Mechanical characterisation of bear bones

MAGDALENA KAPLAN
Mechanical characterisation of bear bones

MAGDALENA KAPLAN
Abstract

Brown bears go into hibernation for several months during the winter period, but regain all bodily functions shortly after waking up, such as the strength of bones. The aim of this thesis has been to characterise the material of active bear’s bones (tibiae) by destructive testing and then fitting a damage model by the use of finite element simulations. Standard beam theory was used to compare with the simulated results and supplemental compression testing was conducted to verify elastic parameters. Examination of results show quite a large distribution in both material parameters and determined stresses for the tested bones, with elastic moduli varying 3-10 GPa, Poisson’s ratio 0.3-0.45, strain for onset of damage 1-2%, damage rate factor of 25-40 and fracture stresses varying proportionally with stiffness between 50-190 MPa.
Sammanfattning

Brunbjörnar går i ide i flera månader under vinterhalvåret, men kort efter vintervilans slut återgår deras kroppar, och däribland ben och skelett, till en normal funktion. Målet med detta arbete har varit att karaktärisera materialet i björnarnas skenben genom förstörandeprovning och inversmodellering med en skademodell. Även balkteori användes för att jämföra med simulerade resultat, samt kompletterande kompressionsprovning för att verifiera elastiska materialparametrar. Granskning av resultaten visar en stor variation i både materialparametrar och spännings i de provade benen, med elasticitetsmodul varierande mellan 3-10 GPa, tvärkontraktionstal på 0.3-0.45, töjning för initiering av skada på 1-2%, skadeutvecklingsgrad på 25-40 och brottspänningar varierande proportionellt med styvheten mellan 50-190 MPa.
Acknowledgements

Many people have been helpful to me during the time of writing this thesis; first and foremost, I would like to thank my supervisor Professor Per-Lennart Larsson, for the guidance, support and many discussions we have had during these months, as well as some heartfelt laughs. Further, I would like to acknowledge Dr. Mathias Haarhaus for taking the time to explain aspects beyond the field of solid mechanics in the project to me and researcher Rodrigo Moreno for a great deal of help with the medical image processing software, as well as researcher Artem Kulachenko for always having an answer for my problems anyhow related to performing simulations.

I would also like to thank the laboratory manager at the Solid Mechanics Laboratory Martin Öberg for quick execution of testing and answering my many questions, as well as laboratory technician Jörgen Jansson, for preparing my specimens, despite the unpleasant smell. Finally, I would like to thank my classmates, Gustav Hultgren and Håkan Jernberg in particular, and my family for support and interesting conversations leading to great progress in my work.
Contents

1 Introduction 1
   1.1 Background .................................................. 1
   1.2 Tools ......................................................... 2

2 Methods 3
   2.1 Four-point bending test and evaluation using standard beam
       theory ......................................................... 3
       2.1.1 Test set-up ............................................. 3
       2.1.2 Estimation of fracture stresses ....................... 5
       2.1.3 Image analysis and polygon creation .................. 7
       2.1.4 Second area moment of inertia ......................... 8
   2.2 FE-simulation for constitutive characterisation .......... 9
       2.2.1 3D-model creation ..................................... 9
       2.2.2 Material model .......................................... 12
       2.2.3 Boundary conditions .................................... 14
       2.2.4 Implementation and evaluation of simulation results . 15
   2.3 Elastic parameter verification by compression tests ........ 16
       2.3.1 Stress-strain relationship determination ............. 17

3 Results 20
   3.1 Bending test .................................................. 20
       3.1.1 Estimation of fracture stresses ....................... 20
   3.2 FE-simulation of bending test ................................ 21
   3.3 Compression tests ............................................ 22

4 Discussion 23

5 Conclusions 26

Bibliography 27
A Methods
A.1 Bending set-up . . . . . . . . . . . . . . . . . . . . . . . . . . 28
A.2 Compression specimens . . . . . . . . . . . . . . . . . . . . 29

B Results
B.1 FEM-fit of load curve . . . . . . . . . . . . . . . . . . . . . . . 31
B.2 Stress and damage plots . . . . . . . . . . . . . . . . . . . . . . 33
B.3 Compression test evaluation . . . . . . . . . . . . . . . . . . . 36
Chapter 1

Introduction

This thesis was performed during the period of February to June of 2019 at the Department of Solid Mechanics at the School of Engineering Sciences, KTH Royal Institute of Technology in Stockholm, in cooperation with the Scandinavian Brown Bear Research Project. The main objective has been to determine material parameters of brown bear tibiae (shin bones, further referred to purely as "bear bones" or "bones"), using testing and simulation techniques.

1.1 Background

The Scandinavian Brown Bear Research Project has followed, monitored and researched brown bears since 1984. One highly interesting objective in the research concerning bears is the difference between the bear’s body properties during the active "summer" period and the hibernation "winter" period. Since bears go into hibernation for about six months, it is surprising how many of the bodily functions remain intact and/or regain their former properties over a very short period of time upon activation after the hibernation is over. This project considers a small part of the research about the bone properties; the aim has been to characterise the material in bear bones by identifying basic material parameters, such as elastic modulus and the Poisson's ratio and also defining how damage develops in the bone, by the use of a damage model. An interesting aspect is the comparison between summer (active) and winter (hibernating) bears’ bone properties. However, since the latter is more difficult to acquire, this study is limited to the mechanics of summer bones.

