1.1 mm), to finite element model prediction (5.95 mm), as shown in Figure 7. Since in the experiment cited clavicles fractured at 5.4 mm of deflection, strain in the FEM of the clavicle was plotted (Figure 8) to ascertain whether it approached commonly used strain failure values for bone (0.03). As shown in Figure 8, the midsection of the FEM clavicle did exhibit a strain value of 0.03 at time of maximum force. This was another validation of the developed clavicle FEM.

Figure 7 shows a comparison of displacement predictions. It was observed that for this loading condition, the use of viscoelastic material properties within the finite element model did not produce a significant difference in predicted axial deflection to that obtained using strictly elastic and isotropic bone material properties (the curves for linear elastic and viscoelastic are practically superimposed as seen in Figure 9).
3.2 Analysis of Displacement-Prediction Sensitivity to Material Model: Loading Case 2

In this case, a faster, half-sine force, equal in maximum magnitude to 1400 N, was applied to the clavicle. The displacement prediction comparison among the linear elastic, viscoelastic and orthotropic are shown in Figure 10; the displacement was measured in a perpendicular direction to the X axis. The three predictions, corresponding to the three material models showed variations (see Figure 10).
3.3 Discussion

The FEM procedure was validated with experimental results obtained from literature. The relative difference in displacement results was less than 10%, when compared to the average experimental deflection value. It was observed that influence of viscoelastic material properties within the FEM was significant only if displacement and stress time-history due to a short-duration (millisecond or less) impact loading was sought.

Table 4 provides a final comparison of displacement predictions. The results obtained as a result of the second loading (traumatic impact) were less by 4.5% if a viscoelastic bone material model was used vs. an elastic bone material model. The influence of viscoelastic behavior was thus clearly shown. Orthotropic material properties lead to a smaller displacement response in both loading cases compared to the same results obtained using a linear elastic model. This was to be expected since bone is generally weaker in the radial and transverse directions than in the longitudinal one.

<table>
<thead>
<tr>
<th>Material model</th>
<th>Longitudinal Deflection in mm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Duprey (validation) Loading</td>
</tr>
<tr>
<td>Linear Isotropic</td>
<td>-5.948</td>
</tr>
<tr>
<td>Orthotropic</td>
<td>-6.151</td>
</tr>
<tr>
<td>Viscoelastic</td>
<td>-5.948</td>
</tr>
</tbody>
</table>

4. Conclusion

The current work validated a FEM model for dynamic loading of the clavicle which takes into account viscoelastic and orthotropic material properties. The effect of viscoelastic, orthotropic and elastic material properties on clavicle displacement predictions was investigated. Future work will include investigations of fixation plate geometry, fixation screw type on clavicle stress distribution during recovery, under both impact loading and under physiological loading. The proposed work would be based around parametric clavicle-plate-screw models (Figure 11).

References


