IMPLEMENTATION OF BONE GRAFT ADAPTATION'S FE MODEL IN HYPERMESH

Martin O. Dóczi
Péter T. Zwierczyk
Department of Machine and Product Design
Budapest University of Technology and Economics
Műegyetem rkp. 3., Budapest 1111, Hungary
doczi.martin@gt3.bme.hu
z.peter@gt3.bme.hu

Róbert Sződy

Péterfy Hospital
National Institute of Traumatology
Fiumei street 17., Budapest 1081, Hungary
robert.szody@gmail.com

KEYWORDS

Acetabular bone defect, