AN ABSTRACT OF THE THESIS OF

Sean Hollenbeck for the degree of Master of Science in Civil Engineering and Wood Science presented on September 12, 2018.

Title: Numerical Modeling of Mass Timber Connections.

Abstract approved: ______________________________________________________

Arijit Sinha    Thomas H. Miller

ABSTRACT

Cross-Laminated Timber (CLT) is a prefabricated building material that is relatively new to the United States and North America. It consists of no less than three layers of graded, dimensional lumber glued in alternating longitudinal and transverse layers to create a panel that can be used for various building applications (walls, floors, etc.). Panels are always constructed in an odd number of layers with the two outer layers oriented longitudinally.

Currently, building codes in the United States do not have an abundance of information on CLT construction. Accurately predicting behavior of panels and connections is imperative for static and dynamic analysis of structures that use this new material. Current information about CLT connections is based primarily on physical test data and visual observations of those tests. Creating computer models of the connections can provide more insight into their structural behavior and possibly ways to improve upon the designs. The models can be used to examine various changes in connection geometry and material properties to complement physical testing of some of those variables. The methodology
outlined in this project is general enough to be applied to many different types of nailed CLT bracket connections.

A single, laterally-loaded nail embedded in Douglas-fir is tested and used to confirm a Finite Element Analysis (FEA) model using ABAQUS and Material Point Method model (MPM) using NairnMPM. The single nail model serves as a simple and efficient way to test and troubleshoot methods of modeling a connection. It also serves as an actual subset of the connection. Being a smaller model than a full connection, results can be calculated and extracted faster. The same methods used to develop the single nail model are used to construct a full connection model once confidence in the single nail model is gained. The model is used to analyze the stress contours and locations of material damage of connections used to fasten CLT walls and diaphragms together. Results from the single nail connection model are encouraging, while the results of the full connection model need further refinement. Recommendations for current applications of connection models are discussed along with recommendations for future work and necessary improvements. Commercial FEA software such as ABAQUS do not have the capabilities to model complex wood connections without the use of user-defined features.
Numerical Modeling of Mass Timber Connections

by

Sean Hollenbeck

A THESIS

submitted to

Oregon State University

in partial fulfillment of

the requirements for the

degree of

Master of Science

Presented September 12, 2018

Commencement June 2019
Master of Science thesis of Sean Hollenbeck presented on September 12, 2018

APPROVED:

__________________________________________
Major Professor, representing Wood Science

__________________________________________
Major Professor, representing Civil Engineering

__________________________________________
Head of the Department of Wood Science and Engineering

__________________________________________
Head of the School of Civil and Construction Engineering

__________________________________________
Dean of the Graduate School

I understand that my thesis will become part of the permanent collection of Oregon State University libraries. My signature below authorizes release of my thesis to any reader upon request.

__________________________________________
Sean Hollenbeck, Author
ACKNOWLEDGMENTS

I would like to thank the United States Department of Agriculture and the Tallwood Design Institute for funding this project.

I thank my advisors, Dr. Ari Sinha, Dr. Tom Miller and Dr. John Nairn, for providing valuable guidance and insight. Dr. Erica Fischer allowed me to use her ABAQUS resources to finish this project. Without the expert knowledge and computing power provided by Dr. Fischer this project would never have been completed.

Thanks to my officemates for putting up with my hot, whirring computer, running for hours on end, in our comically small, windowless office.

I would like to thank my friends and colleagues for providing a truly great community in both the School of Civil and Construction Engineering and the Wood Science and Engineering Department.

Finally, I would like to thank my parents for their support throughout my education and into adulthood. Without their help I would not be able to pursue and achieve my goals.
TABLE OF CONTENTS

1 INTRODUCTION .......................................................................................................................................... 1
  1.1 Motivation and Objectives.................................................................................................................. 2

2 LITERATURE REVIEW .................................................................................................................................. 4
  2.1 Material Models of Wood ................................................................................................................... 4
  2.2 FEA Modeling of Wood and Nails ..................................................................................................... 11
    2.2.1 Elastic-Plastic Dowel Connection Models .................................................................................. 11
    2.2.2 Elastic-Plastic Models Combined with Damage Mechanics Models ................................................. 12
    2.2.3 Continuum Damage Mechanics Models .................................................................................... 13
  2.3 Material Point Method ..................................................................................................................... 16
  2.4 Previous Physical Testing .................................................................................................................. 17

3 METHODS AND MATERIALS ..................................................................................................................... 21
  3.1 Physical Testing ................................................................................................................................. 21
  3.2 FEA and MPM Models ....................................................................................................................... 23
    3.2.1 FEA Input for Single Nail, Laterally Loaded ................................................................................ 24
    3.2.2 MPM Input for Single Nail, Laterally Loaded ............................................................................. 33
    3.2.3 ABR105 Connector ..................................................................................................................... 36

4 RESULTS AND DISCUSSION ....................................................................................................................... 41
  4.1 Single Nail Physical Test Results ....................................................................................................... 41
  4.2 MPM Single Nail Model .................................................................................................................... 44
    4.2.1 MPM Results .................................................................................................................................. 44
    4.2.2 MPM Discussion ......................................................................................................................... 45
  4.3 FEA Single Nail Model ....................................................................................................................... 47
    4.3.1 Global Results............................................................................................................................. 47
    4.3.2 Plastic Strain in the Nail ............................................................................................................. 51
    4.3.3 Stress Contours in the Wood ..................................................................................................... 52
    4.3.4 Single Nail FEA Discussion of Results ......................................................................................... 54
  4.4 FEA Model of ABR105 Connection .................................................................................................... 57
    4.4.1 Results ........................................................................................................................................ 57
    4.4.2 Discussion ................................................................................................................................... 64

5 SUMMARY AND CONCLUSIONS ............................................................................................................... 68
  5.1 Single Nail Testing and Models ......................................................................................................... 68
  5.2 ABR105 Connection Model ............................................................................................................... 69
TABLE OF CONTENTS (Continued)

5.3 Recommendations ............................................................................................................................ 70
5.4 Future Work ...................................................................................................................................... 71
REFERENCES ................................................................................................................................................ 73
Appendix I Input Files .................................................................................................................................. 77
  Single Nail Model Input File .................................................................................................................... 77
  MPM Single Nail Model .......................................................................................................................... 90
  ABR105 Input File .................................................................................................................................... 90
Appendix II Videos of Models ..................................................................................................................... 96
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Plastic material properties without post yielding hardening</td>
<td>5</td>
</tr>
<tr>
<td>2</td>
<td>Plastic material properties with post yielding hardening</td>
<td>5</td>
</tr>
<tr>
<td>3</td>
<td>Equivalent stress versus equivalent displacement. (“ABAQUS Doc.” 2014)</td>
<td>7</td>
</tr>
<tr>
<td>4</td>
<td>Graphical representation of the damage variable. (“ABAQUS Doc.” 2014)</td>
<td>9</td>
</tr>
<tr>
<td>5</td>
<td>Equivalent stress vs equivalent displacement curve. (“ABAQUS Doc.” 2014)</td>
<td>10</td>
</tr>
<tr>
<td>6</td>
<td>Compact tension test sample. (Ivanov and Sadowski 2009)</td>
<td>13</td>
</tr>
<tr>
<td>7</td>
<td>Left, ABR105 connector. Right, dimensions in mm of connector</td>
<td>18</td>
</tr>
<tr>
<td>8</td>
<td>Left, CNA4x60 annular ring shank nail. Right, dimensions in mm of nail.</td>
<td>18</td>
</tr>
<tr>
<td>9</td>
<td>Loading configuration for ABR105 in shear (Mahdavifar 2017)</td>
<td>19</td>
</tr>
<tr>
<td>10</td>
<td>Single nail sample dimensions. (a) Isometric view. (b) Front view. (c) Side view</td>
<td>21</td>
</tr>
<tr>
<td>11</td>
<td>Test setup on the UTS</td>
<td>22</td>
</tr>
<tr>
<td>12</td>
<td>Gap between plate and nail.</td>
<td>27</td>
</tr>
<tr>
<td>13</td>
<td>Newly exposed surface from element deletion (“ABAQUS Doc.” 2014)</td>
<td>29</td>
</tr>
<tr>
<td>14</td>
<td>Left: Reference point four being linked to all nodes on the face. Right: Boundary conditions in blue and orange arrows. RP-1 has displacement applied vertically</td>
<td>32</td>
</tr>
<tr>
<td>15</td>
<td>Material point tartan grid defined by the MPM input file.</td>
<td>34</td>
</tr>
<tr>
<td>16</td>
<td>ABR105 Model geometry</td>
<td>36</td>
</tr>
<tr>
<td>17</td>
<td>Surface contact tie regions between wood layers in red and purple</td>
<td>38</td>
</tr>
<tr>
<td>18</td>
<td>ABR105 connection reference points</td>
<td>39</td>
</tr>
<tr>
<td>19</td>
<td>Boundary conditions applied to the ABR105 connection. Vertical displacement is applied at the top and bottom of the left side. The right side is restrained at the top and bottom faces.</td>
<td>40</td>
</tr>
<tr>
<td>20</td>
<td>Physical test Load v. Deflection results</td>
<td>41</td>
</tr>
<tr>
<td>21</td>
<td>Minimum, maximum, and average of the six test samples as a function of displacement.</td>
<td>42</td>
</tr>
<tr>
<td>22</td>
<td>Broken test samples.</td>
<td>43</td>
</tr>
<tr>
<td>23</td>
<td>5% offset yield example of sample 1.</td>
<td>43</td>
</tr>
<tr>
<td>24</td>
<td>MPM analysis results compared to physical test data.</td>
<td>45</td>
</tr>
<tr>
<td>25</td>
<td>MPM von Mises stress cross section</td>
<td>45</td>
</tr>
<tr>
<td>26</td>
<td>Raw FEA data and filtered FEA data.</td>
<td>47</td>
</tr>
<tr>
<td>27</td>
<td>Filtered FEA data compared to minimum, average, and maximum</td>
<td>47</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>Figure 29: 5% offset yield of filtered FEA results.</td>
<td>49</td>
<td></td>
</tr>
<tr>
<td>Figure 30: von Mises stress of single nail FEA model. Displacement is at increments of 0.5mm for (a) through (g). (h) skips to the final time-step time-step. (c) is at 5% offset yield. View is an isometric cross-section.</td>
<td>50</td>
<td></td>
</tr>
<tr>
<td>Figure 31: Location of data collection and equivalent plastic strain plotted on the undeformed shape.</td>
<td>51</td>
<td></td>
</tr>
<tr>
<td>Figure 32: Plot of equivalent plastic strain vs length of the nail.</td>
<td>52</td>
<td></td>
</tr>
<tr>
<td>Figure 33: Stress parallel-to-grain.</td>
<td>53</td>
<td></td>
</tr>
<tr>
<td>Figure 34: Stress perpendicular to grain.</td>
<td>53</td>
<td></td>
</tr>
<tr>
<td>Figure 35: Shear stress in the longitudinal, tangential plane (parallel to front face).</td>
<td>54</td>
<td></td>
</tr>
<tr>
<td>Figure 36: Plastic hinge location.</td>
<td>55</td>
<td></td>
</tr>
<tr>
<td>Figure 37: FEA vs test data for ABR105 connection.</td>
<td>57</td>
<td></td>
</tr>
<tr>
<td>Figure 38: Nail numbering scheme.</td>
<td>59</td>
<td></td>
</tr>
<tr>
<td>Figure 39: von Mises stress in the connection at equally spaced time increments.</td>
<td>60</td>
<td></td>
</tr>
<tr>
<td>Figure 40: von Mises stress in only the nails over equally spaced time increments.</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td>Figure 41: Tension parallel-to-grain above nail holes.</td>
<td>62</td>
<td></td>
</tr>
<tr>
<td>Figure 42: Tension damage parallel-to-grain.</td>
<td>62</td>
<td></td>
</tr>
<tr>
<td>Figure 43: Compression parallel-to-grain stress.</td>
<td>63</td>
<td></td>
</tr>
<tr>
<td>Figure 44: Stress perpendicular-to-grain.</td>
<td>63</td>
<td></td>
</tr>
<tr>
<td>Figure 45: Damage perpendicular-to-grain.</td>
<td>64</td>
<td></td>
</tr>
<tr>
<td>Figure 46: von Mises stress on ABR105.</td>
<td>65</td>
<td></td>
</tr>
<tr>
<td>Figure 47: Location of plastic strain in the nails at similar levels of deformation. Top: Plastic strain in Nail 2 of left side. Bottom: Single Nail model plastic strain.</td>
<td>67</td>
<td></td>
</tr>
</tbody>
</table>
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 1: Test results of ABR105 loaded in shear for Douglas-fir panels (Mahdavifar 2017)</td>
<td>20</td>
</tr>
<tr>
<td>Table 2: Nail and plate steel material properties</td>
<td>25</td>
</tr>
<tr>
<td>Table 3: Linear properties of Douglas-fir. Direction 11 is longitudinal, 22 is tangential, 33 is radial.</td>
<td>26</td>
</tr>
<tr>
<td>Table 4: Failure criterial for Douglas-fir</td>
<td>26</td>
</tr>
<tr>
<td>Table 5: Fracture energies assumed for Douglas-fir</td>
<td>27</td>
</tr>
<tr>
<td>Table 6: Non-linear material properties for MPM model</td>
<td>33</td>
</tr>
<tr>
<td>Table 7: Physical test data results</td>
<td>44</td>
</tr>
<tr>
<td>Table 8: Percent difference between FEA data and average physical test data at 0.5mm increments</td>
<td>48</td>
</tr>
<tr>
<td>Table 9: Percent contribution of each nail on the left side</td>
<td>59</td>
</tr>
<tr>
<td>Table 10: Percent contribution each nail on the right side</td>
<td>59</td>
</tr>
</tbody>
</table>
1 INTRODUCTION
Cross-Laminated Timber (CLT) is a prefabricated building material that is relatively new to the United States and North America. It consists of no less than three layers of graded, dimensional lumber glued in alternating longitudinal and transverse layers to create a panel that can be used for various building applications (walls, floors, etc.). Panels are always constructed in an odd number of layers with the two outer layers oriented longitudinally.

One use for CLT panels is as shear walls to resist lateral loads due to wind or earthquakes. When CLT shear walls are subjected to lateral loading, tension (uplift) and shear demands are imposed on the fasteners at the base of the panel. Previous research has investigated the performance of fasteners under each of these loading demands (Karacabeyli and Douglas 2013; Mahdavifar 2017; Reynolds et al. 2017). However, previous research has resulted in load-deflection data and observations of failure modes for various fasteners that does not provide detailed information about the behavior of individual parts of the connection and how they are contributing to the overall behavior of the connection.

Computer modeling provides further qualitative and quantitative insights into the behavior of connections. Experimental tests are often time and financially expensive. In addition, limitations of instrumentation do not allow researchers to fully evaluate and investigate all connection components. Therefore, numerical modeling approaches are used to further develop parametric studies and quantify the stresses in each of the connection components throughout the loading. The development of a detailed computer model can provide a complete visualization and quantification of load distribution in a connection. Stress, strain, material damage, displacements, and a multitude of other variables can be extracted, analyzed and used to better understand the overall response of a connection.

Modeling timber connections with dowel type fasteners is a major challenge. Wood is an anisotropic and nonhomogeneous material, and requires detailed material property definitions to quantify and
properly represent its behavior in a model. Each orthogonal direction with respect to the grain
direction, longitudinal, radial, and tangential, has different strength, stiffness and fracture energy
properties. The three orthogonal shear planes also require material property definitions. The dowel
steel material properties are not always known, and the contact interface between the dowel and wood
is also difficult to specify (Sandhaas 2012).

The commercially available finite element method (FEM) software, ABAQUS (2014) was used to perform
the numerical analyses described in this research. ABAQUS (2014) has anisotropic material models in the
elastic and plastic ranges as well as contact surface algorithms built-in. Wood connections are difficult to
model because of the several failure modes of the wood. The material properties of wood vary enough
in each orthogonal direction that FEA can develop numerical artifacts at large deformations. Wood is
ductile in compression while brittle in shear and tension. These failure modes can occur simultaneously,
presenting a numerical modeling challenge (Sandhaas 2012). A number of methods have been
developed for modeling dowel connections in wood and special care must be taken when considering
the strengths and weaknesses of each (Hong and Barrett 2010; Khelifa et al. 2016; Portioli et al. 2010;
Sandhaas 2012).

1.1 Motivation and Objectives
To expand the body of knowledge on the performance of timber connections under various loading
scenarios, numerical modeling techniques need to be benchmarked against experimental tests. There
are limited studies on the performance of timber connections with nails. The majority of the previous
research investigated timber connections using bolts. To better understand how base connections for
shear walls should be detailed and designed, engineers first must understand the load distribution
through the connection and the contribution of all of the components. Therefore, a number of single-
nail timber connection FEA models were developed using ABAQUS and the modeling techniques were
benchmarked against previously performed experimental tests.
The objectives of this study are the following:

1. Develop an FEA model of a single dowel connection with a Douglas-fir main member and a steel plate as the side member. Compare its behavior with physical testing data and also a material point method (MPM) model.

2. Benchmark the modeling methodology against previously performed experimental tests.

3. Use the modeling methodology developed in the previous two objectives to model a CLT nailed connection.

4. Analyze the results to determine where the connection is exhibiting possible material failures and also where modeling features could be improved.

5. Assess the advantages and shortcomings of the modeling method.
2 LITERATURE REVIEW
2.1 Material Models of Wood
There are many previously developed material models that can be used for anisotropic materials. Wood has different material properties in each orthogonal direction and each shear plane (Forest Products Laboratory 2010), leading to different yield and failure criteria. Some of the methods are derived for either anisotropic metal behavior or carbon fiber but can be repurposed to model wood.

Nonlinear plasticity methods can be derived from von Mises stresses to calculate yielding based on a known yield strength and the applied stress gathered from the stress tensor. The yield stress can be plotted similar to Mohr’s Circle and is known as a “yield surface” (Boresi and Schmidt 2003; von Mises 1913). The yield surface is defined by Eq. (1)

\[
\sigma = \sqrt{(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2)}
\]  

(1)

Hill and Hoffman criteria are based on von Mises yield stress with one main difference (Hill 1948; Schellekens and Borst 1990). The von Mises equivalent yield stress is based on a single value. The Hill yield criterion uses six different yield stresses to define the yield surface, one in each principal orthogonal direction and in all three shear planes (Hill 1948). Hoffman uses nine yield stresses, two in each principal orthogonal direction to account for different tension and compression yield stresses and all three shear planes (Hill 1948; Schellekens and Borst 1990). For all three plasticity methods the material can be assumed to be elastic-perfectly plastic or a bilinear model, where the material still has stiffness (hardening) after it has yielded. If the material hardens, the yield surface will expand anisotropically or isotropically. For materials that harden isotropically, when yielding and hardening occurs in one principal direction or shear stress, all of the other yield stresses increase equally (ABAQUS 2014). Anisotropic hardening will expand the yield surface in the direction(s) of yielding and hardening.
only. Hardening will generally provide a small amount of additional post-yield stiffness to the material. See Figures 1 and 2 for representations of elastic-perfectly plastic versus bilinear behavior.

![Figure 1: Plastic material properties without post-yielding hardening](image)

![Figure 2: Plastic material properties with post-yielding hardening.](image)

In addition to nonlinear plasticity material properties, FEA models of timber connections require continuum damage mechanics material models. These models introduce softening of the material after reaching a known stress state, and therefore quantify and predict the failure of the connection. A common approach used in modeling of anisotropic materials is the Hashin failure criterion (Hashin 1980; Hashin and Rotem 1973). It was developed for fiber-reinforced composites like carbon fiber and formulates stress limits by the Invariants of the stress tensor. Four modes of failure are considered for 2D shell and plate elements:
1. fiber rupture in tension;
2. fiber buckling and kinking in compression;
3. matrix cracking under transverse tension and shearing; and
4. matrix crushing under transverse compression and shearing.

The stress, $\sigma$, response of the material is computed from

$$\sigma = C_d \varepsilon$$  \hspace{1cm} (2)

where $\varepsilon$ is the strain and $C_d$ is the elasticity matrix which reflects any damage sustained. It has the form

$$C_d = \frac{1}{D} \begin{bmatrix} (1 - d_f)E_1 & (1 - d_f)(1 - d_m)\nu_{21}E_1 & 0 \\ (1 - d_f)(1 - d_m)\nu_{12}E_2 & (1 - d_m)E_2 & 0 \\ 0 & 0 & (1 - d_s)GD \end{bmatrix}$$  \hspace{1cm} (3)

where $D$ is current state of the combined fiber and matrix damage and is given by

$$D = 1 - (1 - d_f)(1 - d_m)\nu_{12}\nu_{21}$$  \hspace{1cm} (4)

The variables $d_f$ and $d_m$ are the damage to the fibers and matrix, respectively, $d_s$ is the shear damage, $E_1$ is the Young’s modulus in the fiber direction while $E_2$ is the modulus perpendicular to the fiber direction, $G$ is the shear modulus, and $\nu_{21}$ and $\nu_{12}$ are Poisson’s ratios. The damage variables $d_f$, $d_m$, and $d_s$, are derived from the four previously discussed failure modes.

$$d_f = \begin{cases} d^f_f & \text{if } \hat{\sigma}_{11} \geq 0 \\ d^c_f & \text{if } \hat{\sigma}_{11} < 0 \end{cases}$$  \hspace{1cm} (5)

$$d_m = \begin{cases} d^f_m & \text{if } \hat{\sigma}_{22} \geq 0 \\ d^c_m & \text{if } \hat{\sigma}_{22} < 0 \end{cases}$$  \hspace{1cm} (6)

$$d_s = 1 - (1 - d_f^c)(1 - d_f^c)(1 - d_m^c)(1 - d_m^c)$$  \hspace{1cm} (7)
$\sigma_{11}$ and $\sigma_{22}$ are components of the effective stress tensor. The effective stress tensor is primarily used to evaluate the damage initiation criteria.

$$\hat{\sigma} = M\sigma$$  \hspace{1cm} (8)

$$M = \begin{bmatrix}
\frac{1}{(1-d_f)} & 0 & 0 \\
0 & \frac{1}{(1-d_m)} & 0 \\
0 & 0 & \frac{1}{(1-d_s)}
\end{bmatrix}$$  \hspace{1cm} (9)

Where $d_f$, $d_m$, $d_s$ are defined in Eq. (5) through Eq. (7). Prior to any damage initiation and evolution, the operator $M$ is equal to the identity matrix. The effective stress represents the stress acting on the damaged area that can resist internal forces.

To alleviate mesh dependency during material softening, ABAQUS introduces a characteristic length in the constitutive law. Different sized elements effect the energy that the element can absorb. Accounting for this is necessary in continuum damage mechanics approaches. This is expressed as a stress-displacement relationship.

![Figure 3: Equivalent stress versus equivalent displacement. (ABAQUS 2014)](image-url)
Equivalent displacement and stress for each of the four damage models are defined as follows:

Fiber tension ($\hat{\sigma}_{11} \geq 0$):

$$\delta_{eq}^{ft} = L^c \sqrt{\langle \varepsilon_{11} \rangle^2 + \alpha \varepsilon_{12}^2}$$  \hspace{1cm} (10)$$

$$\sigma_{eq}^{ft} = \frac{(\sigma_{11})(\varepsilon_{11}) + \alpha \Gamma_{12} \varepsilon_{11}}{\delta_{eq}^{ft} / L^c}$$  \hspace{1cm} (11)$$

Fiber compression ($\hat{\sigma}_{11} < 0$):

$$\delta_{eq}^{fc} = L^c (-\varepsilon_{11})$$  \hspace{1cm} (12)$$

$$\sigma_{eq}^{fc} = \frac{(-\sigma_{11})(-\varepsilon_{11})}{\delta_{eq}^{fc} / L^c}$$  \hspace{1cm} (13)$$

Matrix tension ($\hat{\sigma}_{22} \geq 0$):

$$\delta_{eq}^{mt} = L^c \sqrt{\langle \varepsilon_{22} \rangle^2 + \varepsilon_{12}^2}$$  \hspace{1cm} (14)$$

$$\sigma_{eq}^{mt} = \frac{(\sigma_{22})(\varepsilon_{22}) + \Gamma_{12} \varepsilon_{11}}{\delta_{eq}^{mt} / L^c}$$  \hspace{1cm} (15)$$

Matrix compression ($\hat{\sigma}_{22} < 0$):

$$\delta_{eq}^{mc} = L^c \sqrt{\langle \varepsilon_{22} \rangle^2 + \varepsilon_{12}^2}$$  \hspace{1cm} (16)$$

$$\sigma_{eq}^{mc} = \frac{(-\sigma_{22})(-\varepsilon_{11}) + \Gamma_{12} \varepsilon_{11}}{\delta_{eq}^{mc} / L^c}$$  \hspace{1cm} (17)$$

The characteristic length, $L^c$, is based on the geometry of the element and is the length of a line across a first-order element.

After damage initiation occurs, ($\delta_{eq} \geq \delta_{eq}^0$), the damage variable is given by the following
The point at which the material has completely failed in the given mode is at $\delta_{eq}^f$, and the equivalent displacement at which damage initiation occurs is $\delta_{eq}^0$. Graphically this is shown in Figure 4.

![Figure 4: Graphical representation of the damage variable. (ABAQUS 2014)](image)

The damage initiation values for $\delta_{eq}^0$ are based on the damage initiation stress and the elastic properties of the material in any of the given directions. They have the following forms:

Fiber tension ($\hat{\sigma}_{11} \geq 0$):

$$ F_f^T = \left( \frac{\sigma_{11}}{X^T} \right)^2 + \alpha \left( \frac{\tau_{12}}{S^T} \right)^2 $$

(19)

Fiber compression ($\hat{\sigma}_{11} < 0$):

$$ F_f^C = \left( \frac{\sigma_{11}}{X^C} \right)^2 $$

(20)
Matrix tension ($\sigma_{22} \geq 0$):

$$F^t_m = \left( \frac{\sigma_{22}}{Y_T} \right)^2 + \left( \frac{\tau_{12}}{S_L} \right)^2$$ \hfill (21)

Matrix compression ($\sigma_{22} < 0$):

$$F^c_f = \left( \frac{\sigma_{22}}{2S_T} \right)^2 + \left[ \frac{Y_C}{2S_T} \right]^2 - 1 \left( \frac{\sigma_{22}}{Y_C} + \frac{\tau_{12}}{S_L} \right)^2$$ \hfill (22)

In the above equations, $X^T$ denotes the longitudinal tensile strength; $X^C$ denotes the longitudinal compressive strength; $Y^T$ denotes the transverse tensile strength; $Y^C$ denotes the transverse compressive strength; $S^L$ denotes the longitudinal shear strength; $S^T$ denotes the transverse shear strength; $\alpha$ is a coefficient that determines the contribution of the shear stress to the fiber tensile failure initiation criterion; and $\delta_{11}, \delta_{22}, \delta_{11}$ are components of the effective stress tensor, $\sigma$, that is used to evaluate the failure initiation criteria.

For each failure mode there is a specified failure energy, $G^C$ which correspond to the area of triangle OAC in Figure 5.
The values of $\delta_{eq}$ depend on the values of $G^c$ that are specified in each direction.

When an element is loaded between O and A in a specific mode of failure (fiber tension, fiber compression, matrix tension, matrix compression, or shear) it is still within the elastic region of the material. After damage initiation has occurred at point A, the material decreases its ability to resist stress in that particular mode. The material stiffness becomes negative and the maximum stress decreases until it satisfies the failure criteria previously described. After failure the element is removed from the mesh.

2.2 FEA Modeling of Wood and Nails

Previous research efforts on the performance of timber connections have employed both numerical and analytical modeling approaches. (Mahdavifar 2017) used the open source structural analysis package SAWS (Seismic Analysis of Wood Structures) to simulate the performance of the CLT connectors. This approach reduces the entire connection to a spring element. This element is attached to a two-dimensional element that represents the entire CLT panel. This is useful for a general model of an entire building. It can be used to estimate forces in CLT panels and connectors at a larger scale. The National Design Specification for Wood Structures (NDS) (AWC 2015) adopted yield mode equations that are presented in the Wood Handbook (Forest Products Laboratory 2010). These are analytical models that predict the yielding of dowel connectors based on material properties of the wood and dowel and the geometry of the connection. These equations are used by engineers to design connections within large structural systems. This research uses three-dimensional numerical modeling to evaluate the performance of dowel connections.

2.2.1 Elastic-Plastic Dowel Connection Models

FEA programs such as ABAQUS and ANSYS have been used to simulate the behavior of timber connections in previous studies. A study by Mirianon et al. (2008) analyzed dowel fasteners in mass timber to locate regions of high stress and strain. From the analysis, design recommendations to avoid
undesirable failure modes were developed. Although the modeling methodologies developed were useful, they were not benchmarked against experimental tests. The authors developed a user-defined subroutine in ABAQUS to represent the material definition constitutive model.

A different application of plasticity material modeling was explored using a “foundation zone” to model the bearing properties around the nailed connections (Hong and Barrett 2010). It was theorized that this zone surrounding the nail has less stiffness and less strength than the surrounding wood. To model the material properties of the wood, the Hill yield criterion was used. The wood primarily failed in compression due to bearing of the dowel on the wood, so it is argued that using the Hill criterion is a valid assumption (Hong and Barrett 2010). Tension yielding would most likely not control, making the Hoffman criterion unnecessary. The authors (Hong and Barrett 2010) benchmarked their FEA methodology against experimental data. The FEA showed good correlation with the experimental data, however, modifications of the material properties were made throughout the analysis to fit the data.

2.2.2 Elastic-Plastic Models Combined with Damage Mechanics Models
Two different models of wood connections that include continuum damage mechanics are presented by Portioli et al. (2010) and Khelifa et al. (2016). Portioli et al. (2010) used the Hashin failure criterion to model wooden peg connections. A model of a double shear connection using a wooden peg as the dowel was developed using ABAQUS. Material properties of Ash for the peg and an unnamed fir for the side members were used. The overall stiffness of their model did not match the physical testing but the overall strength matched well. Khelifa et al. (2016) used the Hill yield criterion along with a user-defined subroutine for failure criteria to model damage around double shear bolted connections. Once the wood reached a specified plastic yield strain, the material failed and could no resist load. The wood elements that bear directly on the bolts met the failure criteria. The force-displacement behavior aligned well with the physical test data (Khelifa et al. 2016). This study highlighted that continuum
damage mechanics that include damage or softening of the wood material better aligns with experimental test data than an elastic-plastic model.

Instead of using Hashin damage mechanics, Franke and Quenneville (2011) used cohesive elements to simulate cracking in timber bolted connections. A two-dimensional model of a simply-supported timber beam that was loaded by a four-bolt connection at mid-span was developed. The model used cohesive elements to allow for crack planes parallel to the longitudinal wood fibers. The cohesive elements failed at a specified stress state derived from given wood material properties, allowing for complex crack propagation to be investigated for connections that load wood perpendicular-to-grain in tension (Franke and Quenneville 2011). This modeling methodology is critical for bolted connections with perpendicular-to-grain loading; however, could not be employed for dowel connections that are loaded parallel to grain, or in compression.

2.2.3 Continuum Damage Mechanics Models
A study performed by Ivanov and Sadowski (2009) is a direct application of a Hashin damage model. A series of different plywood samples were physically tested. Ivanov and Sadowski (2009) then used the data from the physical tests to benchmark numerical modeling techniques in ABAQUS. The two types of samples were dog-bone shaped samples for tensile strength and compact tension samples as seen in Figure 6. The compact tension sample is pulled apart by loading the two holes.

Figure 6: Compact tension test sample. (Ivanov and Sadowski 2009)
From the tests and analyses, it was concluded that the models were useful qualitatively. The modeling results of the compact tension model presented by Ivanov and Sadowski (2009) did not show good correlation with the physical testing results. The dog-bone samples were more promising but still need amendments to the material model. It is possible that the fracture energies and the failure criteria need refining, or that too much viscous dampening is applied to the model to help stabilize the results (Ivanov and Sadowski 2009).

The US Department of Transportation Federal Highway Administration released a publication detailing the modeling of wooden guardrails (Murrey 2007). The FEA package LS-DYNA was used to develop these models. A modified Hashin continuum damage mechanics approach was used to define the material within LS-DYNA (Murrey 2007). Hashin damage mechanics defines linear softening after the failure criterion is reached, as shown in Figure 5. This study defines nonlinear softening after the failure criterion has been reached and as the element approaches complete failure and deletion. Compressive stress is not considered to cause failure of the element, instead it is considered to harden after yielding, similar to a bilinear elastic-plastic material. The guardrails were constructed of both Southern Yellow Pine and Douglas-fir. Fracture energies for Douglas-fir were set equal to those of Southern Yellow Pine because fracture energy data were not available for Douglas-fir. The material model definitions and fracture energies were two important items taken from this study (Murrey 2007).

A comprehensive study on modeling timber connections is presented by Sandhaas (2012). It considers previous methods used to model orthotropic materials and outlines the positive benefits and negatives of each method for modeling wood. The final decision to use continuum damage mechanics was based on its relative ease of implementation with most FEA software packages, the ability to combine brittle and ductile failure modes and visualize the failure modes of the wood. A 3D continuum damage mechanics material sub-routine was written for ABAQUS. The material model was verified using a 1mm X 1mm X 1mm brick element that was loaded in compression, tension and shear in all orthogonal
directions to ensure that the input parameters match the output from the subroutine. After the material model was verified to be working correctly, a dog-bone tension test was modeled. Effects of varying material properties, boundary conditions, mesh size, and large versus small displacement theory were investigated. The same was done for a bolt embedment model. The initial conclusions were that the model was able to simulate the initial nonlinear, ductile behavior. However, numerical problems remained. The first row of elements that softened would distort excessively and suddenly lose load carrying capacity. The stress redistribution after the first band of element softening did not take place. Full connection models with multiple bolts embedded in wood were developed along with slotted steel plate connection models. These models had many of the same assumptions as the dog-bone tension test simulations whereas the elements distorted without stress redistribution. A suggestion for further research in this area was implementing element deletion to eliminate the problems with stress redistribution (Sandhaas 2012).

Two studies resulted from of the methods developed by Sandhaas (2012). Gharib et al. (2017) made improvements to the material model by alleviating mesh size dependency. When performing the same analyses done by Sandhaas (2012), specifically the tension and dowel embedment models, premature simulation errors did not occur and therefore, the complete behavior of the connection was able to be simulated under the imposed loading conditions (Gharib et al. 2017). Then, Hassanieh et al. (2018) used the material model developed by Gharib et al. (2017) to simulate bolt behavior embedded in grout and in CLT. The model was adequate at estimating the peak failure load of the connection but consistently overestimated the initial stiffness. The discrepancy is attributed to pre-existing damage in the wood. Cutting and drilling cause the layer of wood immediately surrounding the connection to be weaker than the bulk material (Hassanieh et al. 2018).

From the results of the different FEA models it is apparent that the continuum damage mechanics material models have produced results that correlate with physical test data when modeling wood. The
material models developed by Sandhass (2012) and improved upon by Gharib et al. (2017), are derived from the physical material properties of the wood. Hashin damage mechanics was chosen because it most closely mimics the attributes of the developed material model and it is commercially available in ABAQUS.

2.3 Material Point Method
The Material Point Method (MPM) is an independent analysis technique can be used to verify that the FEA in ABAQUS is providing valid results. The software package used for MPM is written by John Nairn (Nairn 2015b). Because the code can be modified and viewed, it is easier to interpret all of the calculation steps compared to ABAQUS, where the code is not readily open to the user. MPM solves the same governing equations that FEA solves. The difference is how the two methods discretize the problem. The two methods both solve Eq. (23) which is the expression for virtual work. The sum of virtual work must equal the sum of the internal energy (ABAQUS 2014; Sulsky et al. 1994).

\[
\int_{\delta V} T \cdot \delta u \, dS + \int_V b \cdot \delta u \, dV = \int_V \rho a \cdot \delta u \, dV + \int_V \sigma \cdot \nabla \delta u \, dV
\]  

The left side of Eq. (23) is equal to the work done on the object from virtual displacements, and the right side of the equation is equal to the internal stresses (ABAQUS 2014; Sulsky et al. 1994).

MPM provides another method to solve problems traditionally solved with FEA. The formulation of MPM inherently has different characteristics then FEA (Sulsky et al. 1994, 1995; Sulsky and Schreyer 1996). FEA uses a mesh of elements to create the domain of the problem while MPM uses particles on a background grid (Sulsky et al. 1994). Shape functions in FEA are defined between nodes of an element. In MPM, shape functions are defined between particles and the background grid. The material points are used to discretize the solid body much like pixels in a computer image (Sulsky et al. 1994). Information like the mass, position, velocity, acceleration, stress state, etc. are stored with the material points (Sulsky et al. 1994). The information from the material points is extrapolated to the background grid to perform calculations (Sulsky et al. 1994). The background grid is fixed in space while the material
points are free to move. Once the solution for the time-step is solved on the background grid, the material point properties are updated (Sulsky et al. 1994).

MPM has some advantages over FEA for modeling wood. MPM can handle large deformations and large compressive strains better than FEA because of its meshless nature (Sulsky et al. 1994). There is a major disadvantage in using FEA for large deformation problems. The mesh can become highly distorted and the analysis numerically unstable. Modeling timber connections can involve the crushing of wood which introduces large displacements into the model. MPM in theory can handle the large displacements involved with crushing better than FEA. MPM has not been used to study timber connections but it has been used to investigate cracking and transverse compression, two responses observed in this study and others (Nairn 2006, 2007a).

2.4 Previous Physical Testing
There are numerous studies that investigated the performance of CLT and CLT connections under lateral load, specifically seismic loading. Some involve testing of entire panels (Ceccotti et al. 2018; Reynolds et al. 2017). Others focus on the performance of individual connections (Gavric et al. 2012; Rinaldin et al. 2013). The overall behavior of CLT structures subject to seismic loading has also been investigated in panel tests, connection tests, and full-scale building tests (Popovski and Karacabeyli 2012). An overview of previous research on CLT for resisting lateral loads is provided in Chapter 4 of the CLT Handbook (Lindt et al. 2013).

Past studies have looked into the strength and stiffness of CLT connections (Mahdavifar 2017). Specifically, the ABR105 angle bracket with CNA4X60 nails from Simpson Strong-Tie was investigated, and shown in Figure 7 and Figure 8.
Mahdavifar (2017) investigated the lateral capacity of the CNA4X60 nails fastened with 10 gage (~3.3mm) galvanized Grade 33 steel as a side member and Douglas-fir CLT as the main member. A plastic hinge of 14.6mm is calculated using the NDS (AWC 2015). The test results for yield strength were all 10% to 25% higher than the predicted values from the NDS design equations (AWC 2015; Mahdavifar 2017). It was also determined that the yielding of the nail occurs in the face layer of the CLT panels.

Figure 7: Left, ABR105 connector (Simpson Strong-Tie (2017). Right, dimensions in mm of connector.

Figure 8: Left, CNA4x60 annular ring shank nail (Simpson Strong-Tie (2017). Right, dimensions in mm of nail.
Therefore, it is conservative to use the NDS and the nail yield modes for calculating the strength of the connections using CNA4X60 nails and CLT panels (Mahdavifar 2017).

The study also included cyclic testing of the ABR105 connector, and determined the initial stiffness of the connector, yield strength, peak strength and force versus displacement backbone curves from the cyclic test data. The test setup for the ABR105 in shear can be seen in Figure 9. For Douglas-fir panels the initial stiffness of the connector, yield strength, and peak strength can be seen in Table 1.

The stiffness and strength of a connection loaded cyclically is often very comparable (perhaps a small percentage less) to those from a monotonically loaded connection. A total of six Douglas-fir ABR105 connections were loaded to failure using the CUREE protocol. The backbone curve is recovered from the cyclic test data by recording the maximum force value at each displacement in the elastic and yielding zones of the data. The data for this study only focuses on the elastic and yielding ranges in the positive half of the cyclic data (i.e. the force and displacement are positive).

Figure 9: Loading configuration for ABR105 in shear (Mahdavifar 2017)
Table 1: Test results of ABR105 loaded in shear for Douglas-fir panels (Mahdavifar 2017)

<table>
<thead>
<tr>
<th>Species</th>
<th>Initial Stiffness (N/mm)</th>
<th>Yield Strength (N)</th>
<th>Peak Strength (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Douglas-fir</td>
<td>Mean</td>
<td>2880</td>
<td>9840</td>
</tr>
<tr>
<td></td>
<td>SD</td>
<td>517</td>
<td>1550</td>
</tr>
<tr>
<td></td>
<td>COV</td>
<td>17.9%</td>
<td>15.5%</td>
</tr>
</tbody>
</table>

Qualitatively, the following observations were made from the test results. The failure was concentrated on the floor side, which has ten CNA4X60 nails and connects the bracket to a floor/ diaphragm, compared to the fourteen nails on the wall side, which connects the bracket to a shear wall. The failure was concentrated in the wood fibers which was followed by plastic hinges formed in the nails at the face layers. There was slight damage at the holes in the connector. They were bent in the direction of loading (Mahdavifar 2017). The results of the ABR105 CLT connection tests are used to compare directly with the model results in the Results and Discussion.
3 METHODS AND MATERIALS
3.1 Physical Testing
Individual nail tests for lateral resistance are needed to compare with the FEA results, and load versus deflection data are necessary. Because the Simpson Strong-Tie CNA4x60 nail is the recommended nail to be used with the ABR105 Angle Bracket, the lateral load versus deflection behavior of this particular nail needed to be determined. The nails are 60mm in length and 40mm in diameter. They have a bending yield strength of 1090 MPa (Mahdavifar 2017). Annular rings interlock with the wood fibers to provide greater withdrawal capacity. Solid sawn nominal dimension 2x4 (38mm x 230mm) Coastal Douglas-fir is used as the main member in the test and a 10-gage (3.95mm) galvanized steel plate is used as the side member. Average dimensions of the main member are 38mm X 89mm x 230mm, with the nail driven into the edge 76mm from the top of the 228mm face. The hole in the side member is drilled at approximately 13mm edge to edge from the bottom of the plate at a radius of 2.4mm. The nails are driven in by hand without predrilling. A total of six samples was prepared. Sketches and the dimensions

![Figure 10: Single nail sample dimensions in mm. (a) Isometric view. (b) Front view. (c) Side view.](image-url)
of the sample can be seen in Figure 10. The longitudinal grain runs vertically in the figure. The tangential face is the front face of the specimen while the radial face is along the edge. All samples were selected based on straightness of grain, and minimal presence of knots, checking or other defects.

A test apparatus is used with the Universal Testing Machine (UTM) to ensure that all of the samples are loaded in the same way. A picture of the test setup can be seen in Figure 11. Samples are clamped on the face and edges. The top of the apparatus has a small tab that prevents the sample from slipping upwards once load is applied. The apparatus is clamped to the UTM base plate. This localizes the deformations at the nail and the surrounding wood. The blocks underneath the sample are for ease of replacing the specimen. The UTM is used to record deflection and force at the load cell at the head of the machine. This may introduce a small amount of deflection error due to some displacement of the
UTM. The load cell and its connections may displace a small amount, and depending on the force generated in the test, the entire UTM may displace a small amount as system compliance. This is most evident in small travel and high force tests (Instron 2006). Test data generated from the single nail tests are relatively low force and high displacement with regards to the limitations of the UTM, and the error generated by compliance should be negligible. The test ended when the nail could no longer resist load. This is usually observed as a sharp decline in the load deflection curve or the nail head shearing off.

### 3.2 FEA and MPM Models

FEA is often used by researchers to perform structural analysis and investigation. There are many software packages that can perform FEA for structural systems. ABAQUS was chosen because of the flexibility in analysis features it can perform as discussed in Section 2. ABAQUS allows for modeling both in 2D and 3D and has the ability to define complex contact surfaces and behavior, that are critical for simulating the behavior of a connection. ABAQUS has many material properties built into the program. Another method for analysis is the lesser known Material Point Method (MPM). Instead of meshing the structure, it is constructed with material points that are extrapolated to a background grid. The same principles of virtual work are employed but in a different configuration. NairnMPM written by John Nairn was used to implement MPM analysis (Nairn 2015a). This process was used to benchmark the FEA analysis performed in ABAQUS.

Before modeling an entire connection, an FEA model of a single nail connection was developed to investigate the modeling techniques associated with the material definition within ABAQUS. It was crucial to ensure that the material properties of the wood and nails included failure definitions that demonstrated the splitting of the wood and ductility of the nail at high strain. Different material property modeling methodologies were used based on previous research. Bilinear elastic-plastic material properties for the wood similar to Hong and Barret (2010) were implemented in the early stage models. After many iterations with slight changes in wood material properties, steel material properties...
and geometry, this method failed to replicate the results found in Hong and Barret (2010). A continuum
damage mechanics material model for wood was implemented in later stage models, replacing the
elastic-plastic material model. The load-displacement results from these simulations were compared to
physical test data of the CNA4x60 proprietary nail from Simpson Strong-Tie. After the modeling
methodology was benchmarked, a more complex multi-nail CLT connection model was developed.

There are three models that will be discussed:

1. FEA model with single CNA4x60 nail embedded in a Douglas-fir main member
2. MPM model with single CNA4x60 nail embedded in a Douglas-fir main member
3. FEA model of Simpson Strong-Tie ABR105 bracket connection in Douglas-fir CLT

The first two models are of the single CNA4x60 nail embedded in a Douglas-fir main member, with a
steel side member loaded laterally. It is a model of the previously described physical test in Section 3.1.
This connection behavior is simulated using FEA and MPM in ABAQUS and NairnMPM, respectively. The
third model is of the Simpson Strong-Tie ABR105 bracket connection in CLT. There is one load case for
this model, and it is loaded in shear only.

3.2.1 FEA Input for Single Nail, Laterally Loaded
3.2.1.1 Material Models
The material model for the Douglas-fir is a Hashin continuum damage mechanics model as discussed in
Section 2.2.3 of the literature review (Hashin 1980; Hashin and Rotem 1973). The material has elastic
properties until a defined stress state is reached. After this, the material begins to soften at a rate
defined by the fracture energy of the material until it can no longer resist stress. At that point the
element is deleted from the mesh. The nail and the steel side member use a bilinear isotropic plasticity
model. The material models are considered bilinear because strain hardening is defined after the
material reaches the yield point. If the materials are elastic-perfectly plastic, the model could be
unstable. Having a small amount of stiffness in the nail and plate after yielding increases the stability of
the model. The constants for all steel materials are defined in Table 2.

Table 2: Nail and plate steel material properties

<table>
<thead>
<tr>
<th></th>
<th>Elastic Modulus (MPa)</th>
<th>Yield Strength (MPa)</th>
<th>Post Yield Stiffness (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate (MPa)</td>
<td>200,000</td>
<td>310</td>
<td>500</td>
</tr>
<tr>
<td>Nail (MPa)</td>
<td>200,000</td>
<td>517</td>
<td>500</td>
</tr>
</tbody>
</table>

An elastic modulus of 200,000 MPa is commonly used in most studies for both steel plates and nails
(Hong and Barrett 2010; Sandhaas 2012). The yield strengths of the nails and plates are assumed from
Hong and Barrett (2010). The post yield stiffness is assumed to be small and is .25% of the elastic
modulus. The nails used by Hong and Barret (2010) were smooth shank nails and were tested in bending
to determine the material properties. An FEA model was developed and benchmarked against the
experimental data. This FEA model was used to determine the material properties of the nail. The
results were not conclusive, but a best estimate was used. It would be even more difficult to determine
accurate material properties using this method for the ring shank nails in this study because of the
geometry of the annular rings and residual stresses in the nail from the manufacturing process.

There are a number of wood material models developed by previous researchers. As previously
discussed, an elastic-plastic Hill yield criterion, Hoffman yield criterion and Hashin damage mechanics
are considered (Hill 1948; Schellekens and Borst 1990). Anisotropic elastic-perfectly plastic means that
after yielding the wood would have zero stiffness. This can make the model unstable and unrealistic
plastic deformations occur. A Hoffman yield criterion is not an available material model within ABAQUS.
A user-defined subroutine would need to be constructed. When considering that this model only has
loading of the anisotropic material in compression, investing the time into writing the subroutine
seemed unnecessary. Hill and Hoffman criteria are formulated in a similar way, and if the model will only
need one failure criterion for compression, then the difference between using Hill and Hoffman yield
criteria is negligible. Hill and Hoffman yield criteria cannot simulate the brittle failure modes of wood. Hashin damage mechanics provides a better model for wood because it allows for material softening and failure in ductile and brittle failure modes. Hill and Hoffman criterion cannot remove elements from the mesh, which causes them to become distorted and nonphysical after large deformations. Nonphysical is when elements fold in on themselves and pass through their own faces which is physically impossible. Using a Hill material model, the wood at large local deformations will always cause the analysis to end in an error with highly deformed elements. The Hashin material model in ABAQUS requires the use of two-dimensional continuum shell elements stacked on top of each other to model a three-dimensional object. Deformations can still occur along the y-axis (i.e. vertical direction of loading), the x-axis (i.e. transverse direction), and the z-axis (i.e. parallel to the length of the nail). By deleting elements from the mesh after they have completely softened, the problem of highly distorted elements is avoided.

The material constants for Douglas-fir are in Tables 3 through 5. Both the failure criteria and linear-elastic properties of Douglas-fir were derived from the Wood Handbook (Forest Products Laboratory 2010). It is assumed that the specific species of Douglas-fir is Coastal and at 12% moisture content. The fracture energy for each direction is determined from data from other wood species. Douglas-fir does not have published data for fracture energy. The fracture energy for Spruce from Ivanov and Sadowski (2009) is used instead for all but the longitudinal compression. The longitudinal compression fracture energy for Spruce used by Sandhaas (2012) was found to perform better in their study. Sandhaas (2012) used a much higher fracture energy, 60N/mm versus 1.6N/mm used by Ivanov and Sadowski (2009). Ivanov and Sadowski (2009) stated that the fracture energy parameters in their model needed refinement. The fracture energy parallel-to-grain is difficult to measure with consistent results (Sandhaas 2012). Initial models in their study showed that the 1.6N/mm value acted in a brittle manner
and created a considerable amount of noise in the data. The value used by Sandhaas (2012) led to better initial results numerically, so 60N/mm was used.

Table 3: Linear-elastic properties of Douglas-fir. Direction 11 is longitudinal, 22 is tangential, 33 is radial.

<table>
<thead>
<tr>
<th></th>
<th>E11 (MPa)</th>
<th>E22 (MPa)</th>
<th>E33 (MPa)</th>
<th>Nu12</th>
<th>Nu13</th>
<th>Nu23</th>
<th>G12 (MPa)</th>
<th>G13 (MPa)</th>
<th>G23 (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Douglas-fir</td>
<td>14700</td>
<td>737</td>
<td>1000</td>
<td>0.449</td>
<td>0.292</td>
<td>0.374</td>
<td>1150</td>
<td>943</td>
<td>103</td>
</tr>
</tbody>
</table>

Table 4: Failure criteria for Douglas-fir

<table>
<thead>
<tr>
<th>Failure Criteria (MPa)</th>
<th>Parallel Tension</th>
<th>Parallel Compression</th>
<th>Matrix Tension</th>
<th>Matrix Compression</th>
<th>Longitudinal Shear</th>
<th>Transverse Shear</th>
</tr>
</thead>
<tbody>
<tr>
<td>108</td>
<td>108</td>
<td>29.8</td>
<td>2.3</td>
<td>5.5</td>
<td>7.8</td>
<td>7.8</td>
</tr>
</tbody>
</table>

Table 5: Fracture energies assumed for Douglas-fir

<table>
<thead>
<tr>
<th>Fracture Energy</th>
<th>Longitudinal Tension</th>
<th>Longitudinal Compression</th>
<th>Transverse Tension</th>
<th>Transverse Compression</th>
</tr>
</thead>
<tbody>
<tr>
<td>Douglas-fir (N/mm)</td>
<td>1.6</td>
<td>60</td>
<td>0.06</td>
<td>0.2</td>
</tr>
</tbody>
</table>

3.2.1.2 Geometry
The dimensions of the modeled wood main members are identical to the physical samples, 38mm x 89mm x 230mm with the nail embedded 76mm from the top face. The dimensions of the steel side members are also identical to the physical samples. The Simpson Strong-Tie CNA4x60 is an annular ring shank nail that is approximately 4mm diameter with a taper from the shank to the nail head. The rings are not modeled because they would lead to ill formed elements and potentially an unstable geometry at higher deformations. A sketch of the cross section of the nail can be seen in Figure 8. The connection assembly is shown in Figure 10.
There is a gap left between the head of the nail and the face of the plate as well as a gap between the taper of the nail and the hole in the plate, as shown in Figure 12. This is modeled to more accurately represent the geometry of the test samples.

**Figure 12: Gap between plate and nail.**

**Time-step and Solver Controls**

The ABAQUS software has many different types of time-stepping methods that can be categorized as explicit or implicit. Explicit time-steps are fixed in time increment and do not have a check for force balance at the end of each step. This can lead to convergence problems if the time-step is chosen poorly. An implicit time-step will vary the time increment each step based on the convergence of the previous step. Convergence is checked each step by performing a force balance and comparing it to given allowable error criteria. If the model converges relatively quickly the increment will increase in size. If the model fails to converge the time increment will decrease (ABAQUS 2014).

For this model a dynamic, explicit time-step solution is chosen for a number of reasons. It calculates a large number of small time increments that are relatively low in computational cost compared to dynamic implicit. The smaller time-steps help make the model more stable. It can model contact surfaces where local instabilities at interacting surfaces can control the solution. The explicit dynamic time-step procedure automatically calculates a stable time-step based on the most critical element. The
most critical element is generally the smallest element in the mesh. Because the stability of the model is based on the ratio of the smallest dimension in the mesh and the wave speed of the material, the smaller the elements, the smaller the needed time-step. Even though the explicit method does not perform a self-check after each time-step to confirm the results are valid, having many more, smaller time-steps usually means this is not a concern (ABAQUS 2014).

3.2.1.4 Contact Interactions
The model consists of three different parts, the CNA4x60 nail, the steel plate side member, and the wood main member. To make sure they transfer load to one another, the contact must be defined. Element deletion also adds another layer of complexity to the interactions at the contact surfaces.

Element deletion requires the use of general contact descriptions within ABAQUS. General contact does not define contact between specific regions. It defines contact laws between every external and internal surface within the model. Because elements are deleted, the internal elements must also be included in the surface definition. If an exterior element fails and the element underneath it is not defined for contact, the object will simply pass through it (see Figure 13).

Figure 13: Newly exposed surface from element deletion (ABAQUS 2014)
General contact definitions greatly increase the computation time of the model, but in this case it is necessary to accurately model the interactions happening between the nail and the wood. The contact law that is used to describe the interaction has a normal and tangential definition. The normal contact is defined as “hard contact” which means that overclosure of elements and nodes is not allowed and separation of the contact surface is allowed if the two surfaces pull away from each other. There is no limit on contact force between the two surfaces. The tangential definition is simply friction with a coefficient of friction equal to 0.7. This is an assumed coefficient of friction that is commonly used for wood-steel interfaces (Hong and Barrett 2010).

3.2.1.5 Mesh
Whenever a model has curvature or circular geometry, the mesh must be constructed carefully to make the elements as cubic in shape as possible. The nail has cylindrical and conical features, and the plate and wood have circular holes. The geometry of the nail, plate and wood are partitioned into quarter sections. This allows the mesh generation algorithm within ABAQUS to handle the curved surfaces and produce a more uniform mesh (Figure 14).

Figure 14: Meshing procedure. Hole is quartered.
Elements used for the nail and the plate are continuum 3D stress elements labeled C3D8R in ABAQUS. It is an eight-node brick element with reduced integration. Reduced integration decreases the number of integration points in the element, which can decrease run time as well as remove artificial stiffness for plasticity calculations. This is beneficial because the material model used for the nails and the plate is elastic-plastic with post-yield hardening.

The 3D element used for the wood is a continuum shell element S8R. It is an eight-node shell element where the eight nodes define the thickness of the shell. A 3D stress element like the C3D8R is not used because the Hashin material can only be formulated with a shell element. This element is a combination of both a 3D element and a shell element so the Hashin material can be used while providing eight nodes that can provide the degrees of freedom of a true 3D element.

3.2.1.6 Applied Loads and Boundary Conditions
Finally, the loading and boundary conditions need to be defined. The boundary conditions restrain the model from movement and the loading applies the desired displacement in a similar manner as in the physical testing procedure. The wood has a roller boundary condition applied in the normal direction to every face except the bottom. This is to better represent the clamping and support the wood received from the test apparatus in the physical testing. The plate is loaded to a 20mm displacement specified at the top edge. A reference point at the top of the plate is used to apply the displacement. Using a reference point is a more efficient way to apply load or displacement over a group of nodes. One node in the middle of a group of nodes is used to apply the displacement to the whole group. The reference point is linked to the group of nodes so when the displacement is applied to the reference node, it is also applied to all the nodes it is linked with. This is also a way to make sure that plane sections remain plane within the set of elements. The reference node can be restricted to linear displacements as well as rotations. If the displacement was applied to the whole surface without the reference point, the plate could experience some shear stresses through numerical artifacts. The top edge is also constrained in
both horizontal axes so that the plate cannot move freely back and forth. To load the physical test, the plate was clamped at the top and could only move vertically. Constraining displacement in the horizontal axes more accurately represents the loading conditions of the physical test. The reference points and displacement loading can be seen in Figure 15.

![Figure 15: Left: Reference point four being linked to all nodes on the face. Right: Boundary conditions in blue and orange arrows. RP-1 has displacement applied vertically.](image)

### 3.2.1.7 Post Processing of Results

To capture the load-deflection curve, a macro is recorded within ABAQUS to quickly extract the results. To determine the total load, the reaction force in the vertical axis of reference point at the top edge of the plate is captured at each time-step. The reference point automatically sums all of the reaction forces and the displacement as well. The “COMBINE” command in ABAQUS is used to specify the displacement as the “x” values and the reaction force as the “y” values. This data can be plotted in ABAQUS or
exported to Excel or any other program that plots data. By recording this procedure as a macro, it only needs to be performed once. Any subsequent runs of the same model can use the macro to extract the load-deflection curves.

One problem with the model is the data recorded from the load deflection curve had what seemed like noise. The formulation of the continuum damage mechanics approach has a discontinuity after the damage initiation criterion is met as seen in Figure 3. This may be the cause of the noise in the data. After many failed attempts to solve the problem in ABAQUS by implementing mass scaling and decreasing the load rate, it was decided to use a filter to smooth the data. This was done using a built-in MATLAB filter script that performs a moving average. The nearest plus or minus 5 data points are averaged at each time interval to smooth the data.

3.2.2 MPM Input for Single Nail, Laterally Loaded
3.2.2.1 Material Model
The same material models are used for all of the steel parts within the MPM model as in the FEA model. The wood is a Hill material and is different than in the FEA model (Hill 1948). The same elastic material properties in Table 3 are used. The non-linear properties can be seen in Table 6. These are the yield stresses in each orthogonal axis and shear plane. After an element reaches one of the six defined stress states it will exhibit post-yielding hardening like that in Figure 1.

<table>
<thead>
<tr>
<th>Direction</th>
<th>$\sigma_{11}$</th>
<th>$\sigma_{22}$</th>
<th>$\sigma_{33}$</th>
<th>$\tau_{13}$</th>
<th>$\tau_{12}$</th>
<th>$\tau_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Douglas-fir (MPa)</td>
<td>29.8</td>
<td>4.5</td>
<td>4.5</td>
<td>3.2</td>
<td>3.2</td>
<td>1.3</td>
</tr>
</tbody>
</table>

The plate was assumed to be a completely rigid material around the nail head. Its sole purpose is to apply load to the nail and not deform. This was done so the reaction forces can be recovered. In NairnMPM, global reaction forces can be easily recovered from rigid materials. This adds some stiffness to the final load versus deflection curve.
3.2.2.2 Geometry and Material Points

The geometry is identical to the FEA model with a couple of exceptions. The plate is the same thickness, but because it is a rigid material, there is no reason to make the plate the full height, and it can just be the portion surrounding the nail head. Whether the plate is full height or just surrounding the nail head, it is applying the same amount force because it is a perfectly rigid material. This increases the size of the model for no reason. There are also small discrepancies in dimensions due to cell size resolution. Solid objects are discretized within the model by many material points, similar to pixels on a computer screen. The domain of the material points is determined by the cell size of the point. The larger the cell size the coarser the discretization. To keep the size of the model reasonable, the cell size is 1mm; however, not every cell within the model is 1mm X 1mm X 1mm. A “tartan grid” is used to decrease the overall size of the model and creates a gradient of cell sizes over a given space. For this model, the cell size decreases closer to the contact regions surrounding the nail shank as seen in Figure 16. This allows for more material points in the region of interest and decreases the number of material points in areas where they are not needed. Having less material points decreases calculation time.

Figure 16: Material point tartan grid defined by the MPM input file.
3.2.2.3 Time-step and Solver Controls
NairnMPM uses an explicit time-step, therefore careful attention must be taken in regards to the mass, time-step and loading rates of the simulation. If the loading rate is higher than the wave speed of the material, convergence will not occur. The wave speed of a material is a function of the density and stiffness.

3.2.2.4 Contact Interactions
The same contact controls were defined for the MPM model and FEA model. The wood-to-steel interaction has a coefficient of friction of 0.7. One of the advantages of MPM is it naturally handles normal surface to surface contact. It does not allow for overclosure of material point regions and allows for separation after contact is made. “Multi-material mode” was needed for this model. When there is more than one material in a model and contact is involved “multi-material mode” ensures that the velocity fields of the particles either remain separate for each material, or if there is contact the momenta of the material points are determined by specified contact laws and physics (Nairn 2015a).

When three or more materials are used in a model, problems can arise when all three material velocity fields share a node or group of nodes. There is not enough information in this situation to accurately define a surface normal that defines the contact regions of the interaction. NairnMPM uses an algorithm that can make predictions about the contact region but these can sometimes lead to errors. This model has a region that has three materials sharing nodes. There are no perceived errors from the results of the analysis due to “multi-material mode” mode from the analysis.

3.2.2.5 Applied Loads and Boundary Conditions
The MPM model has the same boundary conditions and loading as the FEA model.

3.2.2.6 Post Processing of Results
The load and time are recorded for each completed time-step using the rigid plate material. The displacement is not explicitly recorded but can be back-calculated from the load rate and the time from each time-step. The program Paraview is used to visualize the stress and strain contours.
3.2.3 ABR105 Connector
The same general modeling techniques used to create the single nail FEA model were used to create the more complex ABR105 connection model. The ABR105 is a common connector used in many CLT structures. It is used to connect CLT shear walls to CLT floor diaphragms.

3.2.3.1 Material Models
The wood is modeled as a Hashin material with the same properties as listed in Table 3 through Table 5. The nail and the steel side member use a bilinear isotropic plasticity model. The same material properties are used as those listed in Table 2. The ABR105 connector is modeled using the same material constants as the plate, and the steel used in both is assumed to be the same.

3.2.3.2 Geometry
The CLT is modeled as three layers of Douglas-fir that are 31.8mm in thickness. The CNA4x60 nail is used as the dowel fastener. The ABR105 bracket geometry is from Simpson Strong-Tie (2017). The geometry of the test specimen can be seen in Figure 17.
3.2.3.3 Time-step and Solver Controls
The same explicit, dynamic time-step procedure is used as for the single nail model. The ABR105 model has even more contact interactions and is a much larger model. Using an implicit time-step would be ill-advised. There is too much contact for the implicit solver to efficiently handle. The maximum time-step calculated by ABAQUS is smaller than in the single nail model. The element refinement around the nails requires smaller elements leading to a shorter time-step. As previously discussed ABAQUS selects a time-step based on the ratio of the smallest element and wave speed of the material. Generally the smallest element in the model will control the maximum time-step. The smaller the element the shorter the time-step and the longer the model must run to complete.

3.2.3.4 Contact Interactions
The same general contact laws are used as in the single nail connection. The only difference between the ABR105 model and single nail connection is the need for surface contact ties. Because CLT is comprised of three layers of wood that are glued together, the model ties together three layers of wood with a rigid link. The glue is not a perfectly rigid link, however, the physical testing revealed that the damage was localized to crushing of the wood fiber in the first layer of CLT and yielding in the connector and nails. Modeling glue lines would be more time intensive to input into ABAQUS and would increase the run time by adding cohesive elements with a computationally intensive cohesive material law. Because the physical testing revealed that most of the damage was in the first layer of CLT, almost non-existent in the second layer, and no damage in the third layer, it was assumed that modeling the glue lines would not have an appreciable effect on the results of the model. The surface contact ties can be seen in Figure 18.
3.2.3.5 Mesh
A similar mesh scheme is used for the ABR105 model as for the single nail. Continuum shell elements, denoted as SC8R in ABAQUS, are used for the wood while 3D stress elements, denoted as C3D8R, are used for the connector and nails. Meshing the connector and wood elements is challenging. The nail pattern dictated by the geometry of the bracket does not allow for the nails to be in straight, even rows. This makes it difficult to mesh the connector because the circular parts of the geometry cannot be as easily isolated. The meshing features in ABAQUS struggled even after creating many partitions. The best method for creating the mesh is to partition cubes around the nail holes to isolated those regions of mesh from the other regions, then work outwards portioning where necessary to maintain a uniform mesh.
3.2.3.6 Applied Loads and Boundary Conditions

The shear connection model is constrained at the top and bottom faces of the longest piece of CLT on the right side of Figure 19. The right piece of CLT in Figure 19 cannot translate in any of the three orthogonal directions. The displacement is specified at the top and bottom faces of the shorter piece of CLT on the left side. This is similar to the physical testing performed by Mahdavifar (2017) where the shorter piece of CLT was clamped on both edges and attached to the loading actuator. The same reference point methodology is used as in the single nail connection. This can be seen in Figure 19. The locations of the boundary conditions can be seen in Figure 20.

Figure 19: ABR105 Connection reference points
3.2.3.7 Post-Processing of Results

The same methods for recovering the load-displacement curves as described for the single nail model are used for the ABR015 connector. Load and displacement were recovered from the reference points and combined into the load-deflection curve.

Figure 20: Boundary conditions applied to the ABR105 connection. Vertical displacement is applied at the top and bottom of the left side. The right side is restrained at the top and bottom faces.
4 RESULTS AND DISCUSSION

4.1 Single Nail Physical Test Results

The load versus deflection plots of the physical data can be seen in Figure 21. The samples consisted of a Douglas-fir main member and a steel side member fastened by the CNA4x60. They were loaded laterally until failure.

The average initial stiffness of the samples is 780N/mm with a coefficient of variance (COV) of 17%. The average ultimate load is 4650N with a COV of 13%. These COV values are reasonable for wood testing. Depending on the material property, mechanical properties of wood can have a COV ranging from 10% to 30% (Forest Products Laboratory 2010). The minimum, maximum and average of the test results are also plotted in Figure 22. Table 7 shows the results of the physical test data as well as the results of the MPM and FEA models.

Qualitatively, all but one sample failed with the nail head shearing off. In this case, the wood sample split parallel-to-grain along a band of early wood directly above the nail. Photos of the failed test specimens are shown in Figure 23. The damage in the wood is localized around the nail and can be

![Figure 21: Physical test Load vs Deflection results.](image)
attributed to crushing of the wood fibers parallel-to-grain. The yield mode was III, with one plastic hinge located in the main member (AWC 2015). Because at least one sample split, it can be inferred that there is also the possibility for a tension perpendicular-to-grain failure.

The average of the 5% offset yield is 1870N with a coefficient of variance of 14%. This is calculated by using the initial stiffness of the sample as the slope of a line that begins at 5% of the nail diameter on the horizontal (displacement) axis and extends until it intersects the load-deflection plot. This intersection is the 5% offset yield load for the connection (Forest Products Laboratory 2010). An example of this calculation can be seen in Figure 24.

![Figure 22: Minimum, maximum, and average load-displacement plots for single nail tests](image)
Figure 23: Failed test specimens

Figure 24: 5% offset yield example of sample CNA4X60_1.
Table 7: Physical test data results

<table>
<thead>
<tr>
<th>Sample</th>
<th>Ultimate Load (N)</th>
<th>Initial Stiffness (N/mm)</th>
<th>5% Offset Yield (N)</th>
<th>Failure Mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>CNA4X60_1</td>
<td>3450</td>
<td>834</td>
<td>1880</td>
<td>Parallel to Grain Crack</td>
</tr>
<tr>
<td>CNA4X60_2</td>
<td>4600</td>
<td>733</td>
<td>1630</td>
<td>Nail Head Shear</td>
</tr>
<tr>
<td>CNA4X60_3</td>
<td>4690</td>
<td>611</td>
<td>2020</td>
<td>Nail Head Shear</td>
</tr>
<tr>
<td>CNA4X60_4</td>
<td>5210</td>
<td>725</td>
<td>1550</td>
<td>Nail Head Shear</td>
</tr>
<tr>
<td>CNA4X60_5</td>
<td>5150</td>
<td>780</td>
<td>1830</td>
<td>Nail Head Shear</td>
</tr>
<tr>
<td>CNA4X60_6</td>
<td>4740</td>
<td>994</td>
<td>2280</td>
<td>Nail Head Shear</td>
</tr>
<tr>
<td>Average</td>
<td>4640</td>
<td>780</td>
<td>1870</td>
<td>N/A</td>
</tr>
<tr>
<td>Standard Dev</td>
<td>637</td>
<td>128.29</td>
<td>265</td>
<td>N/A</td>
</tr>
<tr>
<td>COV</td>
<td>13%</td>
<td>17%</td>
<td>14%</td>
<td>N/A</td>
</tr>
<tr>
<td>FEA</td>
<td>2700</td>
<td>2800</td>
<td>1030</td>
<td>N/A</td>
</tr>
<tr>
<td>MPM</td>
<td>4110</td>
<td>17000</td>
<td>3030</td>
<td>N/A</td>
</tr>
</tbody>
</table>

4.2 MPM Single Nail Model

4.2.1 MPM Results
The results from the MPM model did not match the load-deflection behavior of the test data. The initial stiffness was over three times that of the average test data at 17000N/mm. The MPM analysis could not predict the ultimate load of the specimen because the simulation ended prematurely in an error. The 5% offset yield of the MPM model was 3030N which is almost twice the average of physical tests. Refer to Table 7 for direct comparisons of all the results. One numerical problem occurs with MPM when a material point exits the grid. Sometimes this can be due to not making the grid large enough for the analysis to allow for the deformations of all the material points, or it can be due to a material point detaching itself from the body of material points and shooting off the grid. The latter is a numerical problem at large deformations, and happened in this analysis. To alleviate the problem, the simulation must incorporate some form of material softening and damage mechanics.
The MPM analysis does exhibit some post-yielding behavior that is similar to the post-yielding behavior in the physical test samples. After the wood material begins to yield at around 1mm of displacement, the slopes of the MPM force-deflection data and the physical test data are similar. However, the model yields at almost twice the amount of force. This is evident in Figure 25.

4.2.2 MPM Discussion

![Figure 25: MPM analysis results compared to physical test data.](Figure 25: MPM analysis results compared to physical test data.)

The material model used for the wood in the MPM analysis was not validated against the experimental tests. A Hill elastic bilinear plastic material does not exhibit the realistic behavior of wood at this level of detail. If an entire CLT panel or large wood column or beam were to be modeled to yielding and not failure, a Hill material could work well. Modeling a larger structural member means that the relative size of the deformations is much smaller than in a detailed connection model. The overall strains are usually smaller than the localized ones examined in the study presented in this thesis for connections. At large deformations and strain the Hill material does not perform well for this application of connection modeling.
For the MPM model to work, the wood material needs to soften after a given failure criterion. Because the wood material maintains stiffness after yielding, the ultimate strength of the model is higher and the model maintains a high level of stiffness at larger displacements. The main issue is that the Hill material is not well suited to modeling a connection where complete failure of the material occurs, and it is not mathematically formulated to do so.

Even though the force versus displacement curve does not match the test data, qualitatively the visualization of the results matches in general what happened in the test as seen in Figure 26. The nail exhibits the similar overall displaced shape as in the physical test. It can be noted that the rigid plate is much stiffer by not allowing any rotation at the interface of the nail and plate. This will be further discussed in regards to the FEA model.

Figure 26: MPM von Mises stress cross section
4.3 FEA Single Nail Model

4.3.1 Global Results

The results of the FEA model matched the test data much better than the MPM model. Raw results from the analysis had a large amount of noise. This is believed to be some sort of numerical problem within ABAQUS and the Hashin material model. As stated in the Methods and Materials section, a filter is used to average the data so it is easier to compare with the physical tests. This is shown in Figure 27 and Figure 28.

![Figure 27: Raw FEA data and filtered FEA data](image)

The initial stiffness is still almost over three times that of the test data, at 2800N/mm. The FEA model then begins to soften more than the test data after 2mm or so of displacement as seen in Figure 28.

![Figure 28: Filtered FEA data compared to minimum, average, and maximum.](image)
Table 8 shows that between 2mm and 3mm the model matches the test data closely but diverges shortly after. The 5% offset yield from the model is also much lower than the average test data at 1030 and 1880N, respectively (Figures 29 and 24).

Table 8 shows the comparison between the FEA results and the average physical test data. The percent difference is calculated by subtracting the FEA data from the average force data from the tests and dividing by the average test data. The percent difference increases as the displacement increases.

<table>
<thead>
<tr>
<th>Displacement (mm)</th>
<th>0.5</th>
<th>1</th>
<th>1.5</th>
<th>2</th>
<th>2.5</th>
<th>3</th>
<th>3.5</th>
<th>4</th>
<th>4.5</th>
<th>5</th>
<th>5.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force Test Data (N)</td>
<td>700</td>
<td>1100</td>
<td>1380</td>
<td>1620</td>
<td>1810</td>
<td>1990</td>
<td>2150</td>
<td>2290</td>
<td>2410</td>
<td>2570</td>
<td>2710</td>
</tr>
<tr>
<td>Force Model (N)</td>
<td>1010</td>
<td>885</td>
<td>1250</td>
<td>1580</td>
<td>1620</td>
<td>1700</td>
<td>1640</td>
<td>1820</td>
<td>1910</td>
<td>2010</td>
<td>2150</td>
</tr>
<tr>
<td>Percent Difference</td>
<td>-43%</td>
<td>20%</td>
<td>9%</td>
<td>2%</td>
<td>11%</td>
<td>15%</td>
<td>24%</td>
<td>20%</td>
<td>21%</td>
<td>22%</td>
<td>21%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Displacement (mm)</th>
<th>6</th>
<th>6.5</th>
<th>7</th>
<th>7.5</th>
<th>8</th>
<th>8.5</th>
<th>9</th>
<th>9.5</th>
<th>10</th>
<th>10.5</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force Test Data (N)</td>
<td>2840</td>
<td>2960</td>
<td>3070</td>
<td>3170</td>
<td>3270</td>
<td>3380</td>
<td>3500</td>
<td>3600</td>
<td>3720</td>
<td>3840</td>
<td>3950</td>
</tr>
<tr>
<td>Force Model (N)</td>
<td>2070</td>
<td>2160</td>
<td>2360</td>
<td>2380</td>
<td>2400</td>
<td>2320</td>
<td>2480</td>
<td>2560</td>
<td>2630</td>
<td>2620</td>
<td>2640</td>
</tr>
<tr>
<td>Percent Difference</td>
<td>27%</td>
<td>27%</td>
<td>23%</td>
<td>25%</td>
<td>27%</td>
<td>31%</td>
<td>29%</td>
<td>29%</td>
<td>29%</td>
<td>32%</td>
<td>33%</td>
</tr>
</tbody>
</table>

Table 8: Percent difference between FEA data and average physical test data at 0.5mm increments
Figure 30 shows the progression of the deformed shape and von Mises stress distribution in the nail and the plate throughout the duration of the analysis. The 5% offset yield is shown in Figure 30 (c) and the final time-step is shown in (h). The von Mises stresses are plotted and do not differentiate between positive and negative stress, or tension and compression, respectively. The von Mises stress best represents the stress state of the elements within the nail and plate because it accounts for stresses in all three orthogonal directions as well as shear. Zones of red indicate regions that have reached the yield stress of the material and zones of plastic strain.

Figure 29: 5% offset yield of filtered FEA results.
Figure 30: The von Mises stresses from single nail FEA model. Displacement is at increments of 0.5mm for (a) through (g). (h) skips to the final time-step. (c) is at 5% offset yield. View is an isometric cross-section.
4.3.2 Plastic Strain in the Nail
The plastic strain in the nail model can also be compared to previous testing and calculations performed by Mahdavifar (2017). By plotting the equivalent plastic strain across the bottom of the nail, the location of the plastic hinge can be determined. The FEA mesh in Figure 31 shows a line of nodes along the bottom of the nail where the equivalent plastic strain is being collected to plot in Figure 32. Plastic strain is the amount of strain experienced by an element after it has yielded. Equivalent plastic strain is mathematically formulated similar to von Mises stress, and is the best way to visualize the plastic strain when multiple directions are exhibited. For example, the nail in Figure 31 has plastic shear strains near the nail head as well as plastic strain in tension near the mid-length. Plotting shear plastic strain or tensile plastic strain alone would not represent all of the plastic strains in the nail. It’s a more effective way to visualize the data.

Figure 31: Location of measurements and equivalent plastic strain plotted on the undeformed shape.

Figure 32 shows the equivalent plastic strain along the bottom of the nail (shown in red in the upper portion of Figure 31). There are high plastic shear strains just to the left of where the nail makes contact with the plate. The plastic hinge region begins at approximately 20mm and ends at about 30mm. The
calculation performed by Mahdavifar (2017) indicates that the plastic hinge should be located near 15mm from the nail head (noted by the red vertical line).

4.3.3 Stress Contours in the Wood

Stress contours help visualize how force is distributed through the connection from the application of the load to the boundary conditions. The stress contours in the wood one time-step before any elements are deleted from the mesh indicate some of the failure modes of the connection. The contour plots can be seen in Figure 33 through Figure 35. Negative values correspond to the blue region and compressive stress, while positive values of stress are the red regions and areas of tensile stress.

Figure 33 shows that parallel-to-grain there is a large compressive stress directly where the nail bears on the wood as expected. At the sides of the nail hole are two zones of very high tensile stress in red. Figure
34 shows that directly above and below the hole is a zone of tensile stress perpendicular-to-grain. The locations of high stresses in Figure 35 show that the failure will most likely not be due to shear as none of the elements reach a high enough stress in shear to exceed the failure criteria. The failure criteria are exceeded by compression parallel-to-grain stresses.

Figure 33: Stress parallel to grain

Figure 34: Stress perpendicular to grain
4.3.4 Single Nail FEA Discussion of Results

4.3.4.1 Global Results/Initial Stiffness

The large difference in initial stiffness can be attributed to a number of factors. Table 7 shows that the FEA model had an initial stiffness of 1680N/mm compared to 780N/mm of the average physical test. A likely contributor is the initial damage to the wood as the nail is hammered. This is extremely difficult to quantify or model. Whenever wood is, cut, drilled, hammered, etc., a layer of wood is damaged on the surface next to the contact surface, in this case with the nail. This layer has weaker material properties than the bulk material properties. The damaged layer of wood can decrease the initial stiffness of the sample (Hong and Barrett 2010; Sandhaas 2012). This is generally ignored in modeling because it is difficult to quantify how much this layer should be weakened and the extent of the weakened region (Hong and Barrett 2010; Sandhaas 2012). The boundary conditions of the FEA model are also perfectly rigid. It is impossible to design a test that has perfectly rigid connections to the test fixture. There will always be a small amount of softening in the overall physical results that can be attributed to the somewhat imperfect support conditions of the test sample. The final qualitative observation is that the nails hammered into the sample followed a band of early wood. Early wood has considerably softer than late wood (Nairn 2007b). The model uses bulk material properties throughout and does not model each...
individual band of early wood and late wood. The less stiff early wood could also contribute to the lower initial stiffness in the physical samples compared to the ABAQUS model.

The difference in results at larger deformations out to 10mm is likely due to the bulk material properties. After 3mm of displacement the model differs from the average test data by more than 15% as seen in Table 7. The assumptions made about the fracture energy of Douglas-fir most likely have an effect on the performance of the model at larger displacements. It could be that Douglas-fir has more ability to absorb energy after reaching the failure criterion than Spruce. Douglas-fir has higher strength properties than Spruce in the Wood Handbook and the NDS (Forest Products Laboratory 2010; AWC 2015). This could also account for the large difference in the 5% yield offset. A higher fracture energy would mean that elements can sustain a higher level of stress before failure, and the entire model support a higher load before yielding.

4.3.4.2 Plastic Strain in the Nail
The difference in plastic hinge location between the NDS and the model seems to be primarily an issue with how the measurement is being taken (from the start of the bend or the back of the bend, and a straight line from the head of the nail or following the curve of the bend). A simple way to verify this is to superimpose an image of an actual, failed nail over the model and compare the two. This is shown in Figure 36. They are almost identical. It is also evident that the nail rotated about the head. This helped soften the global restraint at the end of the nail model. The taper and gap that was left between the nail
and the plate in Figure 12 allowed for the nail to rotate as the plate moved upward. This decreased the reaction force/moment at the plate initially because the nail was not bearing directly on the wood or the plate until it settled into its final position. The general behavior of the FEA nail model is very similar to the test. The discrepancies in the single nail connection model and test data are coming primarily from the wood properties because the nail behaved so similarly.

4.3.4.3 Stress Contours
The most likely form of failure of the wood is in compression directly above the nail. This is expected and desirable because the failure is generally ductile. Brittle connection failures should be avoided in design. Crushing wood fibers is a typical way to incorporate ductility. Another result from the FEA analysis shows that tension stresses perpendicular-to-grain develop above and below the nail. Tension perpendicular-to-grain leads to a more brittle failure. The maximum tension stresses perpendicular-to-grain were 98% of the failure criterion defined for the material in the model (2.28 MPa from the analysis versus the 2.30 MPa failure criterion). This could lead to a brittle failure in tension perpendicular-to-grain and splitting the sample instead of crushing the wood parallel-to-grain. This was seen in one of the physical test samples as it suddenly split down the middle.
4.4 FEA Model of ABR105 Connection

4.4.1 Results

4.4.1.1 Global Results
The load versus deflection plot does not match the test data collected by Mahdavifar (2017) as shown in Figure 37. The initial stiffness is 3180 N/mm for the FEA model and 2880 N/mm average for the test data, which is only a 9.2% difference. However, the model does not soften/yield at the same rate as the physical test data. At 5mm of lateral displacement, the model begins to yield but at almost twice the force value than the test data. The data collected from Mahdavifar (2017) are backbone data from cyclic tests of the ABR105 connection in Douglas-fir CLT. The model is a monotonically loaded.

4.4.1.2 Group Action
In Chapter 11 of the NDS, connection behavior with fasteners in rows is evaluated. (AWC 2015). When multiple dowel fasteners are oriented parallel to the direction of load, the exterior dowels carry a higher proportion of the load than the interior dowels (AWC 2015). This negatively impacts the overall strength of the connection. The ABR105 is not a simple linear row of dowels. However, the nails are staggered and the loading is perpendicular to the staggered rows. Moreover, the nails are spaced closely together and are of larger diameter than common nails.

![Figure 37: FEA vs test data for ABR105 connection](image)
One of the benefits of FEA modeling is being able to quantify the complex interactions/contributions of individual elements/aspects/portions of the model. At 1mm of displacement and at the last time-step, at approximately 5mm of displacement, the contact force between each nail and the ABR105 connector was determined. The contact stress was recorded over the surface of the nail and summed together to determine the contact force over the nail. The numbering of the nails is shown in Figure 38 and the percent contribution of each nail to the overall nail resistance is shown in Table 9 and Table 10. The percent contribution of each nail is relative to the total force on either the wall/left, or floor/right side of the connection. The row number for the nails is counted from the center (fold) of the connection moving outwards for both the left and right sides. Images of the von Mises stresses in the ABR105 connector and nails for equally spaced time-steps are shown in Figure 39 and Figure 40. The displacement increases logarithmically with equally spaced time-steps. It can be seen in Figure 39 (c) and Figure 40 (c) that the nail forces in the inner rows (closest to the fold in the connector) are higher than for the outer rows. This corresponds to the contact forces recorded in Table 9 and Table 10. It is also evident that after the connection begins to yield the force is more equally distributed across all of the dowels. After 5mm of displacement the percent contribution of both sides of the connection are much closer to each other and the stress distributions in the nails in Figure 39 and Figure 40 are much more uniform.

