FINITE ELEMENT ANALYSIS OF PORCINE TRABECULAR BONE BASED ON MICRO-CT IMAGES

BY
OLI JIN

THESIS
Submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering in the Graduate College of the University of Illinois at Urbana-Champaign, 2015

Urbana, Illinois

Adviser:
Professor Iwona Jasiuk
ABSTRACT

The main goal of this study was to build a finite element model for trabecular bone from micro-computed tomography (micro-CT) images and study elastic properties of bone. We also studied the relationship between Young’s modulus and porosity of trabecular bone. Researchers in the past have shown that porosity and apparent density were the two main factors that affect Young’s modulus of trabecular bone. However, experiments also showed that trabecular bone samples with similar porosity may have very different Young’s modulus. In this study, trabecular bone samples from porcine femoral head of 6 months old animals were scanned by Xradia micro-CT and finite element models were built and meshed by pre-FEA software ScanIP. Then, Abaqus software was used to conduct uniaxial compression test simulations and calculate Young’s modulus of the samples. Two different boundary condition settings were used during the simulations. In one of the settings, displacement was applied directly on the top surface of the bone model (Method 1), while in the other one, displacement was applied through a rigid plate parallel to the top surface (Method 2).

We compared the results from both computational methods with the experimental data we had on these samples and found that our FEA models generally accurately captured elastic properties of trabecular bone when using boundary condition setting 1. Method 2, however, gave results much stiffer than experimental data. We observed that porosity indeed is a major factor that contributes to Young’s modulus of trabecular bone. Models with higher porosity tended to have lower elastic modulus as described by former researchers. We also studied models with similar porosities that had a large range of elastic modulus values. Therefore, the technique of building FEA model from micro-CT images is suitable for porous structure such as trabecular bone. The models build
in this study will be used in the future study to investigate mechanical properties of trabecular bone beyond the yield point and with account for time dependent properties.
I want to thank my advisor Professor Iwona Jasiuk, who patiently guided me in my research and generously helped and supported me with her professional advices when I run into challenges. I would like to thank Woowon Lee, a former student in my research group, who provided all the experimental data I used for comparison in my computational study. I also want to express my appreciation to Jason Patrick, a PhD student at UIUC, who discussed ideas with me and introduced me to a very powerful software ScanIP. Finally I want to thank all my friends who have always been there to support me.
# TABLE OF CONTENTS

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

CHAPTER 2 METHODS ..................................................................................................................... 3

2-1 Sample material and Micro-CT images ....................................................................................... 3

2-2 Create FEA model from micro-CT images ................................................................................... 3

2-3 Finite element analysis with Abaqus ....................................................................................... 6

2-3-1 Apply displacement directly on bone model ......................................................................... 7

2-3-2 Apply displacement on rigid plate ....................................................................................... 9

CHAPTER 3 RESULTS AND DISCUSSIONS ................................................................................. 13

CHAPTER 4 FUTURE WORK ............................................................................................................ 19

REFERENCES .................................................................................................................................. 20
CHAPTER 1: INTRODUCTION

Bone is a connective tissue consisting of two parts: cortical bone and trabecular bone. Cortical bone is the compact structure at the outer layer, while trabecular bone is a spongy-like structure filling the interior spaces and the ends of long bones. In this study, we focus on a trabecular bone. The trabecular bone has a porous structure, with porosity ranging from 30% to 90% (mostly in the range 75%-95%) [1]. Such porous structure gives the trabecular bone excellent features as a structural biological material: minimal weight and comparatively high strength for loading. Porosity plays an important role in determining mechanical properties. Degradation in mechanical properties of trabecular bone is an essential feature of diseases such as osteoporosis. Thus, the study of the mechanical properties of trabecular bone is important in orthopedics to assess bone quality and to diagnose bone disease. However, the analysis of trabecular bone has been challenging due to the complexity of its structure and variability among individuals. Also, a compression test of trabecular bone is a complex boundary value problem both experimentally and computationally. A wide range of values of the overall elastic modulus have been reported in literature - from several hundreds of mega Pascals to over 1 GPa [2-4]. Earlier studies showed that porosity and apparent density are the two main characteristics that have a big influence on the elastic modulus of trabecular bone [5-8]. However, experimental data also shows that bone samples with similar porosity can have distinct elastic modulus and strength [9]. Motivation of this study was to investigate a suitable computational approach to compute elastic modulus of trabecular bone and to gain a better insight of how stress is transmitted within the trabecular struts. This will lead to more accurate predictions and better understanding of mechanical properties of trabecular bone.
Due to the structural complexity of trabecular bone, experimental methods for testing mechanical properties pose challenges because of number of factors that can have an influence on the result. Alternatively, a micro-finite element (FE) analysis based on high-resolution images makes it possible to build FE model of trabecular bone in detail. In this study, a finite element analysis technique was used to model and simulate a compression test of trabecular bone and calculate its elastic modulus. In this thesis, micro-computed tomography (micro-CT) images from 6-month porcine trabecular bone from femoral head were used to build the FE model. Porcine trabecular bone was selected because of its similarity to human trabecular bone. Other imaging techniques such as three-dimensional Magnetic Resonance Imaging can also be used to create the models [10-11]. In this study, micro-CT imaging is chosen because that it is non-destructive, provides good contrast of bone, and requires minimum sample preparation [12].

