ABSTRACT

JADELIS, CHRISTOPHER THOMAS. Index Finger Extensor Hood Mechanics Using Finite Element and Rigid Body Dynamic Modeling (Under the direction of Dr. Katherine Saul).

Function of the hand and fingers is important for the performance of most daily activities, with manual dexterity being an important indicator for detection and measurement of disability (Duruöz, 2014). Understanding of the underlying mechanics of hand function has been dramatically improved with implementation of biomechanical models, with important information such as passive joint stiffness and muscle force contributions derived from biomechanical models that would otherwise not be possible to derive from experimental results alone (Kamper et al., 2002). However, the hand and fingers present unique challenges in biomechanical modeling in comparison to other limbs, due to their relatively small masses and inertial components (Binder-Markey and Murray, 2017).

One specific component of the hand that presents unique challenges to modeling is the extensor hood aponeurosis that governs the extension of the fingers. This is due to the extensor hood forming a sheath-like surface of tendinous structures and soft tissues comprised of heterogenous materials (Qian et al., 2014). Previous modeling approaches have been implemented, such as modeling the extensor hood as a tendinous rhombus (Valero-Cuevas et al., 1998; Winslow, 1732). However, this approach is limited as it must assume the extensor hood is comprised of homogenous materials (Lee et al., 2008; Qian et al., 2014).

This thesis examines and attempts to validate two different approaches to modeling of the extensor hood aponeurosis: the first approach uses an existing rigid body dynamics model, and the second a newly developed integrated model that combines finite element modeling with rigid body dynamics to simulate the heterogenous nature of the extensor hood. Two models were used in this study. The first model is a previously developed dynamic model of the hand, index finger,
and thumb that has been validated using cadaveric data for isometric fingertip force production (Barry et al., 2018). The second model used in this study is a modified version of a finite element model of the index finger extensor hood and phalanges (Ellis et al., 2011), that implements the heterogenous nature of the extensor hood using material stiffnesses derived from cadaveric dissections (Qian et al., 2014). This study attempted to validate each model using experimental data, with isometric fingertip forces and experimentally acquired material strains used for validation of the dynamic model and finite element models respectively. Lastly, this study compares the isometric fingertip forces developed using the integrated finite element and rigid body dynamic model to the original dynamic model and experimental data to evaluate the performance of the integrated model.

Validation of the dynamic finger model demonstrated a similar pattern of predicted force magnitudes to experimental data within 0.53N, but the orientations did not match the experimental data within one standard deviation. The validation of the finite element model demonstrated similar longitudinal strains to the experimental data, with longitudinal strains at the central and terminal slips lying within one standard deviation of the experimental data; however, lateral strains were underpredicted. Evaluation of the integrated model similarly did not match the experimental results, apart from the force magnitude produced using the first palmar interosseous muscle, which replicated the experimental data within one standard deviation. Discrepancies between the dynamic model results and experimental data may be due to unwanted joint motion during simulation, whilst the discrepancy with the finite element model may arise due to the material model assumptions used. To improve the results from these models, a sensitivity study of joint motion constraint and acquisition of more accurate materials data are necessary.
Index Finger Extensor Hood Mechanics Using Finite Element and Rigid Body Dynamic Modeling

by
Christopher Thomas Jadelis

A thesis submitted to the Graduate Faculty of North Carolina State University in partial fulfillment of the requirements for the degree of Master of Science

Mechanical Engineering

Raleigh, North Carolina 2020

APPROVED BY:

Dr. Katherine Saul
Chair of Advisory Committee

Dr. Derek Kamper

Dr. Scott Ferguson

Dr. Marie Muller
BIOGRAPHY

As a biomedical engineering student during my undergraduate education, I had long been fascinated with the many ways in which technology allows people to move and function.

Following my graduation in 2017, I decided to pursue a master’s degree at North Carolina State University in mechanical engineering with a focus in biomechanics. I was fortunate to have had the opportunity to conduct research in the Movement Biomechanics Laboratory under the direction of Dr. Katherine Saul. My graduate research focused on modeling approaches to force production in the extensor hood apparatus of the index finger.
ACKNOWLEDGMENTS

Firstly, I would like to give special thanks my advisor Dr. Katherine Saul for giving me the opportunity to pursue research in her lab over the past year and half. Her guidance and expertise in biomechanical modeling has made this project possible, and I am very honored to have been a part of her lab.

I would also like to thank Dr. Derek Kamper for his support during this project, and for providing me with many of the models used in this project. His advice and encouragement has been invaluable to the completion of this project. Additionally, I would like to thank my committee members Dr. Scott Ferguson and Dr. Marie Muller for their support and help in reviewing my research.

I would also like to thank Dr. Benjamin Ellis at the University of Utah for being very accommodating and helping to answer my questions about the finite element model of the finger that he had developed. His support and advice helped me to update and use his finite element model in this project and I would not have been able to complete this thesis without his help. I would also like to thank Dr. Steve Maas, Dr. Gerard Ateshian, and all the other members of the FEBio team for all their help in answering my questions on the FEBio forum.

Lastly, I would also like to thank all the members of the Aerosol Science Lab at UNC-Chapel Hill for starting me on the path of academic research, I don’t believe that I would be where I am today without their help. As well as to all my friends in the UNC BME class of 2017, I would not be where I am today without you.
# TABLE OF CONTENTS

LIST OF TABLES .................................................................................................................... viii
LIST OF FIGURES .................................................................................................................... ix

CHAPTER 1: INTRODUCTION ............................................................................................... 1

CHAPTER 2: METHODS .......................................................................................................... 6
  2.1 Co-Simulation Platform ............................................................................................ 6
  2.2 Dynamic Model of the Index Finger ....................................................................... 10
  2.3 Development of the Finite Element Model of the Index Finger ......................... 14
  2.4 Test Simulations ....................................................................................................... 18

CHAPTER 3: RESULTS .......................................................................................................... 20
  3.1 Platform Stability Analysis ..................................................................................... 20
  3.1 Dynamic Model Preliminary Evaluation ............................................................... 21
  3.2 Finite Element Model Validation ........................................................................... 23
  3.3 Co-Simulation Performance Analysis .................................................................... 27

CHAPTER 4: DISCUSSION .................................................................................................... 29
  4.1 Summary ................................................................................................................... 29
  4.2 Discussion .................................................................................................................. 30
  4.3 Limitations ................................................................................................................ 33

CHAPTER 5: CONCLUSION .................................................................................................. 38
  5.1 Conclusion ................................................................................................................ 38
  5.2 Future Work .............................................................................................................. 38
LIST OF TABLES

Table 1  Model Coordinate Mapping ....................................................................................... 9
Table 2  Dynamic Model Index Finger Muscle Parameters .................................................. 11
Table 3  OpenSim Simulation Forces Parameters Template ................................................. 13
Table 4  Extensor hood Experimental Material Properties .................................................... 16
Table 5  Extensor hood Central Region Longitudinal Strains (%) ......................................... 26
Table 6  Dynamic Model Differences From Original ............................................................ 34
Table 7  FEBio Available Force Outputs ............................................................................... 37
LIST OF FIGURES

Figure 1.1 Winslow’s Tendinous Rhombus adapted from (Valero-Cuevas et al., 1998) ........... 3
Figure 2.1 Integrated Model Information Exchange ............................................................... 8
Figure 2.2 Finite Element Model Force Application ................................................................. 9
Figure 2.3 Dynamic Index Finger Thumb Model ................................................................. 11
Figure 2.4 Index Finger Finite Element Model Material Partitions ........................................ 16
Figure 2.5 Modified Dynamic Hand Finger Thumb Model ................................................... 19
Figure 3.1 Dynamic Model Contact Method Comparison Force Magnitudes ................. 21
Figure 3.2 Dynamic Model Contact Method Comparison Force Vectors ......................... 22
Figure 3.3 Experimental Strains (Lee et al., 2008) ............................................................... 24
Figure 3.4 Finite Element Model Central Region Validation ............................................. 25
Figure 3.5 Finite Element Model Insertions Longitudinal Strains ................................. 26
Figure 3.6 Integrated Model Evaluation Force Magnitudes ........................................... 27
Figure 3.7 Integrated Model Evaluation Force Vectors ..................................................... 28
CHAPTER 1: INTRODUCTION

Function of the hand and fingers is essential for completing most daily activities, with manual dexterity regularly used as an indicator of the severity of disease, as well as a measure of recovery from disability (Duruöz, 2014). Improved understanding of the musculoskeletal mechanics that govern the hand and fingers has been derived from anatomical experiments that explore tissue mechanics and hand function, with musculoskeletal modeling being used to directly link these two domains. One example of the link between tissue mechanics and overall hand function is the determination of the ability of the extrinsic flexors to initiate flexion concurrently in each of the index finger joints, and the derivation of passive joint stiffnesses for those joints (Kamper et al., 2002). In this earlier work, experimentally acquired anatomic measurements of the index finger and joint motion resistances were used to drive a computational model of the index finger and tendons to determine passive joint stiffness and the ability of the extrinsic flexors to induce concurrent flexion for all of the index finger joints.

Integration of tissue mechanics and overall motion is essential for developing appropriate rehabilitative therapies to restore function, and requires accurate measures of the behavior of many of the complex structures in the hand, including joints, muscles, and ligamentous structures. Biomechanical modeling is an efficient way to integrate experimentally-derived information into a single mathematical platform for simulating injury and rehabilitative therapies. However, biomechanical models of the hand and fingers have unique challenges in comparison to models of other limbs. For example, the hand and fingers are significantly lighter and smaller, and thus possess less inertia (Binder-Markey and Murray, 2017). Due to this, the mass and inertial contributions to motion have relatively lower impact, and the viscoelastic forces presented by soft tissues generate more noticeable contributions to damping. Additionally,
musculature of the hand is comprised of extrinsic muscles originating in the wrist, as well as intrinsic muscles within the hand, all of which must cross multiple joints and insert into the various soft tissue structures within the hand. This makes muscle moment arms and passive joint properties highly dependent on each joint’s posture.

Existing biomechanical models have addressed some of these challenges using highly articulated models of the hand (Barry et al., 2018; Binder-Markey and Murray, 2017) directly representing muscle force-length-velocity properties and passive joint properties to address the challenge of muscles crossing multiple joints and the low inertia-high damping regime. The Binder-Markey model accomplishes this by modeling passive joint torques as function-defined torsional springs, and modeling finger muscles with effectively constant moment arms that match experimentally-derived moment arms to address the challenge of crossing multiple joints (Binder-Markey and Murray, 2017). The Barry model utilizes a similar approach, with function-defined bushing forces applied to mimic posture-dependent joint properties; however, the Barry model is only defined for the index finger and thumb, and includes interosseous muscles that are not present in the Binder-Markey model. These models have been validated using existing cadaveric datasets, for example to validate model predictions of fingertip force of the index finger and thumb during isometric muscle contraction (Barry et al., 2018) against cadaveric recordings (Qiu, 2014). These models have been applied to study musculoskeletal impairments, including contributions of extrinsic flexor muscles to claw finger deformity in patients with neural impairments (Binder-Markey et al., 2019). This work illustrates how rigid body dynamics simulation can accurately capture musculoskeletal dynamics of the hand and finger as driven by muscle-tendon actuators and yield important information on the nature of disability.
An important limitation of many of these hand and finger models however, is a simplified representation of the geometry of the anatomical structures of the hand, using empirically-derived passive force curves and single line of action muscle-tendons in order to enable rapid computation of dynamic behavior of the hand. This neglects the effects of muscles inserting into soft tissue structures, such as the extensor apparatus of the fingers, rather than the finger bones directly. The extensor hood aponeurosis is complex, with a sheath-like surface of soft tissue that acts as an extension of the extensor digitorum communis (EDC). The surface also connects to tendinous structures of the extensor indicis (EI), lumbricals (LUM), and interosseous muscles. The extensor hood has two primary insertion sites into the phalanges: the central slip which inserts at the proximal end of the medial phalanx, and the terminal slip, which inserts on the proximal end of the distal phalanx. The extensor hood additionally includes attachments from the transverse and oblique bands of the rectilinear ligaments (Haines, 1951). Past mathematical representations of this surface have used a network of straight tendinous paths in the shape of a rhombus (Figure 1.1) to represent each potential force direction (Winslow, 1732). This model assumes homogenous material properties; the extensor hood however, is comprised of heterogenous materials, thereby limiting the applicability of this model (Lee et al., 2008; Qian et al., 2014).

Figure 1.1 Winslow’s Tendinous Rhombus, adapted from (Valero-Cuevas et al., 1998)
To address the complex behavior of the extensor hood, physical experiments in cadaveric tissue have quantified the effects of finger posture change on muscle moment arms and passive joint torques as a result of extensor hood deformation (Kamper et al., 2006). However, a direct link to active function is still needed since these experiments in cadaveric tissue inherently lack the capacity to capture active muscle behavior such as possible coactivation and muscle synergies. Finite element modeling to simulate mechanical behavior of these tissues is promising and has been used to capture soft tissue and joint motion in other complex musculoskeletal structures, such as the foot (Halloran et al., 2010). However, this approach limits the extent to which coordinated dynamic motion of the hand can be modeled due to the computational demands of a multi-muscle dynamic system within a finite element environment (Teran et al., 2005).

Therefore, the goal of this work was to develop and test a software modeling approach to directly link the mechanics of the complex soft tissue structures of the hand and fingers to coordinated hand function. To do so, we developed a framework to integrate two biomechanical models representing the index finger and its extensor hood via cosimulation in both dynamic and finite element environments. The dynamic model extends an existing musculoskeletal model of the index finger implemented in OpenSim 3.3 (Barry et al., 2018; Delp et al., 2007). This dynamic model is integrated with a finite element model of the index finger and its extensor hood implemented in the FEBio software suite (Maas et al., 2012).

This cosimulation approach provides a framework for modeling the role of the extensor hood in the mechanics and function of the hand, considering both the musculoskeletal dynamics and deformation models. Here, we describe the development of the cosimulation framework, test whether simulations are able to successfully pass information between platforms, and perform
initial evaluation of the performance of each platform alone and integrated via cosimulation. As test data, we predicted fingertip force during isometric contraction of index finger muscles.
CHAPTER 2: METHODS

2.1 Co-simulation platform

2.1.1 Overview

To facilitate exchange of information between the models described in the later sections 2.2 and 2.3, a custom MATLAB script (MATLAB r2019b, MathWorks, 2019) was developed (Figure 2.1). The purpose of this script is to be able to read the storage files containing the joint reaction forces, and muscle forces, that are generated using OpenSim (Delp et al., 2007), and extract the tendon force to be used as an input in the generation of a finite element preprocessor file for the model described in section 2.3. After running the finite element solver using the tendon forces as prescribed loads, the script extracts the rigid body reaction forces acting on the phalanges that arise from the extensor hood insertions and ligamentous adhesions. These reaction forces are then exported and applied as external forces in OpenSim to execute a forward dynamics simulation from which fingertip endpoint forces can be predicted and compared against the non-integrated dynamic model, and experimental data.

2.1.2 OpenSim File Importation and FEBio File Generation

To import the storage files from the OpenSim simulations, the OpenSim MATLAB API was used. The storage files containing the muscle tendon forces were imported as storage objects as defined by the API, and the tendon force column was extracted from the force reporter storage object. To apply the tendon force as a load to the finite element model, a template preprocessor file containing the finite element model (section 2.3) was imported into MATLAB using the built-in XML file tools. These were then used to apply the desired tendon force as a nodal load on the corresponding tendon for that simulation. Lastly, an external function called xml_xerces,
provided by the xml_io_tools library, was used to export the newly modified finite element preprocessor file (Tuszynski, 2020).

2.1.3 OpenSim External Forces Generation

To generate the external forces file for use in OpenSim, the log files generated by FEBio were read using the `importFEBio_logfile` function from GIBBON (Moerman, 2018). The log file for each rigid body was imported, and the reaction force vector observed at the center of mass of each rigid body was averaged over the period of time that the tendon force was applied, as shown in Figure 2.2. The forces were then mapped to the OpenSim global coordinate frame for the dynamic model (Table 1). A blank storage object was then created, with a time column matching the length of the desired forward dynamics simulation. A two-second duration was chosen to ensure the model reached steady state behavior. As this set of simulations is static, the remaining columns in the storage object were populated by the average force vectors for each rigid body, with the average value repeated for each time step. Once all columns were populated, the storage file was written as a `.mot` file for use in the forward dynamics simulation.
Figure 2.1 Integrated Model Information Exchange
Table 1. Model Coordinate Mapping

<table>
<thead>
<tr>
<th>OpenSim</th>
<th>FEBio</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>-x</td>
</tr>
<tr>
<td>y</td>
<td>-z</td>
</tr>
<tr>
<td>z</td>
<td>-y</td>
</tr>
</tbody>
</table>

Figure 2.2 Finite Element Model Force Application
This figure demonstrates the location and direction of forces applied on the finite element model. The top panel of this figure demonstrates the EDC with an activation of 10% extending the finger, while the bottom panel demonstrates the location where the equivalent tendon force developed in the top panel is applied to the finite element model.
2.2 Dynamic Model of the Index Finger

2.2.1 Dynamic Model Overview

An existing dynamic model of the index finger, thumb, and distal upper limb was used as a foundation for the current study (Barry et al., 2018) as shown in (Figure 2.3). This model was implemented in OpenSim 3.3 (Delp et al., 2007) and includes 18 muscles (7 crossing the index finger and 11 crossing the thumb). Muscle force-generating capacity is described using the Millard2012 equilibrium muscle model as described in (Millard et al., 2013), with muscle parameters derived from literature (Jacobson et al., 1992; Lieber et al., 1992; Pearlman et al., 2004; Saul et al., 2015; Stamenkovic et al., 2014) for the index finger (Table 2) and thumb. For the current study, only the index finger is considered and the muscles for the thumb are disabled, as the model integration described in later sections is designed initially to simulate the index finger and its extensor muscles. To implement the posture-dependent passive joint properties, a series of function-driven bushing forces were implemented at the joints for the index finger using equations as described in (Kamper et al., 2002). The location of the tip of the index finger was constrained in space using the Hunt-Crossley contact method applied to an 8-mm diameter spherical surface attached to the fingertip via a weld joint. A series of constraining half-planes were set to interact with the sphere, forming a hollow cube surrounding the fingertip sphere. Each side of the cube measured 9mm. To confirm whether fingertip endpoint force predictions are sensitive to implementation of fingertip constraint, we also implemented a new constraint in place of the half planes, using a hollow 9-mm inner diameter sphere placed at the tip of the finger. This constraint was then assigned as a contact mesh using the elastic foundation force model (EFF). The existing 8-mm sphere at the fingertip was defined to collide with this contact mesh. A 0.5-mm gap between the two spheres was chosen to ensure a stable forward dynamics
simulation. Contact parameters were defined to be the same as in the original model for both Hunt-Crossley and EFF methods, with a contact stiffness of 20 GPa, a dissipation factor of 0.99, a static friction coefficient of 0.1, a dynamic friction coefficient of 0.03, and a viscous friction coefficient of 0.6, as described in (Barry, 2016).

![Figure 2.3 Dynamic Index Finger-Thumb Model](image)

**Table 2:** Dynamic Model Index Finger Muscle Parameters (Barry, 2016)

<table>
<thead>
<tr>
<th>Muscle</th>
<th>$f_{\alpha,m}$ (N)</th>
<th>$l_{M}$ (m)</th>
<th>$l_{S}$ (m)</th>
<th>$a$ (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flexor Digitorum Profundis (FDP)</td>
<td>79.12</td>
<td>0.075</td>
<td>0.26812</td>
<td>7</td>
</tr>
<tr>
<td>Flexor Digitorum Superficialis (FDS)</td>
<td>58.9</td>
<td>0.084</td>
<td>0.29932</td>
<td>6</td>
</tr>
<tr>
<td>First Dorsal Interosseous medialis (FDIm)</td>
<td>72.8</td>
<td>0.030</td>
<td>0.02520</td>
<td>15</td>
</tr>
<tr>
<td>First Dorsal Interosseous lateralis (FDII)</td>
<td>72.8</td>
<td>0.033</td>
<td>0.02569</td>
<td>15</td>
</tr>
<tr>
<td>Extensor Digitorum Communis (EDC)</td>
<td>34.69</td>
<td>0.070</td>
<td>0.26468</td>
<td>3</td>
</tr>
<tr>
<td>First Palmar Interosseous (FPI)</td>
<td>30.94</td>
<td>0.031</td>
<td>0.08978</td>
<td>6.3</td>
</tr>
<tr>
<td>Extensor Indicus (EI)</td>
<td>35.47</td>
<td>0.059</td>
<td>0.18479</td>
<td>6</td>
</tr>
<tr>
<td>Lumbrical (LUM)</td>
<td>25.04</td>
<td>0.068</td>
<td>0.05470</td>
<td>1.2</td>
</tr>
</tbody>
</table>
2.2.2 Dynamic Model Sensitivity Test

This model was previously validated for fingertip force prediction (Barry, 2016). To test sensitivity to constraint implementation, we created similar simulations representing a cadaveric study of fingertip forces (Qiu, 2014), with an index finger posture of 30°, 45°, and 15° of flexion for the metacarpal-phalangeal (MCP), proximal interphalangeal (PIP), and distal interphalangeal (DIP) joints respectively. In the forward dynamics simulation, the MCP, PIP, and DIP joints were constrained to remain within ±5° of the desired posture using a coordinate limit force with an upper and lower stiffness of 20, damping coefficient of 1, and transition factor of 5. To replicate the experimental protocol for the cadaveric study, all muscles in the model were disabled, and then each index finger muscle was enabled and excited individually, using a static activation level corresponding to the force applied to the tendon during the cadaveric study (Qiu, 2014). For the index finger, this corresponded to a 10% activation for all muscles, apart from the first dorsal interosseous muscle (FDI) which was activated at 5% on the lateral and medial heads, with both being used together. An example set of parameters used for these simulations is shown in (Table 3). Resulting forces were observed using the joint reaction analysis tool, with the fingertip weld joint being the joint of interest.
### Table 3. OpenSim Simulation Forces Parameters Template

<table>
<thead>
<tr>
<th></th>
<th>Activation (%)</th>
<th>Disabled? (Y/N)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Muscles</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>EDC1</td>
<td>10</td>
<td>N</td>
</tr>
<tr>
<td>(all others)</td>
<td>0</td>
<td>Y</td>
</tr>
<tr>
<td><strong>Actuators</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(all)</td>
<td>N/A</td>
<td>Y</td>
</tr>
<tr>
<td><strong>Contact Forces</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>contact_force_index</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>contact_force_thumb</td>
<td>N/A</td>
<td>Y</td>
</tr>
<tr>
<td><strong>Other Forces</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MCP2FE</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>PIP2FE</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>DIP2FE</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>(all flex limit)</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>(all AA limit)</td>
<td>N/A</td>
<td>N</td>
</tr>
<tr>
<td>(all others)</td>
<td>N/A</td>
<td>Y</td>
</tr>
</tbody>
</table>
2.3 Development of the Finite Element Model of the Index Finger

2.3.1 Model Definition

A finite element model of the index finger and extensor hood developed at the University of Utah was used as a foundation and modified for use in this study (Ellis et al., 2011). The finite element model was developed using the FEBio software suite (Maas et al., 2012). The geometry of the bones and hood were derived from magnetic resonance (MR) images of a cadaveric hand (Ellis et al., 2011). The MR images were manually segmented (Amira, Visage Imaging Inc., San Diego, CA) into the extensor hood and individual phalanges. The proximal, medial, and distal phalanges were defined as distinct surfaces, whilst the extensor hood was defined as a single continuous surface. The phalanges and extensor hood were meshed (TrueGrid, XYZ Scientific, Livermore, CA), with phalanges treated as rigid bodies and assigned a mesh using constant-strain triangular shell elements, whilst the extensor hood was assigned a deformable mesh using four node quadrilateral shell elements. The extensor hood mesh was partitioned as shown (Figure 2.2) using PreView (Maas et al., 2012).

Tendon insertion sites for the central slip (CS) on the medial phalanx and the terminal slip (TS) on the distal phalanx were modeled using rigid node sets placed on the proximal ends of the distal and medial phalanges, and constrained to move with the rigid bodies. The TS insertion directly joined the extensor hood surface with the rigid node set, while the CS insertion was modeled using discrete tension spring elements (k=100,000 N/mm) to join the extensor hood surface with the rigid nodes. In addition to the tendon insertions, ligamentous insertions were determined via dissection of cadaveric fingers (Ellis et al., 2011) and represented in the model as a series of discrete tension spring elements (k=1000 N/mm) appearing proximally on the radial and ulnar sides of the proximal phalanx (Figure 2.4).
Three tendinous connections were represented in this model: a central tendon representing the EDC and EI muscles, the radial tendon representing the lumbrical that inserts into the index finger, and the ulnar tendon representing the FPI muscle.

2.3.2 Finite Element Model Modifications

Modifications were made to the finite element model for use in this study. Modifications included alterations to material properties, inclusion of rotational degrees of freedom at the index finger joints, redefinition of boundary and initial conditions, as well as updating the model for use with the latest version of the FEBio solver, which at the time of writing was FEBio 3. Material properties used in the original model were based on material stiffnesses determined by (Garcia-Elias et al., 1991). These material properties included stiffnesses and Poisson’s ratios for each material segment. The material stiffnesses in the model used in this study were redefined to utilize material stiffness moduli from (Qian et al., 2014) as shown in (Table 4). This was done as the stiffnesses defined in (Garcia-Elias et al., 1991) were defined in units of N/mm, which is equivalent to spring stiffness and a useful metric; however, it is not equivalent to the tangent modulus of the material which is defined in (Qian et al., 2014), which should provide a more accurate measure of the material stiffness. Rotational degrees of freedom were included in the model via the inclusion of single degree of freedom joints at the joint centers of the MCP, PIP, and DIP joints. This was done to facilitate index finger rotation during simulations using the integrated model described in (Section 2.1). The metacarpal phalanx was anchored by prescribing displacements for each degree of freedom to be zero; this was done to enable measurement of the rigid force on this body. Three sliding contact surfaces were defined between the extensor hood and each phalanx to prevent penetration, as well as to measure the
contact force present on each rigid body due to the extensor hood. Nodal boundary conditions were defined to constrain each tendon from moving vertically or laterally but allowed motion distally with the extensor hood as the finger flexes.

Figure 2.4 Index Finger Finite Element Model Material Partitions

Table 4: Extensor hood Experimental Material Properties (Qian et al., 2014)

<table>
<thead>
<tr>
<th>Extensor hood Sample Location</th>
<th>Tangent Modulus (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lateral Bands</td>
<td>105.38±85.06</td>
</tr>
<tr>
<td>Soft Tendon (Sagittal Band)</td>
<td>54.68±14.13</td>
</tr>
<tr>
<td>Central Slip</td>
<td>100.76±46.77</td>
</tr>
<tr>
<td>Terminal Slip</td>
<td>89.67±48.29</td>
</tr>
<tr>
<td>Extensor Tendon (Central Band)</td>
<td>100.61±50.82</td>
</tr>
<tr>
<td>Extensor Tendon (Proximal to MCP)</td>
<td>76.22±24.53</td>
</tr>
<tr>
<td>Extensor Tendon (Distal to MCP)</td>
<td>53.16±21.24</td>
</tr>
</tbody>
</table>
2.3.3 Finite Element Sensitivity analysis and Validation

To preliminarily validate the finite element model, simulations were performed to replicate experiments measuring the lateral and longitudinal strains in a cadaveric extensor hood (Lee et al., 2008). Strains were measured in the experimental model by loading the EDC tendon with 11.8N with the finger in a posture of 0°, 0°, 0° for the MCP, PIP, and DIP joints respectively, and observing the change in position of motion capture markers (Figure 3.3a). In the finite element model, these conditions were replicated by placing the FE model in the desired posture and applying an EDC tendon load of 11.8N. Resulting maximum, minimum, and average longitudinal (proximal, distal), and lateral (radial, ulnar) strains were measured over the central region of the extensor hood as described in the experimental definition, as well as longitudinal strains at the CS and TS insertions; experimental lateral strains were not reported for the CS and TS. These were then compared to the experimental strains. To evaluate sensitivity to material stiffness, this was performed for the average material stiffnesses, and material stiffnesses one standard deviation higher (high stiffness properties) and lower (soft properties) than the experimentally reported stiffnesses (Table 4).
2.4 Test Simulations

To evaluate the performance of the integration of the finite element model with the dynamic model, test simulations were conducted to ensure simulation stability and compare the fingertip contact forces developed using the dynamic model without finite element integration to those predicted with the integrated model using the co-simulation platform described in (section 2.1). The simulations for the dynamic model with finite element integration used simulation parameters similar to those used in the sensitivity study described in (section 2.2.2). However, only the elastic foundation force contact method was used for simulating the fingertip contact force. The tendon forces developed from these simulations were then used to drive the finite element model as nodal forces applied on the residual tendons in that model. External forces generated from the finite element simulations were then applied to a modified version of the dynamic model (Figure 2.5), with the external forces acting on the center of mass of each corresponding phalanx. The modified version of the dynamic model was created by disabling all the muscles and passive forces in the model, such that the applied external forces are the only forces present. The resulting fingertip contact forces from the integrated model were then compared to the dynamic model results without finite element integration, and to experimental results to evaluate the ability for the integrated model to accurately produce fingertip contact forces.
Figure 2.5 Modified Index Finger Thumb Model
Modified version of the dynamic finger thumb model with passive forces and muscles removed. External forces developed in the finite element model are applied at the locations indicated by the pink markers in the figure.
CHAPTER 3: RESULTS

3.1 Platform Stability Analysis

The co-simulation platform successfully performed simulations integrating both the dynamic and finite element simulation platforms. All test simulations ran to completion and converged on stable solutions with no errors. Forward dynamics simulations with the musculoskeletal dynamic model converged within one minute, with the Hunt-Crossley contact method requiring more computation time than the elastic foundation force method. The Hunt-Crossley method required 50 seconds of computation time for a two-second simulation, while the EFF method needed five seconds of computation time. The finite element model converged within five minutes, with the simulation time for each muscle lasting between four minutes and 40 seconds, and five minutes. Total simulation time for the integrated model was within five minutes and 15 seconds.

Simulations confirmed that the dynamic model performance was not sensitive to constraint implementation for fingertip contact force. There were minimal differences between the Hunt-Crossley and elastic foundation force methods (Figure 3.1). Differences in force magnitude between the contact method implementation were less than 0.02N for each muscle. There were also minimal differences in the direction of the contact force vector produced by each contact method.
3.2 Dynamic Model Preliminary Evaluation

We performed a preliminary validation of the dynamic model implementation using test data from experimental studies of fingertip force (Qiu, 2014). The simulated force magnitude was within 0.53N of the experimental values, with the largest discrepancy being the FPI muscle, with simulated data overpredicting the force magnitude by 0.53N using the Hunt-Crossley method, and 0.51N using the EFF method in comparison to experimental data. However, each of the simulated values, apart from the FDP muscle, were more than one standard deviation from the average experimental values. Only the FDP muscle simulations fell within one standard deviation of the experimental value, with simulation data being 0.17N less than the experimental data for both the Hunt-Crossley and EFF methods. Comparison of simulated data to experimental data yielded differences in direction and magnitude of individual components (Figure 3.2). None of the simulated data was within one standard deviation of the experimentally recorded values.

