An advanced course on finite element analysis, with application to the stress distribution in teeth

Eric Yoon  
*Washington University in St. Louis*

Guy M. Genin  
*Washington University in St. Louis*

Follow this and additional works at: https://openscholarship.wustl.edu/mems500

**Recommended Citation**
https://openscholarship.wustl.edu/mems500/72

This Final Report is brought to you for free and open access by the Mechanical Engineering & Materials Science at Washington University Open Scholarship. It has been accepted for inclusion in Mechanical Engineering and Materials Science Independent Study by an authorized administrator of Washington University Open Scholarship. For more information, please contact digital@wumail.wustl.edu.
An advanced course on finite element analysis, with application to the stress distribution in teeth

DongHwan Yoon
Department of Mechanical Engineering and Materials Science
Washington University, St. Louis, MO 63130

yoondh91@gmail.com

May 08, 2018

Abstract

The overall goal of my work is to gain insight into how tooth shape relates to its function. As a step towards this, I undertook an independent study project to further improve my skills on finite element analysis (FEA) this semester, and to combine this into my Master’s thesis project work. Continuing from the previous independent study course, the tooth model was improved to eliminate singularities and a contact surface model was included to simulate contact stress problems. I believe that these series of problems will be useful to my research. This report contains an overview of some literature that I studied, and a summary of several finite element output plots that I found to be particularly instructive.

1. Introduction

The context in which this study was undertaken is the attachment of tendon to bone, which is a major challenge from the surgical, mechanical engineering, and tissue engineering perspectives [1-3]. For surgery, up to 94% of rotator cuff reattachments fail [4]. From the mechanical engineering perspective, the mechanisms of resilience at the insertion site are an area of ongoing research [5-11], and must overcome the free edge singularity problem [12-29]. From
the tissue engineering perspective, the natural tendon to bone attachment does not grow back [4], and it is important to find ways to stabilize tissue without this attachment [20-23] and to guide regrowth of the transitional tissue [24-28]. Stabilization of tissue during healing is a topic that I am focusing on and have contributed to a conference paper on [29]. The question of resilience of tissues motivated my ongoing study of how carnivores capture and tear through flesh.

As a step towards this, I studied some basic solid mechanics, including some specialized problems from the textbook by Budynas [30], and studied an introduction to finite element analysis [31].

In this report, I present a few finite element results that overcome the issues from the previous independent study and demonstrate the results that could be useful for my Masters research.

2. Methods

The numerical portion of the study was conducted using the finite element method, and using commercial software (Abaqus/CAE) for the analysis. The steps involved in a finite element analysis are coming up with an idealized geometry, assigning idealized material properties, choosing boundary conditions, making a mesh, implementing the boundary conditions, solving the equations (equilibrium, strain displacement, and constitutive equations) by a matrix-based energy minimization method, and then validating results by mesh refinement [31].

Numerical simulations were performed to assess how teeth might be optimized to switch from cutting teeth that induce high principal stresses on an isotropic continuum to trapping teeth that induce compression of an isotropic continuum against a rigid simulated gumline. The first step in this study was a review of basic solid mechanics solutions for curved beams [30]. Thereafter, I evaluated how teeth both stress and constrain soft tissues. As mentioned above, the
goal was to determine what shapes lead to high stresses at tooth tip, and what shapes lead to constriction of the soft tissue against a rigid gumline.

Teeth were modeled parametrically to shift from a nearly pyramidal canine to a hooked python-like tooth. Analyses were performed under plane strain conditions. Each tooth was treated as a pair of splines that intersected at a curved top. The teeth were each of a base \( w \) and a height of \( 2w \). The curvature of the tooth was determined by moving the tip (sharp end) to the right in the Abaqus/CAE sketch interface, thereby increasing the distance \( w' \) (Figure 2). The tooth has a dimension of \( 2 \times 4 \times 1 \) (length, height, width) in arbitrary units (in this study mm). The top region of the tooth from the left end to the tip is referred to \( w' \) and the base of the tooth is referred to \( w \). The parameter that determines the degree of curvature can be expressed as \( w'/w \) where in this study ranges from 0.5 to 2.25.