In summary, the objective of this study was to build finite element models of trabecular bone samples from micro-computed tomography images to guide future studies and investigate the elastic properties of the bone samples. This study complements other computational studies in this area.
CHAPTER 2 METHODS

2-1 Sample material and Micro-CT images

Samples tested for this study were obtained from femurs from six-month old pigs from the Meat Science Lab of University of Illinois at Urbana-Champaign. Porcine and human trabecular bone share similar structures, therefore conclusions gained from this study may also be applied to human trabecular bone. Seven samples were picked for the computational simulations. Each of the samples was made from femoral head and was cut in the shape of cylinder.

All samples were scanned by an Xradia micro-CT. After scanning, all files were converted into tif files which were compatible with FEA pre-processing software Simpleware.

2-2 Create FEA model from micro-CT images

In this study, both geometry and finite element models were generated with FEA pre-processing software ScanIP (Version 6.0, Simpleware Ltd).

The procedures of creating geometry and FEA model of the trabecular bone samples from micro-CT images were as follows:

1. Import stack of images. Select the folder of one sample which contains over 1000 tif images converted from micro-CT images. After selecting the folder, look for a file named ‘Thumbs.db’ and remove it from the stack. This file is automatically generated during importing and will cause errors if is not removed. Leave other settings as default.
2. Crop and realign the images. After scanning, the images have remaining margins and sometimes, are not perfectly aligned along z axis. In the ‘Imaging processing’ tag, the ‘Align’ tool can rotate the images along x, y and z axis separately. To save computer memory and minimize the size of the file, ‘Crop’ tool is used to get rid of the margins of the images.

3. Resample. Seven samples in total had been analyzed in this study and each sample contained over 1000 slices of images after scanning. Because of the size of the data and in order to avoid generating a model containing too many elements, all samples has been down sampled before segmentation. Despite a complex geometry, down sampling the data should not appreciably affect the final model geometry, as the high resolution of the original image mean features are still described by many voxels. ‘3.0’ has been used as the new pixel spacing in all three directions, x, y and z.

4. Select value for threshold. Different choice of the value of threshold can make a large difference in the geometry model. Simpleware will only consider the image part that meets the threshold density as the bone material. The rest of the image will be considered as void or air. The higher the value of threshold is, the less of image will be treated as bone, which will lead to a higher porosity of the model. Open the threshold tool and check ‘Enable’ interactive thresholding. In the threshold window, activate the ‘Profile line’ tool. Then draw a line across one of the 2D slice views and obtain a histogram of the grayscale intensity along the line. Upper limits for all samples were 225 in this study. Lower limits were in the range of 115 to 128. After selecting threshold, create new mask for the sample. The bone part of the images will turn red.
5. Smooth the model. Once the initial mask is created, use ‘shrink wrap’ tool to reduce the blank image boundaries and cut down the memory requirements for the computer. ‘5 pixels’ is used for all models in this study. Some voxels which do not belong to the bone sample may have been captured when selecting the threshold. In order to isolate just the bone sample, ‘floodfill’ tool is used to get rid of the voxels mentioned above. Next, perform a ‘cavity fill’ operation to fill up any spurious voids that may have been generated within the bone material. “Smoothing’ can smooth any rough surfaces that may remain in the segmentation, which will make it easier for the meshing algorithm to generate good quality elements. Make sure ‘binarise’ is active and apply smoothing of 1.5 pixels strength in all three directions. In the end, apply another ‘floodfill’ to make certain that any small, disconnected voxel islands have been removed.