Apart from aspects concerning purely the quality of the research and convenience, the ethics of the project are of high importance. No bears were harmed purely for this project or this thesis, but since the testing requires com-
plete specimens, taking only samples from living bears is not possible. The bones are acquired from sources where the individuals are already killed for other purposes. The summer bears are, in most parts, acquired during hunting season. The drawback here is that the knowledge of those bears is limited, since only gender and approximate age can be known. Another source for the research objects is Orsa Predator Park, where the animals live in their natural habitat and can be considered as wild; the medical history of the bear is known, but the behaviour and way of living of the individuals may diverge from those truly living in the wild.

1.2 Tools

The testing was performed at the Solid Mechanics Laboratory and was led by laboratory manager Martin Öberg. The software used for computational parts of the projects are presented in Table 1.1.

Table 1.1: Software used in project.

<table>
<thead>
<tr>
<th>Software</th>
<th>Version</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Matlab</td>
<td>r2018b</td>
<td>Calculations</td>
</tr>
<tr>
<td>MeVisLab</td>
<td>3.1.1</td>
<td>Medical image processing</td>
</tr>
<tr>
<td>ANSYS SpaceClaim</td>
<td>18.2</td>
<td>3D-model adjustments</td>
</tr>
<tr>
<td>ANSYS Worbench</td>
<td>18.2</td>
<td>Simulations</td>
</tr>
</tbody>
</table>
Chapter 2

Methods

As stated before, the objective of this project is to characterise the material in bear bones. The bones were subjected to testing in both bending and compression, separately. In an initial step, the fracture stresses from the bending test were estimated using standard beam theory. For further, more reliable, evaluation of the bending tests, it was replicated with finite element simulations, where material parameters were fitted to reach agreement in response between the physical testing and simulation. An appropriate damage model was chosen to describe the inelastic behaviour. Additionally, compression tests were performed, to compare linear material parameters with those concluded from the simulation.

2.1 Four-point bending test and evaluation using standard beam theory

Seven bones were tested during the first round of testing which was limited to bending testing only. These bones were given the names Bear1, Bear1v and the remaining five were numbered Bear2-Bear6. The specimens will be further referred to by these names in the report. However, not all bones were evaluated in all further steps because of various limitations and eligibility. Which bones were excluded and why will be explained in each coming section.

2.1.1 Test set-up

The bending tests were performed during 2016 in the Solid Mechanics Laboratory at KTH, Stockholm. Complete tibiae (shinbones) from seven wild bears
with varying ages were tested under four-point bending. All bones tested during the bending test qualify as "summer bears", since the bones were acquired months subsequent to the hibernation period, when the effects on the mechanical properties of the long rest have vanished. The set-up of the four-point bending test can be seen in Figure 2.1 and the specific parameters for each test set is shown in Table 2.1. A picture taken of the real test can be seen in Appendix A.1.

Figure 2.1: Set up of the bending test. Displacement was applied to the upper, smaller roller pair.

Table 2.1: Test-set up dimensions. *Unreliable value, probably wrongly recorded. See Figure 2.1 for explanations of the presented parameters.

<table>
<thead>
<tr>
<th></th>
<th>$L_y$ [mm]</th>
<th>$L_i$ [mm]</th>
<th>$D_y$ [mm]</th>
<th>$D_i$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bear1</td>
<td>128.5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bear1v</td>
<td>120.5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bear2</td>
<td>60.5*</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bear3</td>
<td>112.5</td>
<td>10</td>
<td>40</td>
<td>40.5</td>
</tr>
<tr>
<td>Bear4</td>
<td>129.7</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bear5</td>
<td>129.7</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bear6</td>
<td>129.7</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The advantage of a four-point bending test over the conventionally used three-point bending test is the fact that the moment is constant between the
two middle rolls, since they both act with the same force on the bone. This moment can be determined as

\[ M_y = \frac{P(L_y - L_i)}{4}, \] (2.1)

where \( P \) is the resulting force.

### 2.1.2 Estimation of fracture stresses

From the standard four-point bending test, it is possible to determine the stresses in the cross section using beam theory, when the moment acting in between the two loads in the bone is known. The stress at a point in the cross section is defined as

\[ \sigma(x) = \frac{M(x)}{I(x)} z, \] (2.2)

where \( M(x) \) is the total moment acting on the beam, \( I(x) \) is the second area moment of inertia of the cross section and \( z \) is the vertical distance between the centre of gravity of the cross section and the investigated point. However, the above equation is only valid for simple bending, i.e. when the material is isotropic, only deforms elastically, has a cross section that is the same throughout the beam and also is symmetrical around the centre of gravity. None of these conditions are fulfilled for this case, making this analytical solution a mere approximation of the actual stress state. For the sake of finding a simple, though to some extent reliable method, it is used and later on compared to FEM-results, which better reflect the reality of the stress state.

In the case of uniaxial bending, and in this case bending around the \( y \)-axis, the Equation (2.2) can be simplified and sought for the maximum stress in the cross section as

\[ \sigma_{\max} = \frac{M_y}{I_y} z_{\max}, \] (2.3)

where \( M_y \) is the applied moment as explained in Section 2.1, \( I_y \) the second area moment of inertia in the \( y \)-direction and \( z_{\max} \) the maximum vertical distance between the centre of gravity of the cross section and a point in the cross section. Determining the second area moment of inertia is not straightforward for a complex cross section, as it includes calculating an integral:

\[ I_y = \int z^2 dA. \] (2.4)
When dealing with more complex shapes, the parallel axis theorem is very useful, as it allows separate computation for individual bodies and then summarises them with respect to the distance between the centre of gravity of the complete and partial bodies. The principle of discretising an arbitrary area body this way is shown in Figure 2.2.

The parallel axis theorem, as shown previously, can be summarised in the equation:

\[
I_\eta = \sum_i (I_{yi} + b_i^2 A_i).
\]  (2.5)

One method of determining the second area moment of inertia for a complex cross section is therefore to try to divide the shape manually into sections with known properties, i.e. simple geometric shapes; rectangles, triangles etc., and then use the formula given in Equation (2.5) to obtain the total value. Doing so, however, requires a lot of attention for each analysed cross section and the results might be inconsistent as the cross sections most likely differ in shape. In order to automate this process, a Matlab-script was written, doing mainly two things:

- finding boundaries of the cross-sectional images and transforming those into polygon shapes with the real dimensions, and
• iterating the second area moment of inertia with Equation (2.5) by the use of rectangles fitted inside the polygon shape.

2.1.3 Image analysis and polygon creation

A bone cross section can roughly be described as a thick-walled cylinder; the outer layer is made of strong, thick cortical bone, while the centre is made up of porous, non-load bearing cancellous (trabecular) bone. Prior to testing each of the tested bones was photographed using a CT-scan, which allows imaging of the cross section. The scans were conducted by Torkel Brismar, M.D., Ph.D, Associate Professor at the Department of Clinical Science, Intervention and Technology at Karolinska Institutet. When the bending tests were finished, the localisation of the fracture was determined and the cross sectional images from this location were then post-processed to obtain the shape of the cross-section. The images were loaded into a Matlab-script, where a reference length in the image was set, allowing for conversion from dimensions in pixels to real dimensions. Further, the cross section was cropped out, converted to a binary image, then filtered by transforming contours in the image to closed curves and filtered once more by removing small shapes. Finally, the boundaries of the resulting image were chosen, allowing for optical inspection and further processing if the chosen boundaries were not correct. The final boundaries then created a polygon, which includes only the cortical part of the cross section. The filtering process is illustrated in an example in Figure 2.3.
2.1.4 Second area moment of inertia

As previously mentioned, the cross sectional property $I_y$ was determined using the parallel axis theorem as given in Equation (2.5), by the use of fitting rectangles inside of the polygon determined in the previous step (see Figure 2.4a for example). This was done iteratively, by specifying how many times the cross section should be divided in the vertical and horizontal directions, creating a grid over the cross section (see Figure 2.4b). The rectangles created in this grid were then examined; if the centre point of the rectangle fell within the cross section its contribution to the second area moment of inertia was computed, by the use of Equation (2.5) (see Figure 2.4c). When the whole grid was examined, the resulting second area moment of inertia was compared to the previous step and if the relative error was over a set value, the numbers of divisions, $N_x$ and $N_y$, were increased and the process was repeated until the error was sufficiently small. The area of the rectangles was also computed.
and compared with the actual area of the polygon, to check for convergence additionally. The process is illustrated in Figure 2.4.

Figure 2.4: Stepwise illustration of geometric estimation of cross section, $y$ and $z$ are the total lengths in the horizontal and vertical directions respectively, $N_y$ and $N_z$ are the number of divisions for each direction respectively.

### 2.2 FE-simulation for constitutive characterisation

In order to calculate reliable material parameters and include nonlinear effects, a finite element simulation was conducted for appropriate specimens (see Results, Section 3.1.1 for which bones were excluded and why). The resulting force in the simulation was compared to that of the physical testing; the idea was to adjust material parameters until agreement between the two was reached.

#### 2.2.1 3D-model creation

The models used in the simulations were generated from CT-scanned images of the bones. The file used as the basis for this process were segmented images, meaning that only the load-bearing, cortical bone was left with an even intensity and all other parts were filtered out. These segmentations were created by Rodrigo Moreno, Ph.D., Docent, at the department of Biomedical Engineering and Health Systems, KTH Royal institute of Technology. The images were then processed using the medical and image processing software MeVisLab. Because of the size of the bones, the CT-images could not capture each
bone completely in one scan, the bones where therefore scanned in three turns, capturing the distal, middle and proximal parts of the bone separately. The first step in the model creation was therefore to connect the images into one, leaving a continuous and clear bone. In the next step, the segmented images were cropped, so that the middle of the bone was the only remaining part, i.e. the joints were omitted, mainly because the segmentation did not capture them completely, leaving a discontinuous bone. Further, the bones were transformed to isosurfaces by the use of the built-in WEM-format (Winged Edge Mesh). In order to assure a nice surface without any peaks, which may have caused problems in the simulation, smoothing and polygon reduction were applied to the WEM. Finally, the surface mesh object was saved as a stereolithography-file (STL), allowing for import into SpaceClaim; the built-in geometry editing software in Ansys Workbench. The schematic of this process can be seen in Figure 2.5, which shows how the usage of built-in models in MeVisLab was used to achieve this shell-model.

As the geometry was, at this point, only a surface object and not a solid, the model was after the export to SpaceClaim transformed to a solid, using a built-in function. After this step, the rollers used for the bending test were drawn with real dimensions, leaving the 3D-model complete for analysis. An example of the complete 3D-model can be seen in Appendix A.1.
Figure 2.5: Schematic of shell-model creation in MeVisLab.
2.2.2 Material model

Bone, as a material, behaves rather differently than for instance metal, since the nonlinear deformation is not caused by plasticity, but rather development of microcracks in the structure: both in tension and compression. However, the initiation of nonlinear behaviour occurs later in compression than in tension [1]. This means that the behaviour is not symmetric; it behaves different in tension and in compression. As the bone is loaded in bending, it is subjected to both compression (on the upper side) and tension (on the bottom), but since it can endure higher loading in compression without fracture, the tensile behaviour is assumed to be critical. Therefore, the bone has here been modelled symmetrically, as the conclusions drawn will be conservative.

In order to create a reliable model of the bending tests, the material model used had to exhibit microcrack damage behaviour. In order to simplify the procedure, a model predefined in ANSYS was used, allowing fairly easy implementation. This model is called the microplane model, as it assumes several microscopically small planes in the material, where the degradation is progressing under applied load. These planes are then integrated over the whole surface of the material, resulting in degraded material properties. The basics of the model, which was developed by Zdenek, et.al. [2],[3], are explained below. The stress-strain relationship is defined as below

\[
\sigma = \frac{3}{4\pi} \int_{\Omega} \frac{\partial \psi^{\text{mic}}}{\partial \varepsilon} d\Omega,
\]  

(2.6)

where \( \sigma \) is the stress, \( \Omega \) the surface of the microplanes, \( \varepsilon \) the strain and \( \psi^{\text{mic}} \) is the microscopic free energy, which is defined as

\[
\psi^{\text{mic}}(\varepsilon_V, \varepsilon_D, d^{\text{mic}}) = (1 - d^{\text{mic}}) \psi^{\text{mic}}(\varepsilon_V, \varepsilon_D).
\]

(2.7)

Here, \( \varepsilon_V \) and \( \varepsilon_D \) correspond to the volumetric and deviatoric strains respectively, while \( d^{\text{mic}} \) is the damage variable at microscale, defined by

\[
\left\{ \begin{align*}
    d^{\text{mic}} &= 1 - \frac{\gamma_0^{\text{mic}}}{\eta^{\text{mic}}} \left( 1 - \alpha^{\text{mic}} + \alpha^{\text{mic}} e^{\beta^{\text{mic}}(\gamma_0^{\text{mic}} - \eta^{\text{mic})})} \right), \\
    0 &\leq d^{\text{mic}} \leq 1.
\end{align*} \right.
\]

(2.8)

In Equation (2.8) \( \gamma_0^{\text{mic}} \), \( \alpha^{\text{mic}} \) and \( \beta^{\text{mic}} \) are microplane material parameters and \( \eta^{\text{mic}} \) defines the equivalent strain as

\[
\eta^{\text{mic}} = k_0 I_1 + \sqrt{k_1^2 I_1^2 + k_2 J_2},
\]

(2.9)
where $I_1$ is the first invariant of the strain tensor, $J_2$ is the second invariant of the deviatoric part of the strain tensor; $k_0$, $k_1$ and $k_2$ are material parameters of the microplane model, which determine how the material behaves in compression as compared to tension. As explained previously, these parameters have here been chosen so that the behaviour is symmetrical, i.e. the same equivalent strain is computed for both compressive and tensile loads. Finally, Equation (2.6) can be evaluated to determine the stress state in the material. It can also be interesting to determine the damage in the whole material at macroscale, which is done by integrating the microscale damage variable over the surface of the microplanes:

\[
\begin{align*}
    d^{\text{mac}} &= \frac{1}{4\pi} \int_{\Omega} d^{\text{mic}} \, d\Omega, \\
    0 &\leq d^{\text{mac}} \leq 1.
\end{align*}
\] (2.10)

As can be seen in Equation (2.7), the damage is not initiated unless the microscopic damage parameter $d^{\text{mic}} > 0$, this means that up to a certain point, the material behaves elastically with linear material parameters. This behaviour will then be described by Hooke’s law:

\[
\sigma = C \varepsilon,
\] (2.11)

where $C$ is the stiffness tensor defined by material parameters $E$ - elastic modulus and $\nu$ - Poisson’s ratio. The onset of the material degradation is as seen in Equation (2.8) $\eta^{\text{mic}} \geq \gamma^{\text{mic}}$. Important to mention is that the microplane model only allows values for the Poisson’s ratio smaller than 0.5. All material parameters for the damage model are summarised in Table 2.2.