Table 9: Percent contribution of each nail on the left side

<table>
<thead>
<tr>
<th>Nail Number</th>
<th>Percent Force Contribution of Each Nail at 1 mm and final time-step</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nail 1</td>
</tr>
<tr>
<td>Percent Contribution at 0.15mm</td>
<td>19%</td>
</tr>
<tr>
<td>Percent Contribution Final Time-step</td>
<td>10%</td>
</tr>
<tr>
<td>Row Number</td>
<td>1</td>
</tr>
</tbody>
</table>
Table 10: Percent contribution each nail on the right side

<table>
<thead>
<tr>
<th>Nail Number</th>
<th>Nail 11</th>
<th>Nail 12</th>
<th>Nail 13</th>
<th>Nail 14</th>
<th>Nail 15</th>
<th>Nail 16</th>
<th>Nail 17</th>
<th>Nail 18</th>
<th>Nail 19</th>
<th>Nail 20</th>
<th>Nail 21</th>
<th>Nail 22</th>
<th>Nail 23</th>
<th>Nail 24</th>
</tr>
</thead>
<tbody>
<tr>
<td>Percent Contribution at 0.15mm</td>
<td>14%</td>
<td>15%</td>
<td>14%</td>
<td>10%</td>
<td>9%</td>
<td>9%</td>
<td>7%</td>
<td>9%</td>
<td>8%</td>
<td>1%</td>
<td>1%</td>
<td>2%</td>
<td>0%</td>
<td>0%</td>
</tr>
<tr>
<td>Percent Contribution Final Timestep</td>
<td>6%</td>
<td>7%</td>
<td>7%</td>
<td>9%</td>
<td>7%</td>
<td>8%</td>
<td>8%</td>
<td>8%</td>
<td>6%</td>
<td>8%</td>
<td>6%</td>
<td>6%</td>
<td>6%</td>
<td>6%</td>
</tr>
<tr>
<td>Row Number</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>3</td>
<td>4</td>
<td>4</td>
<td>5</td>
<td>5</td>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

Figure 38: Nail numbering scheme
Figure 39: The von Mises stresses in the ABR105 connection at equally spaced time increments.
Figure 40: The von Mises stress in only the nails for equally spaced time increments
4.4.1.3 The von Mises Stresses and Damage Variables

Visualizing the stress and damage variables in the wood can help understand the potential failure modes of the wood and connection. The areas that directly bear on the nail holes not only exhibit compression perpendicular-to-grain stresses but also tension parallel-to-grain stresses. These are shown in Figure 41 and Figure 42 as load is being applied vertically on the left side. The compression parallel-to-grain stresses are shown in Figure 43. It can be seen that the compression zones interact with one another.

Figure 41: Tension parallel to grain above nail holes

Figure 42: Tension damage parallel to grain
even though the holes are staggered. In Figure 44 there is compressive stress (blue) perpendicular-to-grain where the nails are bearing on the wood and tension on the opposite side (red).

Figure 43: Compression parallel to grain stress.

Figure 44: Stress perpendicular to grain

Figure 45 shows the locations of damage perpendicular-to-grain. The regions in red can be seen not only where the nails bear on the wood but also where the ABR105 makes contact with the wood. These are the square shaped regions that are near the intersection of the two CLT panels of the connection.
4.4.1.4 ABR105 Stress Contours
The von Mises stresses in the connector can be seen in Figure 46. This shows that most of the stress is concentrated on the rib and fold areas as well as at the locations of bearing on the nails.

4.4.2 Discussion
4.4.2.1 Global Results
Differences in the global results of initial stiffness and strength of the connection are most likely due to three key reasons. The first two have already been discussed in the single nail model. Damage to the wood occurs when the nails are driven. This is not modeled and could produce more stiffness in the model than in the actual test specimen with the softened areas from the nailing. The second factor is the assumed values for Douglas-fir fracture energy. The third possible reason could be that the test data are from backbone curves extracted from cyclic test data as discussed in section 2.4. This could account

Figure 45: Damage perpendicular to grain
for some of the discrepancy, but backbone curves are generally similar to monotonic test data, and wouldn’t account for almost 1.5 times the force in the FEA model compared to the test. The reason why the load-deflection curve is above that of the physical test data at greater deformations is hard to determine because the opposite occurred in the previous single nail model. The FEA single nail model had lower ultimate and yield strengths than the physical test data. This can be seen in Table 7 and Figure 28. The ABR105 model did not follow this trend even though the modeling techniques used in both are exactly the same. The ABR105 bracket plays a much larger role then the steel plate in the single nail model. The bracket is digging into the sides of the CLT which could be providing more resistance to load. The plate in the single nail model stays almost perfectly vertical and slides across the surface of the sample. Physical test samples are never constructed perfectly, unlike the model which has perfectly rigid restraints and well-defined contact surfaces. Any combination of these reasons could account for more stiffness and strength in the model.
4.4.2.2 Group Action

The contact forces on the nails recorded at 0.15mm displacement show that group action does occur in this connection. The ABR105 bracket is not a classic example of a connection that would exhibit this behavior as discussed in the results section 4.4.1.2. However, the results in Tables 9 and 10 show the phenomenon. It was assumed initially that the inner nails (closest to the fold) of the connection in rows one and two on would contribute more resistance than the outer nails in rows four, five, and six. This proved to be correct. It is shown in Tables 9 and 10 the outermost nails in rows five and six contribute much less in the early stages of loading. As the connection yields at 5mm of displacement the force is distributed more uniformly to all of the nails. The nails in rows 1 and 2 have reached yielding and cannot provide more resistance, therefore the nails in rows four, five and six begin to increase significantly in their contributions.

This can be seen in the stress contours as well. Figure 39 (c)-(e) and Figure 40 (c)-(e) qualitatively illustrate the same progression. In Figures 39 (c) and 40 (c), the locations of yield stress, which are a von Mises stress of 517 MPa, are shown in red. Rows one and two have a higher concentration of red while the outer rows four, five and six are mostly blue. If the nails are not highly stressed then they are not contributing significantly to the resistance of the connection. This indicates that the nails in the first two rows are contributing to the resistance of the connection in greater force than the outer rows. In Figures 39 (e) and 40 (e) the stresses in red in each nail show that the resistance from each nail is now distributed more evenly.

4.4.2.3 Stress and Damage Variable

The tension parallel-to-grain stresses above the bearing surface of the nails in Figure 41 are to be expected. The wood material is much stiffer and stronger parallel-to-grain. As the material is crushed perpendicular-to-grain the Poisson's effect will mean that there is stress induced parallel-to-grain stress in tension. This tension zone is most likely not an area to worry about a brittle failure. Even if there is a
tension failure there are more layers of wood that the nail must move through and crush before it completely fails. A tension failure from dowel bearing parallel-to-grain can have a zipper effect, meaning the crack can propagate to the end of the sample suddenly. Wood is much weaker in tension perpendicular-to-grain than in tension parallel-to-grain. A tension failure parallel-to-grain from dowel bearing does not exhibit this behavior. Inducing a tension failure by crushing through the wood perpendicular-to-grain is much more difficult than crushing the wood parallel-to-grain.

The compression zones induced parallel-to-grain stresses and from Figure 43 show some interaction between each other even though they are staggered. This may not be of concern since the main failure is compression perpendicular-to-grain and not parallel-to-grain. It may help to increase the stagger of the holes and avoid having the holes in any sort of continuous line to reduce and interaction between the holes.

The damage perpendicular-to-grain in Figure 45 is to be expected around the locations of bearing with the nails. The damage perpendicular-to-grain away from the holes is most likely from the ABR105 connector. As the connection begins to yield the plate is deformed and also rotates, digging into both sides of the CLT. The rotation is due to eccentricity of the connection when loaded laterally. This is not a concern for failure of the connection.

### 4.4.2.4 ABR105 Behavior Compared to the Single Nail

The ABR105 connector and single nail models are both constructed using the same modeling techniques. This does not mean that the same behavior is always observed. For example, the location of plastic strain in a nail from the connector model and the single nail model are shown in Figure 47.
The nails in the figure are bent in opposite directions because the load was applied to the nails in opposite directions. This should not influence the location of plastic strain along the length of the nail. These images are taken at times when the nails are at a similar level of deformation, not when the connection has deformed the same amount. The nail head of each nail has moved approximately 0.5mm. This makes for a more direct comparison to how the nails behave in either model. The nail from the bracket connection is typical of the plastic strain distribution of the nails in the connection.

The difference could be due to many factors. The ABR105 bracket and the surrounding nails, to some extent, all influence one another directly and indirectly. As previously discussed in section 4.4.2.2 and 4.4.2.3 the connection suffers from some effects of group action and the stress contours in the wood interact with one another. This could potentially weaken the wood elements surrounding the nail, allowing the nail to bend and yield more easily, which could shift the plastic strain region.
The bracket could also be applying a force to the nail that is not purely lateral. The ABR105 bracket has an eccentricity when loaded laterally. This caused a rotation in the plate as discussed in section 4.4.2.3. This rotation could be applying an axial load to the nails and affecting the location of plastic strain and yielding. The single nail model does not allow for any rotation of the plate as it is loaded.

The single nail model was bearing directly on wood fibers parallel to grain while the ABR105 model had the nail bearing on both perpendicular and parallel to grain fibers. Loading fibers perpendicular to the grain is a less stiff and weaker direction relative to parallel-to-grain.

The reason for the discrepancy in plastic strain location could be the cause of the discrepancy in the ABR105 model and physical testing results. If the nails in the connection are only laterally loaded the plastic strain location should be similar. There is a complex interaction occurring in the connection model that requires further investigation.

5 SUMMARY AND CONCLUSIONS
5.1 Single Nail Testing and Models
The single nail FEA model showed the most promising results of all the models. It most closely matched the physical test data. Hill or Hoffman material models do not account for the localized damage to the wood which is imperative to capturing the effects of dowel bearing (Hill 1948; Schellekens and Borst 1990). Continuum damage mechanics is able to model brittle and ductile failure modes simultaneously which most material models are not able to do. Paired with element deletion, continuum damage mechanics is a well-known approach to modeling wood in FEA software packages (Sandhaas 2012). This study used the commercially available material model known as “Hashin Damage” in ABAQUS (ABAQUS 2014; Hashin, Z. 1980; Hashin, Z., and Rotem, A. 1973).

The MPM model failed to match the test data most likely because a Hill material was used to model the wood. Even though MPM is expected to be more effective at high deformations, the analysis was aborted with an error because the deformations were too great. The MPM model could benefit from a
continuum damage mechanics approach or another form of material failure. Having a material that does not fail results in errors at high deformations regardless of using FEA or MPM.

There is a region of wood surrounding the dowel that is damaged by the installation of the nail. This is a region that is hard to quantify and leads to a decrease in the initial stiffness of the connection (Hong and Barret 2010; Sandhaas 2012). The model was not able to capture the effects of this phenomena.

When a dowel is loaded parallel-to-grain, tension forces are also induced perpendicular-to-grain. This could lead to a sudden brittle failure of the connection. This was evident as one of the test samples failed suddenly in tension parallel to grain.

5.2 ABR105 Connection Model
The ABR105 model was not able to match the test data from previous studies (Mahdavifar 2017). The model was too stiff initially and had a much higher yield force. The greater initial stiffness of the ABR105 FEA model may be attributed to comparing the FEA results to cyclic backbone data, a lack of initial damage in the wood and using fracture energies that are for Spruce and not Douglas-fir. The greater yield force is difficult to explain because the single nail model was weaker than the test data. It is difficult to pinpoint a reason in a large complex model. The same reasons for the high initial stiffness may also apply to the higher yield force. Although the model did not match test data quantitatively, there are still qualitative insights that can be drawn from the results.

The connection displayed group action effects. The innermost rows of nails (closest to the fold in the connector) carried significantly more load than the nails in the outer rows. It was only after the overall connection yielded that the force was more evenly distributed. Even though the nails are staggered, the compression zones parallel to the grain, that surround the nails to the left and right of each hole, still interact with each other. Staggering the nails further could avoid this and lead to even more equal sharing of load.
The tension stresses induced in the parallel-to-grain direction from compression perpendicular to grain are close to initiating the failure criterion of the wood, however they do not necessarily lead to failure of the connection as a whole. The wood surrounding the failed elements is still stiff and strong enough to be able to bear the load from the nails. This is the opposite of what was observed in the single nail model. The tension stresses in the single nail model were induced perpendicular to grain which is a significantly weaker direction and could lead to sudden brittle failure.

The nails in the ABR105 model and the single nail test did not behave similarly. The region where the nail exhibited plastic strain is different. The ABR105 model exhibits plastic strain further down the length of the nail. This could be due to the effects of the bracket and surrounding nails, eccentricity in the connection applying axial load to the nail or, the grain orientation of the wood bearing on the nail.

5.3 Recommendations
Both of the FEA models were computationally expensive as well as the MPM model. The FEA connection models required the use of server clusters to be able to perform the analysis in a more timely manner. The single nail model would take three or more days running on 24 CPU’s to complete and the ABR105 model would need more than one to two weeks running on 48 CPU’s to complete. Finding ways to reduce the computational costs of the model would be beneficial to the continuation of connection modeling research. Use of the general contact algorithm in ABAQUS greatly increased the run time. It is necessary for element deletion. Investigation into possible ways to limit the need for the general contact feature could lead to much shorter run times.

Finding ways to reduce the number of elements needed in the CLT pieces would also be beneficial. A better meshing scheme that efficiently allows for a fine mesh in areas of interest and coarse mesh elsewhere would help. This is a challenge because the connector studied staggers the nails in such a way that it forces the use of fine mesh over large areas. The right balance of mesh refinement and
convergence would help as well as it was never fully achieved. The run times were far too long and the number of elements was far too high.

5.4 Future Work
The models used in this study are far from ideal or complete in their development. Important information can still be gleaned from them but there is always room for improvements. Further study into modeling initial damage and quantifying a damaged region in the physical specimen is needed. This is theorized to be a main reason why the initial stiffnesses were too great.

Doing further material testing to quantify the fracture energy of Douglas-fir could also lead to better results. Using Spruce values was done as an expedient but may not accurately represent the wood material in the model. Modeling the bands of earlywood and latewood may also more accurately represent the wood material.

ABAQUS has a commercially-available material model for continuum damage mechanics in orthotropic materials formulated by methods from Hashin (1980) and Hashin and Rotem (1973). It is only available for use with 2D shell elements. Implementing a 3D material model in a 3D stress element may provide better results. The Hashin material model had a large amount of noise in the results and may be alleviated by using a 3D element with a similar material formulation and a continuous damage initiation criterion.

The MPM model would benefit from using a continuum damage mechanics approach. The Hill material was not able to perform the analysis effectively. Modeling the plate out of steel instead of a rigid material would also be more accurate. Finding a way to reduce the material point resolution further around the nail and plate is also needed.