6. Measure geometry dimensions. Measure porosity, height and diameter of all the models. Open the ‘measurement’ tag and click ‘volume fraction’. The value gave the volume fraction of the bone. Therefore, porosity equals to that value subtracted by 100%. To measure height and diameter of the sample, use the ‘distance’ tool. Porosity and geometry measurement of each sample after selecting threshold value is shown in Table 1.

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Threshold</strong></td>
<td>128</td>
<td>125</td>
<td>125</td>
<td>123</td>
<td>120</td>
<td>120</td>
<td>123</td>
</tr>
<tr>
<td><strong>Porosity</strong></td>
<td>79.2%</td>
<td>90.24%</td>
<td>81.7%</td>
<td>71.1%</td>
<td>84.4%</td>
<td>89.4%</td>
<td>86.9%</td>
</tr>
<tr>
<td><strong>Height/mm</strong></td>
<td>630</td>
<td>732</td>
<td>758</td>
<td>786</td>
<td>849</td>
<td>747</td>
<td>840</td>
</tr>
<tr>
<td><strong>Diameter/mm</strong></td>
<td>398.2</td>
<td>413.8</td>
<td>430</td>
<td>441</td>
<td>432</td>
<td>475</td>
<td>446</td>
</tr>
<tr>
<td><strong>Cross section area/mm²</strong></td>
<td>124410</td>
<td>134484</td>
<td>145220</td>
<td>152745</td>
<td>146574</td>
<td>177205</td>
<td>156431</td>
</tr>
</tbody>
</table>

**Table 1** Porosity, threshold and geometry measurement of each sample
7. Create mesh. The geometry model has already been generated after step 6. Next step is to mesh the model and convert it into an FEA model. Right click on ‘models’ in the dataset browser, select ‘Create new FE model’. Drag the mask previously created into this new model. Right click on the model and select ‘Model configuration…’ to setup model. Active the ‘Smart mask smoothing’ option. Set the export file type to Abaqus input file. In the volume meshing tab, change the algorithm to ‘FE free’. In this setting, more positive number means finer meshing and more negative means the opposite. Set the ‘Compound coarseness’ slider to -50. In the 'FE model' tag, click ‘Full model’ and start generating mesh. I was using Pollock at Visualization lab at Beckman Institute, with 16 cores and 128 GB memory. It took 30-45 minutes to finish meshing for each model. Take sample 7 as an example, the total number of elements is 905816 and the total number of nodes is 279662. Models were exported as Abaqus input files.

2-3 Finite element analysis with Abaqus

In this study, material property and boundary condition settings were done in FEA software Abaqus 6.13-2. After meshing in Simpleware, the FEA model was imported to Abaqus. Material was considered to be linear elastic. According to prior research, the elastic modulus of pig trabecular bone tissue is in the range of 10 GPa to 18 GPa [13-16]. In this study, all elements were given linear elastic, isotropic material properties with Young’s modulus of 10 GPa and Poisson’s ratio of 0.3 for all seven studied samples.

There is more than one way to conduct a numerical simulation of a compression test with FEA software. Multiple approaches have been considered during this study. Details of these methods are discussed in the following sections.
2-3-1 Apply displacement directly on bone model

In this method, no extra part is needed. The only part is the trabecular bone sample meshed by Simpleware.

1. Set boundary conditions. In the Initial step, fix the bottom of the model in all six degrees of freedom. In order to do so, select a thin layer of element at the bottom of the model which is perpendicular to the z axis. In the ‘Edit boundary condition’ window, select ‘Encaster’, it will set displacement in three directions (U1, U2, U3) and rotation along three axes (UR1, UR2, UR3) to zero.

2. Apply loading. Create a new step “disp load” and create a new boundary condition “top”. Select a thin layer of element at the top of the model and in the ‘Edit boundary condition’ window, check U3 and give it a value of 15. The unit of displacement here is 0.01 millimeter. In this way, a total displacement of 0.15 mm along z axis is added to the top of the bone model.

3. Select elements of the middle part. During compression, stress concentration will occur near the bottom and the top of the model. To make sure that the results will not be affected by that, only results of elements of the middle part will be collected. Therefore, we need to create a set for the middle part elements. In ‘Tools’ tag, click ‘Sets’ and then create sets. In the pop up window, chose type to be ‘Element’ and then select the middle part of the model, approximately 60% of the total volume. Name the new set ‘middle element’.
4. Step setting. Under the created step ‘disp load’, edit field output. To save memory and simulation time, select only the created set ‘middle element’ for domain and only calculate result for the last increment. The output variables calculated will be EVOL (element volume), S (stress components and invariants) and U (translations and rotations). For history outputs, change frequency to ‘last increment’.