Table 2.2: Microplane model parameters. Values marked with "*" are determined individually for each specimen, remaining are fixated as indicated.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Given value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$</td>
<td>*</td>
<td>Young’s modulus for elastic deformation.</td>
</tr>
<tr>
<td>$\nu$</td>
<td>* (&lt;0.5)</td>
<td>Poisson’s ratio for elastic deformation.</td>
</tr>
<tr>
<td>$k_0$</td>
<td>0</td>
<td>See Equation 2.9.</td>
</tr>
<tr>
<td>$k_1$</td>
<td>0</td>
<td>See Equation 2.9.</td>
</tr>
<tr>
<td>$k_2$</td>
<td>1</td>
<td>See Equation 2.9.</td>
</tr>
<tr>
<td>$\gamma^{\text{mic}}_0$</td>
<td>*</td>
<td>Strain at initiation of damage, see Equation 2.8.</td>
</tr>
<tr>
<td>$\alpha^{\text{mic}}$</td>
<td>1</td>
<td>Maximum allowed damage, see Equation 2.8.</td>
</tr>
<tr>
<td>$\beta^{\text{mic}}$</td>
<td>*</td>
<td>Rate of damage, see Equation 2.8.</td>
</tr>
</tbody>
</table>
2.2.3 Boundary conditions

In order to get trustworthy results from the simulation, the initial idea was to model the test set-up exactly as in reality, i.e. one bone and four rollers, see Figure 2.1. However, this introduces several problems related to the created contact regions. The bones’ complex geometry together with the high applied displacement results in numerical issues. The rollers are, compared to the bone, stiff, which means that when the bone deforms, more nodes come in contact with the rollers; modelling this sort of development requires a great knowledge about contact problems and is also very time consuming, as new contact definitions constantly need to be added; otherwise the rollers might penetrate the bone. Therefore, the model was simplified by replacing the lower (bigger) roller pair with a remote displacement (RD1), applied directly to the previous bone-roller contact surfaces. The upper rollers were not replaced, as this would lead to model instability due to very strict boundary conditions, but also since the contact region was here much simpler than at the bigger rollers. The contact between the rollers and bone was defined by the linear contact No Separation; allowing sliding between the contacts, but not separation in the normal direction. A remote displacement was then applied to the upper rollers as well (RD2), locking them in place. A remote displacement in Ansys has six degrees of freedom (DOFs): displacement in the $x$-, $y$- and $z$-directions and rotations around those axes. See Figure 2.6 for a visualisation of the boundary conditions and Table 2.3 for complete parameters applied in those regions.

Table 2.3: Defined boundary conditions with their DOFs; $rot_i$ indicates rotation around the $i$-axis, see Figure 2.1 for definition of the axes directions. *Control variable: applied displacement.

<table>
<thead>
<tr>
<th>DOF</th>
<th>Behaviour</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$x$ [mm]</td>
</tr>
<tr>
<td>RD1</td>
<td>Free</td>
</tr>
<tr>
<td>RD2</td>
<td>0 0 0 0 0</td>
</tr>
</tbody>
</table>

Deformable

Rigid
2.2.4 Implementation and evaluation of simulation results

When the set-up of the simulation was completed, the characterisation was initiated. The process of determining the material parameters consisted of trying different material parameters, as defined in Table 2.2, against an applied displacement corresponding to that endured during the physical experiments. The resulting force was recorded and then visually compared to the experimental results; adjustments were then made to the parameters, until sufficient agreement was reached between the two.

The resulting force was measured by applying a reaction force probe at the regions of applied displacement (defined as RD1 in Table 2.3). This region was chosen for force measurement because of how Ansys estimates forces in the model; the resulting stresses are integrated over a surface, which means
that the total force in the model might not be zero (even in the absence of rigid body motion), if singular stress concentrations appear. Because of difficulties at the contact regions at the upper rollers, it was evident that such singularities were present, which was confirmed by examining the strain energy; it was consistently unevenly distributed among elements at the contacts.

In order to evaluate and compare the results to that acquired from standard beam theory (see Section 2.1.2), two values were chosen for comparison:

- the maximum principal stress in the cross section with the maximum strain at the bottom of the bone at maximum load and
- the maximum principal stress in the cross section with the maximum strain at the bottom of the bone at highest load foregoing onset of non-linear deformation.

These two points were chosen in order to determine if standard beam theory yields reasonable results, even if all requirements for simple bending are not fulfilled; the difference being one of the cases estimating the stresses at inelastic deformation. The results were then compared to the stresses estimated at corresponding displacements.

2.3 Elastic parameter verification by compression tests

Compression testing was done as a complement to the bending test, but only on some of the specimens. After the completion of the bending tests, the remains of the specimens were sliced into a few millimetres thick slices, which could then be tested in compression. In order to get reliable results and to optimise the amount of testing performed, only a few of the specimens were tested; specimens with estimated probable fracture stresses (see Section 2.1.2) and preferable geometry were chosen.

The specimens were then prepared for testing by levelling of the surfaces, so that they would be perfectly flat with the two surfaces parallel to each other. The testing was performed with displacement control at a displacement rate of 0.005 mm/s; recording the resulting force, until nonlinearities appeared, at which point the specimen was unloaded. The set-up of the compression test is illustrated in Figure 2.7.
2.3.1 Stress-strain relationship determination

The compression test is in some ways easier to evaluate than the bending test, since stresses and strains can easily be determined when the geometry of the specimen is known. However, since the test is performed with a uniaxial loading, it limits the amount of information that can be extracted; in the linear region, only the elastic modulus can be found, the effect of Poisson’s ratio is not determined, since results are only recorded in the axial direction.