Making the previously mentioned improvements to the model before moving forward with new ideas and different models is necessary. After improving the agreement of the model with the test data, it
could then be used for sensitivity and parameter studies to improve designs. Comparing the effects of changing the wood species in the CLT, nail diameter, nail length, plate thickness, nail patterns or many other variables in the model could lead to better connection design in the future without the need to physically test hundreds of samples.
REFERENCES


Forest Products Laboratory, USDA Forest Service. (2010). Wood handbook: wood as an engineering material. Madison, WI.


Appendix I Input Files
Input files exclude the list of node locations for brevity.

Single Nail Model Input File
**
** ELEMENT CONTROLS
**
*Section Controls, name=EC-1, ELEMENT DELETION=YES, MAX DEGRADATION=0.99
1., 1., 1.
*Section Controls, name=EC-2, ELEMENT DELETION=YES
1., 1., 1.
*Amplitude, name=Amp-1, definition=SMOOTH STEP
0., 0., 1., 1.
**
** MATERIALS
**
*Material, name=NAIL
*Density
8.05e-08,
*Elastic
200000., 0.3
*Plastic
500., 0.
500.5, 0.001
501., 0.002
501.5, 0.003
502., 0.004
502.5, 0.005
503., 0.006
503.5, 0.007
533., 0.066
533.5, 0.067
534., 0.068
534.5, 0.069
535., 0.07
535.5, 0.071
536., 0.072
536.5, 0.073
537., 0.074
537.5, 0.075
538., 0.076
538.5, 0.077
539., 0.078
539.5, 0.079
540., 0.08
540.5, 0.081
541., 0.082
541.5, 0.083
542., 0.084
542.5, 0.085
543., 0.086
543.5, 0.087
544., 0.088
544.5, 0.089
545., 0.09
545.5, 0.091
546., 0.092
546.5, 0.093
547., 0.094
547.5, 0.095
548., 0.096
548.5, 0.097
549., 0.098
549.5, 0.099
550., 0.1
550.5, 0.101
551., 0.102
551.5, 0.103
552., 0.104
552.5, 0.105
553., 0.106
553.5, 0.107
554., 0.108
554.5, 0.109
555., 0.11
555.5, 0.111
556., 0.112
556.5, 0.113
557., 0.114
557.5, 0.115
558., 0.116
558.5, 0.117

*Material, name=PLATE
*Density
8.05e-08,

*Elastic
200000., 0.3

*Plastic
<table>
<thead>
<tr>
<th>Value</th>
<th>Measurement</th>
</tr>
</thead>
<tbody>
<tr>
<td>300.0</td>
<td>0.0</td>
</tr>
<tr>
<td>300.3</td>
<td>0.001</td>
</tr>
<tr>
<td>300.6</td>
<td>0.002</td>
</tr>
<tr>
<td>300.9</td>
<td>0.003</td>
</tr>
<tr>
<td>301.2</td>
<td>0.004</td>
</tr>
<tr>
<td>301.5</td>
<td>0.005</td>
</tr>
<tr>
<td>301.8</td>
<td>0.006</td>
</tr>
<tr>
<td>302.1</td>
<td>0.007</td>
</tr>
<tr>
<td>302.4</td>
<td>0.008</td>
</tr>
<tr>
<td>302.7</td>
<td>0.009</td>
</tr>
<tr>
<td>303.0</td>
<td>0.01</td>
</tr>
<tr>
<td>303.3</td>
<td>0.011</td>
</tr>
<tr>
<td>303.6</td>
<td>0.012</td>
</tr>
<tr>
<td>303.9</td>
<td>0.013</td>
</tr>
<tr>
<td>304.2</td>
<td>0.014</td>
</tr>
<tr>
<td>304.5</td>
<td>0.015</td>
</tr>
<tr>
<td>304.8</td>
<td>0.016</td>
</tr>
<tr>
<td>305.1</td>
<td>0.017</td>
</tr>
<tr>
<td>305.4</td>
<td>0.018</td>
</tr>
<tr>
<td>305.7</td>
<td>0.019</td>
</tr>
<tr>
<td>306.0</td>
<td>0.02</td>
</tr>
<tr>
<td>306.3</td>
<td>0.021</td>
</tr>
<tr>
<td>306.6</td>
<td>0.022</td>
</tr>
<tr>
<td>306.9</td>
<td>0.023</td>
</tr>
<tr>
<td>307.2</td>
<td>0.024</td>
</tr>
<tr>
<td>307.5</td>
<td>0.025</td>
</tr>
<tr>
<td>307.8</td>
<td>0.026</td>
</tr>
<tr>
<td>308.1</td>
<td>0.027</td>
</tr>
<tr>
<td>308.4</td>
<td>0.028</td>
</tr>
</tbody>
</table>
308.7, 0.029
309., 0.03
309.3, 0.031
309.6, 0.032
309.9, 0.033
310.2, 0.034
310.5, 0.035
310.8, 0.036
311.1, 0.037
311.4, 0.038
311.7, 0.039
312., 0.04
312.3, 0.041
312.6, 0.042
312.9, 0.043
313.2, 0.044
313.5, 0.045
313.8, 0.046
314.1, 0.047
314.4, 0.048
314.7, 0.049
315., 0.05
315.3, 0.051
315.6, 0.052
315.9, 0.053
316.2, 0.054
316.5, 0.055
316.8, 0.056
317.1, 0.057
317.4, 0.058
317.7, 0.059
318., 0.06
318.3, 0.061
318.6, 0.062
318.9, 0.063
319.2, 0.064
319.5, 0.065
319.8, 0.066
320.1, 0.067
320.4, 0.068
320.7, 0.069
321., 0.07
321.3, 0.071
321.6, 0.072
321.9, 0.073
322.2, 0.074
322.5, 0.075
322.8, 0.076
323.1, 0.077
323.4, 0.078
323.7, 0.079
324., 0.08
324.3, 0.081
324.6, 0.082
324.9, 0.083
325.2, 0.084
325.5, 0.085
325.8, 0.086
Material, name=WOOD_1
*Damage Initiation, criterion=DUCTILE
0.01, 6., 0.1
*Damage Evolution, type=DISPLACEMENT
0.5,
*Density
5.3e-10,
*Elastic, type=ENGINEERING CONSTANTS
14740., 737., 1002., 0.449, 0.292, 0.374, 1150., 943., 103.,
*Plastic
29.668,0.
57.668,2.
*Potential
1., 0.152, 0.152, 0.18682, 0.18682, 0.0759
*Material, name=WOOD_1-HASHIN
*Damage Initiation, criterion=HASHIN
107.6, 29.8, 2.3, 5.5, 7.8, 7.8
*Damage Evolution, type=ENERGY
1.6, 60., 0.06, 0.2
*Density
5.3e-10,
*Elastic, type=ENGINEERING CONSTANTS
14740., 737., 1002., 0.449, 0.292, 0.374, 1150., 943., 103.,
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=STEEL-STEEL
*Friction
0.3,
*Surface Behavior, pressure-overclosure=HARD
*Surface Interaction, name=WOOD-STEEL
*Friction
0.7,
*Surface Behavior, pressure-overclosure=HARD
**

** BOUNDARY CONDITIONS
**

** Name: Boundary Conditions Type: Displacement/Rotation
*Boundary
Set-60, 1, 1
Set-60, 2, 2
Set-60, 3, 3
Set-60, 4, 4
Set-60, 5, 5
Set-60, 6, 6
**

** STEP: Step-1
**

*Step, name=Step-1, nlgeom=YES
*Dynamic, Explicit, element by element
, 1.
*Bulk Viscosity
0.06, 1.2
** Mass Scaling: Semi-Automatic
** Whole Model
*Variable Mass Scaling, dt=1e-07, type=uniform, frequency=1
**
** BOUNDARY CONDITIONS
**
** Name: Loading Type: Displacement/Rotation
*Boundary, amplitude=Amp-1
Set-55, 1, 1
Set-55, 2, 2, 20.
Set-55, 3, 3
Set-55, 4, 4
Set-55, 5, 5
Set-55, 6, 6
*Adaptive Mesh Controls, name=Ada-1
1., 0., 0.
**
** INTERACTIONS
**
** Interaction: Int-5
*Contact, op=NEW
*Contact Inclusions
SURFALL ,
*Contact Property Assignment
, , WOOD-STEEL
**
** OUTPUT REQUESTS
**
*Restart, write, number interval=1, time marks=NO
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, time interval=0.001
*Node Output
A, RF, U, V
*Element Output, directions=YES
DAMAGEFC, DAMAGEFT, DAMAGEMC, DAMAGEMT, DAMAGESHR, EMSF, EVF, LE, PE, PEEQ, PEEQVAVG, PEVAVG, S, STATUS, SVAVG
*Contact Output
CSTRESS,
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
MPM Single Nail Model

<?xml version="1.0"?>

<!DOCTYPE JANFEAInput SYSTEM "C:\Users\Sean\Desktop\VisTool\bundle\bundle\NairnMPM.dtd">

<JANFEAInput version="3">
<Header><Description>Title: Nail in Shear User Name: Sean Hollenbeck Modeling nail test using OSParticulas</Description><Analysis>12</Analysis></Header>

<MPMHeader><MPMMethod>2</MPMMethod><GI type="lCPDI"/></GiMP type="/ArchiveRoot>/ArchiveTime>1.50000e-05</ArchiveTime><MPMArchiveOrder>iYYYYYNYYNNN</MPMArchiveOrder><CrackArchiveOrder>iYYNN</CrackArchiveOrder><GlobalArchive type="contactx"/><GlobalArchive type="contacty"/><GlobalArchive type="contactz"/></GlobalArchive><MultiMaterialMode RigidBias="100" Normals="2" Dcheck="1" Vmin="0"><Friction
law="#1"/></Friction>
<ContactPosition>0.92</ContactPosition></MultiMaterialMode>

<Mesh output="file"><Grid zmax="19" zmin="-19" ymax="76" ymin="-76" xmax="94.5" xmin="0"><Horiz style="1" rx="1.5" nx="189"/><Vert symmin="-76" ny="304"/><Depth symmin="0" nz="76"/><Border zmax="" zmin="1" ymax="4" ymin="2" xmax="2" x1="1"/><AreaOfInterest nx="" ny="24" nz="24" z2="6" z1="-6" y2="6" y1="-6" x2="94.5" x1="26"/></Grid></Mesh>

<MaterialPoints>

<Body vz="38" vy="0" vx="0" mat="5"><Cylinder zmax="2" zmin="-2" ymax="92.5" y1="-76" x2="94.5" x1="26" radius="-0.8"/></Body>

<Body vz="38" vy="0" vx="0" mat="3" angle="90"><Box zmax="19" zmin="-19" ymax="76" y1="-69" x2="92.5" x1="89" angle="1"/></Box>

<Body vz="38" vy="0" vx="0" mat="7"><Box zmax="6" zmin="-6" ymax="92.5" x1="89" x2="92.5" y2="-6" x2="94.5" y1="-69" x1="26" angle="1"/></Box></Body>

ABR105 Input File

**

** ELEMENT CONTROLS

**

*Section Controls, name=EC-1, ELEMENT DELETION=YES

1., 1., 1.

*Amplitude, name=Amp-1, definition=SMOOTH STEP

0., 0., 0.1, 1.
** MATERIALS

*Material, name=NAIL
*Density
  8.05e-09,
*Elastic
  200000., 0.3
*Plastic
  517., 0.

*Material, name=PLATE
*Density
  8.05e-09,
*Elastic
  200000., 0.3
*Plastic
  250., 0.

*Material, name=WOOD11
*Density
  5.3e-10,
*Elastic, type=ENGINEERING CONSTANTS
  14740., 737., 1002., 0.449, 0.292, 0.374, 1150., 943., 103.,
*Plastic
  29.668, 0.
  57.668, 2.
*Potential
  1., 0.152, 0.152, 0.18682, 0.18682, 0.0759

*Material, name=WOOD_2
*Density
5.3e-10,

*Elastic, type=ENGINEERING CONSTANTS
10400., 152., 566., 0.449, 0.292, 0.374, 550., 450.
50.,

*Plastic
29.668,0.
57.668,2.

*Potential
1., 0.152, 0.152, 0.18682, 0.18682, 0.0759

*Material, name=WOOD_HASHIN

*Damage Initiation, criterion=HASHIN
107.6, 29.8, 2.3, 5.5, 7.8, 7.8

*Damage Evolution, type=ENERGY
1.6, 60., 0.06, 0.2

*Density
5.3e-10,

*Elastic, type=ENGINEERING CONSTANTS

14740., 737., 1002., 0.449, 0.292, 0.374, 1150., 943.
103.,

**

** INTERACTION PROPERTIES

**

*Surface Interaction, name=STEEL-STEEL

*Friction
0.3,

*Surface Behavior, pressure-overclosure=HARD

*Surface Interaction, name=WOOD-STEEL

*Friction
1.,
*Surface Behavior, pressure-overclosure=HARD
*Surface Interaction, name=WOOD-WOOD
*Friction
0.5,
*Surface Behavior, pressure-overclosure=HARD
**
** INTERACTIONS
**
** Interaction: Int-1
*Contact, op=NEW
*Contact Inclusions
SURFALL ,
*Contact Property Assignment
, , WOOD-STEEL
**  ****************************************************************************
**
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=YES
*Dynamic, Explicit
, 0.1
*Bulk Viscosity
0.06, 1.2
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
Set-14, 1, 1
Set-14, 2, 2
Set-14, 3, 3
Set-14, 4, 4
Set-14, 5, 5
Set-14, 6, 6

** Name: BC-3 Type: Displacement/Rotation

*Boundary, amplitude=Amp-1

Set-19, 1, 1
Set-19, 2, 2, 20.
Set-19, 3, 3

**

** OUTPUT REQUESTS

**

*Restart, write, number interval=1, time marks=NO

**

** FIELD OUTPUT: F-Output-1

**

*Output, field, time interval=0.005

*Node Output
A, RF, U, V

*Element Output, directions=YES
DAMAGEC, DAMAGEFC, DAMAGEFT, DAMAGEMC, DAMAGEMT, DAMAGESHR, DAMAGET, EVF, LE, PE, PEEQ, PEEQAVG, PEAVG, S, SVAVG

*Contact Output
CSTRESS,

**

** HISTORY OUTPUT: H-Output-1

**
*Output, history, variable=PRESELECT

*End Step
Appendix II Videos of Models

YouTube link for various videos of model results:

https://www.youtube.com/channel/UCnmSpVozDpponpuDgZkMi8w/playlists?view_as=subscriber