5. Create job and start simulation.

6. Export result data from odb file. Since the volume of each element is different in the model, the stress result needs to be averaged. In odb file, open ‘Report’ tag and then click ‘XY…’. Export both stress value S and element volume EVOL data to a txt file then average the result in Matlab R2013a with the equation:

\[
\bar{S} = \frac{\sum S_i V_i}{\sum V}.
\]

7. Calculate elastic modulus of the model.

\[
E = \frac{Stress}{Strain} = \frac{\bar{S}}{\frac{d'}{L}},
\]

where \(d'\) is the total displacement and \(L\) is the length before deformation.
Fig 2. Simulation result of sample 4

2-3-2 Apply displacement on rigid plate

This method completely simulates the compression test experiment. During the test, bone sample was placed in the middle of the bottom plate of the testing system (MTS systems Corp., Eden Prairie, MN), and the top plate was adjusted above the sample without any preload applied. Then, simplified uniaxial compression test was conducted.

In this method, instead of directly applying displacement on the top surface of the model, a rigid plate was created as the top plate from the testing system.
1. Create rigid plate. Build it as a 3D analytical rigid extruded shell, approximate size was 800 (unit: 0.01mm).

2. Assembly. Since there is more than one part, assemble is necessary in this method. Create instance first. Instance should include both the meshed bone part and the rigid plate. Instance type is ‘Independent (mesh on instance)’. Since it is known that the top plate is parallel to the top surface of the bone sample, use the ‘Create constraint: parallel faces’ tool: select the rigid plate as the movable instance and then select one element face on the top surface of the bone part as the fixed instance, click OK. Next, move the rigid plate to the top of the bone part and adjust until the bone part top surface is fully covered by it.

3. Initial step settings. Same as step 1 in chapter 2-3-1.

4. Apply loading. Create a step ‘load’. Define displacement/rotation boundary condition ‘top’ on the reference point of the rigid plate, set the displacement in z axis U3 equals to the compression displacement. Set U1=U2=UR1=UR2=UR3=0 to avoid unwanted rigid body displacement. Select CF (concentrated forces and moments), RF (reaction forces and moment), S (stress components and invariants) and U (translations and rotations) as field output. Set the frequency as ‘Last increment’ to save memory of the computer.

5. Apply interaction settings. Different from method one, the two parts in this model will be in contact with each other. Therefore, an interaction setting is necessary. In ‘interaction properties’, set the tangential behavior as frictionless and set the normal behavior as ‘Hard contact’. Then create ‘surface to surface contact’ interaction, pick the lower side of the rigid body as a master surface since it is stiffer. Select the faces of elements which will contact with the rigid plate as the slave surface. After create interaction, the interface should look like Figure 3.
6. Apply step settings. In the ‘Incrementation’ tag, all settings are shown in Figure 4.

![Fig.3 Interactions setting](image)

![Fig.4 load step settings](image)

7. Create job and start simulation.
8. In the odb file, collect reaction force data for the final increment. Calculate elastic modulus with the equation:

\[ E = \frac{FL}{dA} \]

where \( F \) is the reaction force at the rigid plate, \( L \) is the height of the sample, \( d \) is the total displacement, \( A \) is the cross section area of the sample.

*Fig 5.* Simulation result of method 2 for sample 4
CHAPTER 3 RESULTS AND DISCUSSIONS

Elastic moduli gained by both numerical methods and experimental tests are summarized in Table 2 and Figure 6.

Table 2 Comparison of different methods (unit: MPa)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>140.16</td>
<td>105.86</td>
<td>473.57</td>
<td>358.57</td>
<td>943.84</td>
<td>338.82</td>
<td>637.51</td>
</tr>
<tr>
<td>Method 1</td>
<td>1713.14</td>
<td>460.97</td>
<td>1243.25</td>
<td>2033.37</td>
<td>1816.38</td>
<td>914.37</td>
<td>2661.96</td>
</tr>
<tr>
<td>Method 2</td>
<td>857</td>
<td>285</td>
<td>1326</td>
<td>1582</td>
<td>916</td>
<td>387</td>
<td>428</td>
</tr>
</tbody>
</table>

Fig 6. Comparison of different methods

It can be seen from the table and the figure above that the elastic modulus of numerical solutions for each sample is significantly higher than the ones from experimental tests. The average value of the results from method 1 is 1549.06 MPa and 825.86 MPa for method 2, while the average for the experimental test result is 428.33 MPa. The average of the difference between
experimental data and the average of the two numerical data is 274.82%. If we compare the two methods of numerical analysis, the results of method 1 are much stiffer than the ones of method 2. The average difference between the two methods is 723.21 MPa.