First of all, the machine compliance must be accounted for; when the test is performed the machine also deforms, in addition to the specimen deformation. This means that the recorded displacement is not what the specimen is actually experiencing. This can in the form of equations be expressed as

\[
\delta_{\text{total}} = \delta_{\text{machine}} + \delta_{\text{specimen}}. \tag{2.12}
\]

The machine deformation can be determined if the machine compliance is known:

\[
\delta_{\text{machine}} = CP, \tag{2.13}
\]

where \(C\) is the machine compliance and \(P\) is the recorded force. The actual deformation of the specimen can now be determined by combining Equations (2.12) and (2.13) as
\[ \delta_{\text{specimen}} = \delta_{\text{total}} - CP. \]  

(2.14)

The compliance was determined through performing a compression test with a very stiff specimen; the force-displacement curve was recorded and hence the machine compliance could be determined as the slope relating the two. Technically, the displacement of the stiff specimen should also be accounted for, but since this was a very stiff material, the contribution to the displacement would be very small and, in fact, negligible.

Now, the true displacement-force relationship is known, and stresses and strains can be determined. The natural strain in a compression test is calculated as follows:

\[ \varepsilon = \log\left(\frac{t}{t_0}\right), \]  

(2.15)

where \( t_0 \) is the initial thickness of the specimen and \( t \) is the deformed thickness, which can be determined by the use of the corrected displacement as

\[ t = t_0 + \delta_{\text{specimen}}, \]  

(2.16)

where \( \delta_{\text{specimen}} \) is continuously smaller than or equal to zero. Since the compression test subjects the specimen to loading in only one direction, the resulting stress state will also be unaxial. Hence, the resulting engineering stress is determined by use of the specimen’s initial cross-sectional area, \( A_0 \) and the resulting force, as

\[ \sigma = \frac{P}{A_0}. \]  

(2.17)

In order to determine this area, the specimens were photographed with a reference length scale and then processed using the algorithm described in Section 2.1.3; polygons with the real dimensions were created and the area could be determined using built-in functions in Matlab. This approach assumes that the cross section remains constant throughout the thickness, which is reasonable, given the thickness of the specimen (\( \lesssim 10 \) mm): changes over this thickness will be very small and can be assumed negligible.

In the linear region of the uniaxial loading test, the relationship between stress and strain is described by the elastic modulus as

\[ \sigma = E\varepsilon, \]  

(2.18)

which allows determination of the E-modulus, when both stress and strain are known. This was done by visually estimating a linear part of the graph,
to which a slope was fitted. Since the onset of the test does not exhibit linear behaviour (because of sliding of the specimen, contact creation between machine and specimen, etc), this slope’s intersection with the $x$-axis (strain-axis) was used to determine the point at which the compression of the specimen begun. The relationship between stress and strain in the linear region could then be described by this slope, i.e. the elastic modulus.
Chapter 3

Results

3.1 Bending test

3.1.1 Estimation of fracture stresses

The recorded force, calculated moment and estimated resulting stresses are presented in Figure 3.1. The estimated cross-sectional parameters used for the stress determination are presented in Table 3.1 along with the values for the absolute maximum and maximum elastic stresses determined.

Table 3.1: Estimated second area moment of inertia, $I_y$, highest distance from centre of mass $z_{max}$, resulting fracture stresses and elastic stresses (corresponding displacement to elastic stresses can be found in Appendix B.2). *Unreliable result, dimensions of test set-up corrupted. **Not calculated, as these bones were not simulated.

<table>
<thead>
<tr>
<th></th>
<th>$I_y$ [mm$^4$]</th>
<th>$z_{max}$ [mm]</th>
<th>$\sigma_{max}$ [MPa]</th>
<th>$\sigma_{el}^{max}$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bear1</td>
<td>8269</td>
<td>9.43</td>
<td>231.3</td>
<td>187.1</td>
</tr>
<tr>
<td>Bear1v</td>
<td>19401</td>
<td>11.73</td>
<td>85.5</td>
<td>47.7</td>
</tr>
<tr>
<td>Bear2</td>
<td>15787</td>
<td>11.20</td>
<td>42.1*</td>
<td>**</td>
</tr>
<tr>
<td>Bear3</td>
<td>7314</td>
<td>8.72</td>
<td>258.6</td>
<td>197.4</td>
</tr>
<tr>
<td>Bear4</td>
<td>11558</td>
<td>9.43</td>
<td>109.0</td>
<td>**</td>
</tr>
<tr>
<td>Bear5</td>
<td>17495</td>
<td>10.58</td>
<td>126.0</td>
<td>50.1</td>
</tr>
<tr>
<td>Bear6</td>
<td>14038</td>
<td>10.93</td>
<td>210.7</td>
<td>**</td>
</tr>
</tbody>
</table>
Figure 3.1: Measured force, calculated moment and estimated stresses versus applied displacement.