Figure 3.1 Dynamic Model Contact Method Comparison Force Magnitudes

<table>
<thead>
<tr>
<th>Index Finger Muscle</th>
<th>Hunt-Crossley Contact Model</th>
<th>Elastic Foundation Force Model</th>
<th>Experimental Values (Qiu 2014)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EDC</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>EI</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>FDI</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>FDP</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>FDS</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>FPI</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>LUM</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
Figure 3.2 Dynamic Model Contact Method Comparison Force Vectors
3.3 Finite Element Model Validation

Our preliminary validation of the finite element model over the central region of the extensor hood yielded comparable values to the experimental data for longitudinal strain (Figure 3.3 and 3.4). For the three stiffness cases tested, the soft material properties provided the best result, with a minimum longitudinal strain of -2.99%, maximum longitudinal strain of 6.95%, and an average longitudinal strain of 1.81%. Experimentally observed longitudinal strains for the central region fall between -5% and 5% (Lee et al., 2008). However, longitudinal strain values did not compare as well for the average and high stiffness cases (Table 5). The simulated lateral strains for any of the three material stiffness cases tested underpredicted the lateral strains described experimentally. The soft material case predicted the largest magnitude lateral strains, with a minimum lateral strain of -2.56%, a maximum lateral strain of 1.48%, and an average lateral strain of -0.78%. This is in comparison to the experimental lateral strains, which ranged from less than -15% to a maximum of 5% (Lee et al., 2008).

In the CS and TS insertions, predicted longitudinal strain values were comparable to experimental values for each of the three material stiffness cases tested (Figure 3.5). The longitudinal strains observed at the TS insertion were within one standard deviation of the experimental data for all cases, with the average and high stiffness material models overpredicting the average strain by 0.12% and 0.10% respectively, while the soft material model underpredicted the TS longitudinal strain by 0.41%. For the CS insertion, only the soft material case was within one standard deviation of the experimental values for the CS insertion, underpredicting CS longitudinal strain by 0.94%. Average and high stiffness material models underpredicting the CS longitudinal strain by 2.74% and 3.28% respectively.
Because the soft material case presented values more closely matching the experimental values than the average or high stiffness cases for both the tendon insertion sites as well as the extensor hood surface, the soft material model was chosen for generating the contact forces used with the integrated dynamic model.

Figure 3.3 Experimental Extensor Hood Strains (Lee et al., 2008)
Figure 3.4 Finite Element Model Central Region Strains

Longitudinal and lateral strains over the central region of the extensor hood for each of the material cases tested, with lateral strains reported for the soft material properties (A), average material properties (B), and high stiffness properties (C). Similarly, longitudinal strains are reported for the soft (D), average (E), and high stiffness (F) material cases.
Figure 3.5 Finite Element Model Longitudinal Strains at Slips

Table 5. Extensor Hood Central Region Longitudinal Strains (%)

<table>
<thead>
<tr>
<th>Material Case</th>
<th>Experimental</th>
<th>Soft Properties (-1 SD)</th>
<th>Average</th>
<th>High Stiffness Properties (+1 SD)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum Strain (%)</td>
<td>&lt; -4</td>
<td>-2.99</td>
<td>-1.27</td>
<td>-1.12</td>
</tr>
<tr>
<td>Average Strain (%)</td>
<td>N/A</td>
<td>1.81</td>
<td>0.81</td>
<td>0.57</td>
</tr>
<tr>
<td>Maximum Strain (%)</td>
<td>&gt; 4</td>
<td>6.95</td>
<td>3.06</td>
<td>2.30</td>
</tr>
</tbody>
</table>
3.4 Co-Simulation Performance Analysis

Initial tests of the co-simulation platform using the test fingertip force data predicted larger fingertip contact forces in comparison to the dynamic model alone (Figure 3.6). The magnitude of the fingertip contact forces for the integrated model exceeded one standard deviation of the experimental values, with the exception of the results for the FPI muscle, which was within one standard deviation of the experimental value and overpredicted the experimental force magnitude by 0.19N. The EDC, EI, and LUM results each overpredicted the experimental values by 4.69N, 4.66N, and 1.76N, respectively. Additionally, the force vector orientations produced using the integrated model were outside the range of orientations defined by the experimental values by more than one standard deviation (Figure 3.7)

![Integrated Model Contact Method Comparison Force Magnitudes](image_url)
Figure 3.7 Integrated Model Contact Force Vector Comparison

Integrated Model Contact Force Vector Comparison (Sagittal Plane)

Figure 3.7 Integrated Model Contact Force Vector Comparison
CHAPTER 4: DISCUSSION

4.1 Summary

We implemented a co-simulation platform to combine the simulation capability of two existing biomechanical models, combining the strengths of a dynamic musculoskeletal model of a hand and index finger, with a finite element model of an index finger and extensor hood apparatus. The co-simulation platform successfully performed simulations integrating both the dynamic and finite element simulation platforms. All test simulations ran to completion and converged on stable solutions with no errors. The musculoskeletal dynamic model ran to completion within one minute for both contact methods, with the Hunt-Crossley contact method requiring more computation time than the elastic foundation force method. The finite element model ran to completion within five minutes, and the integrated model additionally converged within five minutes. Initial validation tests suggested that the dynamic simulations alone produced fingertip contact forces that exhibited a similar pattern to experimental contact forces within 0.53 N, but over predicted or underpredicted force magnitude by greater than one standard deviation. The finite element model produced longitudinal strains that were similar to experimentally-observed strains within one standard deviation at both the CS and TS insertions, but did not reproduce experimentally-observed lateral strains over the central region of the extensor hood surface for the three material cases tested. Integrating the two models resulted in stable simulations of both models, and the successful exchange of information between the two; however, the predicted fingertip contact forces that did not match experimental data. Thus, to accurately use the integrated model in future studies, further improvement and tuning of the underlying dynamic and finite element models is required.
4.2 Discussion

4.2.1 Dynamic Model Discussion

Evaluation of the dynamic model to produce fingertip contact forces, and the sensitivity of the model to contact constraint implementation, demonstrated the ability of the dynamic model to produce similar fingertip contact forces to experimental values. The results were not sensitive to contact implementation using the Hunt-Crossley and elastic foundation force methods. This was expected, as the mathematical implementation that is different between the two methods is primarily based on the inclusion of damping. The Hunt-Crossley contact model is a point contact model and uses a modified version of Hertz law (Hunt and Crossley, 1975). The contribution that Hunt and Crossley proposed is the inclusion of a damping term $b$ as shown in (Equation 1), which accounts for energy dissipation during contact which scales with the depth of penetration. The terms $k$ and $n$ are material constants.

$$F_n = k\delta^n + (b\delta^n)\dot{\delta}$$  \hspace{1cm} (1)

This contrasts with the elastic foundation force model, which discretizes the surfaces into a series of springs with a rigid base. This approach is relatively computationally efficient due to only a simple set of equations being required; however, the number of equations required to be solved may scale with the density of the meshes used. As this study was interested only in the static behavior of the index finger and steady state contact forces, the damping term is not used. Thus, the only term differentiating the two methods being the material constant $n$.