When studying the relationship between the elastic modulus of models and porosity, the overall trend in method 1 and method 2 is that models become stiffer with smaller porosity. However, in both method 1 and experimental results, models with similar porosity have very different value of elastic modulus around 65%.

Fig 7. Porosity vs. elastic modulus for computational methods and experimental data

One possible reason for this phenomenon is that there is stress concentration occurring at thin truss within the model. When picking a threshold value to transfer micro-CT images into geometry model, there will be some extremely thin truss structures in the model which are impossible to exist in real bone samples, as shown in Figure 8. Within the red circle is the thin truss mentioned above, it can be seen that obvious bending behavior has occurred within this
structure during compression. The von Mises stress at these locations is due to stress caused by both uniaxial compression and bending. Therefore, the local elastic modulus at these trusses is sometimes twenty times higher than the average value.

![Fig 8. Zoom in view of Max Von Mises stress location](image)

Another possible reason is the choice of the element type we used to mesh the model. Linear tetrahedron element was used in both methods for all seven samples. Modeling of trabecular bone has been conducted many times in previous studies. Different types of elements including both linear and quadratic hexahedral meshes [2, 15, 17-19] and tetrahedron meshes [2] were used. According to researches, hexahedral meshes have better performance than tetrahedral meshes with better accuracy, smaller number of elements and increased reliability. Research has shown that linear tetrahedron elements are much too stiff for most cases, including bending,
calculating stiffness and stress concentrations [20]. The quadratic tetrahedral elements are better if the element size is fine enough. Quadratic hexahedral elements are very accurate but computationally expensive. The stiffer elastic modulus obtained from both computational methods in this study may be a result of the properties of linear tetrahedral elements. The main reason linear tetrahedron mesh was chosen was to save memory and simulation time. With this choice of element, the meshing and simulation time was 40 minutes and three hours, respectively.

**Fig 9.** Cross section view of sample 1
For the big difference between the numerical and experimental results, another possible explanation is that there might be some flaws in the experiment.

1. In the compression tests, samples are placed between two parallel places, this method has been used by most researchers. However, some research showed that in this way, there will be singularity point and therefore cause stress concentration which will lead to stiffer results [21]. Instead, the equipment is glued to the lateral surface of the sample while not in touch with the top and bottom surfaces.

2. During preparation, the samples might not be perfect cylinder and top and bottom surfaces are sometimes not parallel.

3. Trabecular bone is an extreme porous structure. However, it does not have to be symmetric. One side of the sample might be more porous than the other and therefore less stiff than the other side. In this situation, there will be bending moment exist in the sample during compression.

4. Choice of threshold was not accurate.

Comparing the experimental and finite element analysis methods, both methods have their advantages and limitations. For finite element analysis method, multiple outputs can be extracted as needed. And after simulation is done, it is easy to have a cut view of the model and learn better about how forces and stresses are transported within the porous structure. However, the technique for FEA nowadays cannot guarantee accuracy of the result. To gain better result, models need to be meshed with fine quadratic meshes which are very time consuming on current computers. For experimental tests, very good results can be obtained. However, the problem with experiments is that people can make mistakes during experiment and it will largely affect the accuracy of the results.
When comparing the result from method 1 and method 2, it can be seen that the result from method 2 is much closer to the experimental result. The main difference between method 1 and method 2 is that in method 1 displacement is applied directly on the top of the bone model while in method 2 it’s applied on a rigid plate on top of the bone model. Method 2 is exactly simulating the experimental compression test. That’s another possible reason that method 2 gives better results.
CHAPTER 4 FUTURE WORK

This study has several limitations. First of all, the samples used in this study are porcine femurs instead of human bone. Although the macrostructure and microstructure of porcine bones are very similar to human bones [22], the composition of trabecular bone is different. Therefore, the result may not be compatible to human bone. In addition, only trabecular bone from femoral head was studied. Other anatomical locations may give different mechanical properties. To further conduct this study, samples from various types of human bones should be prepared.

Second, the element type and size in this study has been proved to be not very accurate. If we have access to better computational resources, finer quadratic hexahedral or tetrahedron elements should be used to mesh the model instead of linear tetrahedral elements.

Also, this study was focused on the linear region of trabecular bone material property and, plastic and viscoelastic properties have not been applied to the model. Future studies should include more mechanical properties, such as yield stress, fracture toughness and viscoelastic properties.
REFERENCES