### 3.2 FE-simulation of bending test

Four bones were included in the simulation process: Bear1, Bear1v, Bear3 and Bear5. Bear2 was excluded because of corrupted set-up dimensions, Bear4 because of the odd appearance of the force-displacement curve (see Figure 3.1) and Bear6 because of missing CT-scanned images.

The resulting material parameters for the microplane model for the four simulated bones can be seen in Table 3.2 along with determined stresses. See Appendix B.1 for resulting curve fits and divergence from experimental results and Appendix B.2 for plots of stress and macroscopic damage at the investigated loads.
Table 3.2: Resulting material parameters from the bending test simulation, determined highest stress at maximum displacement and elastic stresses.

<table>
<thead>
<tr>
<th></th>
<th>$E$</th>
<th>$\nu$</th>
<th>$\gamma^\text{mic}_0$</th>
<th>$\beta^\text{mic}$</th>
<th>$\sigma_{\text{max}}$</th>
<th>$\sigma^\text{el}_{\text{max}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bear1</td>
<td>9.8</td>
<td>0.32</td>
<td>1.2</td>
<td>37</td>
<td>150.6</td>
<td>134.9</td>
</tr>
<tr>
<td>Bear1v</td>
<td>3.3</td>
<td>0.45</td>
<td>1.9</td>
<td>4.8</td>
<td>79.1</td>
<td>58.7</td>
</tr>
<tr>
<td>Bear3</td>
<td>9.3</td>
<td>0.38</td>
<td>1.04</td>
<td>29</td>
<td>122.6</td>
<td>109.7</td>
</tr>
<tr>
<td>Bear5</td>
<td>5.7</td>
<td>0.40</td>
<td>0.64</td>
<td>27</td>
<td>47.2</td>
<td>40.1</td>
</tr>
</tbody>
</table>

3.3 Compression tests

The evaluated results from the compression tests can be seen in Table 3.3. See Appendix B.3 for complete graphs acquired from the tests.

Table 3.3: Estimated elastic modulus from compression tests, along with determined geometrical parameters.

<table>
<thead>
<tr>
<th></th>
<th>$E$</th>
<th>$t_0$</th>
<th>$A_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bear1v</td>
<td>5.25</td>
<td>7.26</td>
<td>372.4</td>
</tr>
<tr>
<td>Bear2</td>
<td>9.64</td>
<td>9.79</td>
<td>462.7</td>
</tr>
<tr>
<td>Bear3</td>
<td>10.18</td>
<td>10.23</td>
<td>362.9</td>
</tr>
<tr>
<td>Bear6</td>
<td>11.31</td>
<td>9.29</td>
<td>458.8</td>
</tr>
</tbody>
</table>
Chapter 4

Discussion

Bone is a porous, inhomogeneous material and the material parameters generally vary with density and calcium content; in this project it has been approximated as a solid body with homogeneous properties. This might be one of the causes for diverging results between the bending and compression tests. In the former case, the whole bone is subjected to testing and the resulting properties can be regarded as a sort of average over the whole bone. In the compression test, only a small part of the bone is tested and the properties determined are localised representations. When comparing the results for Bear1v and Bear3 between the bending (Table 3.2) and compression (Table 3.3) tests, it seems as the elastic modulus is underestimated in the former case as compared to the latter, but since the specimen for compression were chosen from bone that had not been fractured during the bending test, it is reasonable to conclude that those parts were simply stiffer than the average of the whole bone, explaining this slight difference. However, the parameters are still in the same order of magnitude and fairly close between the two tests, showing sufficient agreement.

It is generally known that bones are anisotropic, which means that the Poisson’s ratio can be greater than 0.5. As mentioned in Section 2.2.2, this is not allowed in the chosen material model. However, sufficient agreement has been reached between the experiments and simulations, but many parameters influence the behaviour and form of the curve, raising the question: are the generated parameter combinations unique? It may be possible to reach the same load solution for a different set of parameters. In different loading scenarios the chosen combinations might not yield conformable results. To reach reliable conclusions, different loading scenarios should be tested against the results concluded here.
Further, when examining the determined material properties, as seen in Table 3.2, two obvious things are noticeable: i) the linear properties, $E$ and $\nu$ vary between the bones, with the range of $5.7$-$9.8$ GPa and $0.32$-$0.45$ respectively, ii) the damage parameters are also quite similar, with two exceptions: $\gamma_0^{\text{mic}}$ is greater than $1\%$ for all bones except Bear5 and the value of $\beta^{\text{mic}}$ is significantly lower for Bear1v than the other bones. This may have several causes; all this might be within the actual distribution but might appear singular here because of the small number of specimens tested; errors in testing and/or simulations; or as previously mentioned the proposed parameter combinations may not be unique. It may also simply be an effect of differences between the bears; such as age or other properties not considered in this project. In all of those cases, investigations of more specimens may clear up some of the question marks.