The dynamic model predictions with either contact method did not yield results within one standard deviation of experimentally measured force magnitude and differed markedly in the direction of the force vector (Figure 3.2). As this model has been previously validated against the experimental dataset used in this study (Barry et al., 2018), and using the same muscle
parameters as defined in (Table 1) it is likely that the parameters used to define the forward 
dynamics simulation, as well as assumptions made during the setup of the model used in this 
study may explain the discrepancy between this study and previously published literature. This 
may be in part due to the movement of the joints and spacing of the contact surfaces, as some 
motion is required to engage the contact surfaces of the dynamic model and to ensure a stable 
simulation, and the spacing of the contact surfaces may allow for undesired translation of the 
fingertip. The subtle joint motion produced using the forward dynamics simulations in this study, 
as well as small translations of the fingertip may not be equivalent to the motions produced 
during the original validation study, resulting in a different posture and force vector. This is 
notable, as fingertip force production has been shown to be very sensitive to changes in index 
finger joint posture (Valero-Cuevas et al., 1998).

4.2.2 Finite Element Model Discussion

The development and evaluation of the finite element model of the extensor hood and 
index finger resulted in a finite element model that accurately reproduces longitudinal strains at 
the CS and TS insertions and longitudinal strains across the surface of the extensor hood, and 
resulted in stable simulations. Lateral strains were not accurately predicted. One possible reason 
for this discrepancy may be due to the isotropic material model implemented here, as biological 
tissues such as tendons demonstrate nonlinear stress strain curves, and their fibrous composition 
results in anisotropic material properties (Fang and Lake, 2016). Additionally, as the material 
properties chosen for this model were taken from samples tested along their fiber direction (Qian 
et al., 2014), the measured stiffness may have been higher than in the transverse direction, which 
may help to explain why using softer material properties gave better results. As the extensor 
hood apparatus is an extension of the tendinous structures that insert into it, it may be necessary
to utilize an anisotropic material model and determine the tendon fiber orientation to more
accurately simulate lateral strains, as well as to incorporate transverse material properties which
are likely softer than the reported moduli for the along-fiber direction.

4.2.3 Co-simulation Platform Comparison and Test Simulations Discussion

Implementation of the co-simulation platform resulted in successful sharing of
information between the dynamic model and the finite element model. However, the prediction
of fingertip contact forces using this integrated model resulted in contact forces that were higher
than the experimental data. As the modified dynamic model used to generate these contact forces
is driven only by the reaction forces developed in the finite element model, some possible
explanations for these high forces is the manner in which the finite element model simulates
contact between joints, and the forces developed by the extensor hood during flexion.
Specifically, OpenSim calculates ground reaction forces and joint forces as part of the forward
dynamic simulations conducted, and thus implementing the reaction forces developed in the
finite element model may be duplicating force application, resulting in the observed contact
forces being higher than expected. Further work is needed to identify the specific contributions
to reaction forces in both OpenSim and in FEBio, and account for this in the transfer of force
information between platforms.
4.3 Limitations

Although the dynamic model and finite element model were based on experimentally collected values, several assumptions exist underlying the finite element and dynamic models that influence simulation predictions. In particular, simulation parameters were defined here to be consistent with earlier literature, but not all necessary parameters were described in the earlier publications. Further, assumptions regarding material and joint parameters were made when developing the models. Further discussion of these limitations follows below.

4.3.1 Dynamic Model Limitations

Differences in the preliminary validations performed here and prior reports of simulation performance previously published for the dynamic model (Barry et al., 2018) may be due to differences in simulation parameters, with known differences listed in (Table 6). The primary differences that may exist between the original model and the model used in this study are the joint postures in steady state and the spacing of the contact surfaces away from the fingertip. The forward dynamics simulations with the dynamic model in this study allowed for the MCP, PIP, and DIP joints in the index finger to move within ±5° of the desired posture on their flexion/extension axis; some small joint motion is required to allow muscle force to transmit to the fingertip. However, this freedom of rotation allows the model to take on a different posture than the one used in the cadaveric study. The earlier simulation descriptions did not specify permitted joint rotation. Slight differences in posture may lead to large differences in predicted fingertip contact vectors (Barry, 2016; Qiu, 2014). Additionally, the contact surfaces used for both contact methods in this study were spaced apart from the fingertip by 0.5 mm; the original study did not specify spacing. Spacing is required in OpenSim to achieve a stable simulation, but
differences in fingertip translation may also contribute to observed differences in force magnitude and direction.

Table 6. Dynamic Model Differences From Original

<table>
<thead>
<tr>
<th>Dynamic Model Properties</th>
<th>Original Model Source</th>
<th>Current Model Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>(Holzbaur et al., 2005)</td>
<td>(Holzbaur et al., 2005)</td>
</tr>
<tr>
<td>Model Muscles</td>
<td>(Barry, 2016)</td>
<td>(Barry, 2016)</td>
</tr>
<tr>
<td>Muscle Properties</td>
<td>(Barry, 2016)</td>
<td>(Barry, 2016)</td>
</tr>
<tr>
<td>Rigid Body Mass and Inertia</td>
<td>Not listed</td>
<td>Provided in model file, uncited</td>
</tr>
<tr>
<td>Muscle Moment Arms</td>
<td>(Barry, 2016)</td>
<td>(Barry, 2016)</td>
</tr>
<tr>
<td>Contact Surface Spacing</td>
<td>Not listed</td>
<td>0.5mm</td>
</tr>
<tr>
<td>Joint Rotational Freedom</td>
<td>Not listed</td>
<td>±5°, based on correspondence with author</td>
</tr>
<tr>
<td>Simulation Length</td>
<td>Not listed</td>
<td>Until steady state</td>
</tr>
</tbody>
</table>

For future use, it is important to note that the dynamic model uses muscle and joint parameters derived from both cadaveric studies and in vivo passive joint torques. The use of cadaveric tissues may limit generalization to living subjects since cadaveric tissues exhibit architectural and material changes in comparison to relaxed and contracted living muscle tissues and tendons (Martin et al., 2001).
4.3.2 Finite Element Model Limitations

The finite element model in this study makes use of an isotropic elastic material model, with stiffnesses defined using cadaveric tissue specimens with the stiffnesses measured along the fiber axis of each tissue sample (Qian et al., 2014). However, the isotropic material model implemented in the finite element model assumes uniform stiffness in each direction, which may cause the model to be too stiff overall. This may help to explain the results of the preliminary validation in which using the soft material properties performed better than use of average properties. Additionally, as the isotropic model does not include any anisotropic behavior, any material stiffness that is not in the fiber direction is not accurately simulated.