![Figure 1. General view of tooth model with annotations of \( w \) and \( w' \)](image)

A tissue was placed over the tooth (Figure 3). The right and left boundaries of the tissues, placed a distance \( 4w \) away from the middle of the tooth. The height of the tissue was \( 4w \) away from the base of the tooth. A gumline was placed at the bottom of the tooth and was assigned the same material properties as the tooth, described below. The tooth/gumline and the tissue were not allowed to interpenetrate.
The problems studied here were idealized teeth on an elastic foundation contacting with a softer material, which would ideally resemble a tendon. As a first approximation, the teeth and softer tissue were modeled as linear hyperelastic and as isotropic. The hyperelasticity was irrelevant for the tooth due to its high relative stiffness and strength. Also, the tooth was modeled as a solid rather than multilayered structure due to the stresses that were very small compared to its failure strength: the tooth was effectively rigid compared to the soft tissue. The Young’s modulus and Poisson’s ratio were set to 14 GPa and 0.3 for the tooth, respectively. These values correspond to human cortical bone which I have used as a reference material that would allow me to gain insight [32-36].

The models were two dimensional, and plane strain, linear interpolation quadrilateral and triangular elements were used.

Abaqus was used to refine the mesh until the strain energy and peak principal stress in a model did not change more than a few percent with additional refinement. The corresponding plots of the maximum principal stress, strain tensor energy and the strain energy density were studied.

The mesh size can be controlled through the graphical user interface in Abaqus/CAE. However for the purpose of this study the finer upper is the part which simulates the soft tendon, is the part of more interest, therefore used a finer mesh (Figure 3). The bottom part which resembles the tooth has a larger mesh. The graphic interface in Abaqus/CAE allows the user to change the mesh size and element type (quadratic or linear). For the purpose of this study, quadratic elements were used.
Abaqus was used to refine the mesh until the strain energy and peak stress did not change more than a few percent with additional refinement. The corresponding plots of the maximum principal stress, strain tensor energy and the strain energy density will be attached.

3. Results and Discussion

Results showed that stresses in the tendon were highly localized to the tip of the tooth, with stress concentrations well above 10 at the contact point (Figure 4). This is consistent with the sharp nature of the rounded tip of a tooth, and is expected for an appendage that must penetrate tissue. In subsequent analyses, the objective was to determine the degree to which changes to the tooth affected the degree of this stress concentration. The deformed shape of the tendon and tooth model implies that the model created acts as expected. (Figure 5).

The maximum tensile principal stress follows what would be expected in a cantilever beam with the boundary conditions used (Figure 2). For the curved tooth, the tensile stresses were in general higher on the loaded face, and the principal stress was zero on the back face, consistent with what is expected for flexure of a beam [30]. Two artifacts appear. The first is a stress concentration at the point that was fixed, in the lower left hand corner. This arose because of the choice made to have rollers on the bottom boundary and one fixed point. However, in other simulations where the bottom boundary was “encastré” [31], meaning that the displacement was fixed to zero, a stress concentration known as a Williams free-edge singularity appeared at that corner [44]. The stress concentrations or stress singularities can be suppressed by choosing different boundary conditions, such as a foundation that is elastic in shear or a cohesive zone model, which is used in fracture studies [31,46]. Although the understanding of these mechanisms falls under multi-scale modeling that is beyond the scope of what is needed for this study, phenomenological models can be used to account for how microstructure relates to continuum behavior [45-46]. The second is an hourglass effect [31]. Here, the oscillatory
nature of the free edge singularity shows up as a series of errors in the estimation of displacements, which makes neighboring quadrilateral elements look like hourglasses [31]. The hourglass effect can be suppressed by choosing elements with “hourglass control” or by choosing triangular elements [31].

The results up to this point showed that increasing the tooth length and curving the tooth more towards a python shape caused an increase in the contact area and improved the normal force. The most important aspects for an optimal shape were to have a firm grip on the tendon, while having low stress values. In order to relieve edge effects, the tooth model was set to penetrate the tendon model until half its total height. As expected the highest region of stress was found to be near the tip area.

Figure 2. Maximum principal stress contour on tooth model of $w'/w = 1.5$
Figure 3. Normal strain in the vertical direction of tooth model of $w'/w = 1.5$

Figure 4. Stress contour of maximum normalized principal stress of $w'/w = 1.5$
Figure 5. Deformed tendon and tooth model with scale factor of 30
4. Conclusions

I am confident that I have become proficient in using Abaqus in order to create and analyze multiple models in contact. This skill sets that I have acquired during the semester shall contribute to completing a Master’s thesis project which is the learning objective of the study.

5. Acknowledgments

I thank my mentors Guy Genin, Victor Birman, and Stavros Thomopoulos, and also my labmate Steve Linderman, for their guidance on this study.
References


A.E. Anderson, Complete Tyrannosaurus rex skull, AMNH 5027, American Museum of Natural History, 1912.