An important aspect of damage development in bone is that it happens differently in compression and tension, but as mentioned in Section 2.2.2 this phenomenon has in this project been disregarded. In the conducted compression tests, the nonlinear behaviour was apparent, but not further investigated. It is not certain if it is caused by plasticity or damage, but as stated previously literature supports the theory that the nonlinear behaviour is in fact damage development. The compression tests, however, suggest that plastic deformation is not negligible; examining the shape of the curves recorded during the compression tests (see Figure B.6 in Appendix B.3) the unloading curves exhibit behaviour characteristic for plasticity. Bones are in nature more likely to be loaded in compression than tension, and hence their development has been adapted to this, yielding lower damage for the same load magnitude in compression as opposed to tension. Because of this, the conclusions drawn from this work will be conservative, as the tensile loads occurring in subjection to bending have been considered as limiting. However, in other cases, it might be of interest to determine the bone behaviour in pure compression, e.g. when examining real-life situations and damage developing over a long period of time. For this purpose, further investigations need to be executed, adjusting the expression for the equivalent strain in the microplane model.

When comparing the stresses estimated from standard beam theory in Table 3.1 to those computed numerically with the damage model in Table 3.2, it is apparent that the analytic calculation overestimates the fracture stresses quite a lot; except for Bear1v, all stresses are overestimated by 50-170\%. Why one of the bones has a better agreement is hard to say, but since this is a singular incident, it is reasonable to assume that this is circumstantial. Looking at the elastic stresses, the estimation is slightly better. First of all, the same bone once again stands out; Bear1v is underestimated by about 18\%, while
all other stresses are overestimated by 25-80%. This confirms the former theory that something is not quite right with this one specimen, as it has quite diverging material parameters, as compared to the others. In conclusion, the standard beam theory can give some idea about the stresses in the elastic region of deformation, but the results should be treated with caution, as they will most likely be overestimating the actual stress state and hence not yield conservative results.

Overall, it is difficult to reach any reliable conclusions about the fracture stresses, as the variation between specimen is quite large, but it is clearly seen that stiffer bones also endure higher stresses. However, this tendency is not correct when comparing Bear5 and Bear1v; the former has a higher stiffness, but lower stress at maximum load. As Bear1v has previously been deemed unreliable, it is difficult to say if the resulting stress for Bear5 is reliable. A larger test set would give more reliable conclusions.
Chapter 5

Conclusions

- The elastic moduli determined from the verifying compression tests agree quite well with results from the FE-simulation of bending tests, confirming the legitimacy of the results. The conclusion is that the elastic modulus of bear bones is in the range of 3-10 GPa.

- With the chosen material model, Poisson’s ratio for bear bones falls within the range of 0.3-0.45.

- The damage parameters of the microplane model are concluded to be 1-2% for $\gamma_{0}^{mic}$ and 25-40 for $\beta^{mic}$.

- Standard beam theory can be used to get an idea of the stress state in the region of elastic deformation but should not be considered as more than very a rough and unconservative estimation, as the estimated value can be almost twice as big as the true value.

- The fracture stresses for bear bones seem to be highly related to the elastic modulus; bones with a stiffness of around 9-10 GPa can endure around 120-160 MPa, while bones with a modulus below 9 GPa likely have a fracture stress below 100 MPa.

- Further investigation is needed to determine the bones’ behaviour in compression as compared to the tensile behaviour, which has mainly been investigated in this thesis.

- The developed method could be used to determine difference in material properties between winter and summer bears.
Bibliography


Appendix A

Methods

A.1 Bending set-up

Figure A.1: Picture taken during test execution. Courtesy of Mathias Haarhaus, M.D., Ph.D., Associated at the Department of Clinical Science, Interventions and Technology at Karolinska Institutet.

Figure A.2: Example of the finished 3D-model set-up.
A.2 Compression specimens

Figure A.3: Compression specimens.
Appendix B

Results

B.1 FEM-fit of load curve

Bear1

$E=9.8$ GPa, $\nu=0.32, \gamma_0=0.012, \beta=37$

Bear1v

$E=3.3$ GPa, $\nu=0.45, \gamma_0=0.019, \beta=4.8$
Figure B.1: FEM-curves fitted to load-displacement curves recorded experimentally.
B.2 Stress and damage plots

(a) Maximum principal stress at maximum load.

(b) Macroscopic damage at maximum load.

(c) Maximum principal stress at elastic deformation, $\delta=2.8$ mm.

(d) Macroscopic damage at elastic deformation, $\delta=2.8$ mm.

Figure B.2: Maximum principal stress and macroscopic damage for Bear1.
APPENDIX B. RESULTS

(a) Maximum principal stress at maximum load.

(b) Macroscopic damage at maximum load.

(c) Maximum principal stress at elastic deformation, $\delta = 2.5$ mm.

(d) Macroscopic damage at elastic deformation, $\delta = 2.5$ mm.

Figure B.3: Maximum principal stress and macroscopic damage for Bear1v.
APPENDIX B. RESULTS

(c) Maximum principal stress at elastic deformation, $\delta = 1.6$ mm.

(d) Macroscopic damage at elastic deformation, $\delta = 1.6$ mm.

Figure B.4: Maximum principal stress and macroscopic damage for Bear3.

(a) Maximum principal stress at maximum load.

(b) Macroscopic damage at maximum load.

(c) Maximum principal stress at elastic deformation, $\delta = 1.3$ mm.

(d) Macroscopic damage at elastic deformation, $\delta = 1.3$ mm.

Figure B.5: Maximum principal stress and macroscopic damage for Bear5.
B.3 Compression test evaluation

(a)

(b)
Figure B.6: Results from compression test with step wise applied correction.