Initial testing of the integrated model suggests that the available force outputs from FEBio may duplicate forces applied by OpenSim. These outputs are outlined in (Table 7). The outputs chosen for use with the integrated model were the rigid body reaction forces, which are the net forces on each of the rigid bodies. These include force contributions from the tendinous insertions, ligamentous adhesions, as well as any joint contact forces. Because individual contributions to the reaction force are not listed individually, it is difficult to isolate forces such as joint forces that may already be applied within the OpenSim simulation. This may explain undesirably high contact forces. It may be possible to identify force contributions due to the tendinous insertions and adhesions individually; however, the output that currently exposes that information is the “Contact Force” variable. This variable only available in the plot file in FEBio, which is not currently able to be imported directly into MATLAB for use with the integrated model. Additional approaches may allow for this information to be exposed, such as clever use of “dummy” rigid bodies to collect forces at the insertions and adhesions; however, current attempts have resulted in an unstable simulation. Collection of the forces at the insertions
using these methods may also allow for other useful metrics to be used to gauge model performance, such as the ratio of the forces produced at the central and terminal slips.

4.3.3 Integrated Model Limitations

The performance of the integrated model and co-simulation platform could be improved by addressing the above limitations of the dynamic and finite element models. However, there are limitations of the current software implementation as well. For example, the current framework is specific to the dynamic and finite element models used in this study, and has been tested for static simulations. The specificity of the integrated framework to only the models used in this study arises from the need to import the FEBio preprocessor file into MATLAB. In this file, it is necessary to include specific names for the sets of nodes describing the residual tendons as well as names for the forces applied to enable MATLAB to search for them in the preprocessor file. As the current implementation only looks for the tendons described in the finite element model for this study, it is currently not generalizable to other models. Using the GIBBON toolbox to build the model or developing a more robust import method for FEBio preprocessor files into MATLAB may address this issue and allow for this integration method to be used for other models. The integrated model is also currently implemented for static simulations. For dynamic movements, an iterative approach in which the dynamic simulation is integrated forward in time would be required. A similar approach has been previously implemented for other applications, but not has been implemented for this application in which the muscle actuator itself is implicated in the finite element simulation (Dixit et al., 2020, 2019).
Table 7. FEBio Available Force Outputs

<table>
<thead>
<tr>
<th>FEBio Force Output</th>
<th>Force Contributions</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reaction Force</td>
<td>Tendons, adhesions, joint contact</td>
<td>Net force on a given rigid body.</td>
</tr>
<tr>
<td>Joint Force</td>
<td>Rigid body reactions on joints</td>
<td>Available only on 1 DOF joints.</td>
</tr>
<tr>
<td>Joint Torque</td>
<td>Rigid body reactions on joints</td>
<td>Available only on 1 DOF joints.</td>
</tr>
<tr>
<td>Rigid Force</td>
<td>Ground reaction force</td>
<td>Rigid body must use prescribed displacements.</td>
</tr>
<tr>
<td>Contact Force</td>
<td>Tendons</td>
<td>Available only in plot file.</td>
</tr>
</tbody>
</table>
CHAPTER 5: CONCLUSION

5.1 Conclusion

In conclusion, we have demonstrated an integrated platform to combine two biomechanical models for the purposes of simulating contributions of the index finger extensor hood to force production during isometric contraction of the index finger. This integrated framework leverages the strengths of an existing dynamic musculoskeletal model of a hand and index finger, as well as a finite element model of an index finger and extensor hood. However, this integrated framework has also revealed the need for accurate material properties and simulation of the anisotropic nature of biological tissues to reproduce experimental results.

5.2 Future Work

To improve the finite element and dynamic models and thus the resulting integrated model developed in this study, further work is required to address the limitations described previously. This may be done by evaluating the impact that joint motion and contact spacing have on the output fingertip contact forces for the dynamic model to improve validation efforts. For the finite element model, an improvement of the material model is necessary to improve the performance of the model in validation against experimental data. In absence of further experimental testing, an optimization approach may be used in conjunction with an improved material model, such as a transversely isotropic Mooney Rivlin material. This material may provide better results as it is used as the base for the tendon material in FEBio and enables simulation of tendon fibers. An optimization approach may be used to identify and estimate the most influential parameters in defining material properties. Additionally, modifications to the finite element model are necessary to include additional anatomic features, and to identify contributions of individual
anatomic elements to force production, including contributions that the tendinous insertions, ligamentous adhesions, and joint contact have on the total reaction forces. This is important, as it is likely that the reaction forces applied in the integrated framework may be duplicating forces applied natively in OpenSim. This may be explored by implementing “dummy” rigid bodies at the locations of the tendinous insertions, as well as parsing the plot file exported by FEBio for use with MATLAB. Additional ligamentous adhesions may also be necessary to include, to model the joint capsule surrounding the PIP joint as defined in (Ellis et al., 2011) as this may impact forces observed at the central and terminal slips.

As current material properties utilized in the finite element model are taken along the fiber direction of the material, is it likely that the off-fiber axis moduli of the different extensor hood regions may be softer than the moduli currently reported along the fiber direction. One possible method of acquiring material properties in vivo is through the use of ultrasonic indentation, such as using the methods described in (Fougeron et al., 2020). The approach described in this work enabled a hyperelastic constitutive model to be implemented in a finite element model using data collected in vivo, albeit for the soft tissues of the thigh, which are thicker and larger than the extensor hood apparatus of the fingers.

Lastly, improvements may be made to the data exchange in the integrated framework, in which dynamic simulations could be conducted over short time steps and iterated through the finite element simulation. This would enable dynamic simulations to be conducted. This can be performed by repeating the simulations performed here for multiple timesteps, and would require the framework to report and duplicate joint configurations at the beginning of each timestep.
REFERENCES


https://doi.org/10.1016/j.jbiomech.2018.06.017


https://doi.org/10.1016/j.jbiomech.2017.06.026

https://doi.org/10.1109/TBME.2007.901024


https://doi.org/10.1016/j.jbiomech.2019.01.040


Millard, M., Uchida, T., Seth, A., Delp, S.L., 2013. Flexing computational muscle: Modeling and

https://doi.org/10.1115/1.4023390


https://doi.org/10.1016/j.orthres.2003.08.001


https://doi.org/10.1016/j.jbiomech.2014.06.035


https://doi.org/10.1080/10255842.2014.916698


https://doi.org/10.1016/S0021-9290(98)00082-7


Winslow, J.B., 1732. Exposition anatomique de la structure du corps humain. chez Guillaume Desprez... et Jean Desessartz.

XYZ Scientific, n.d. TrueGrid.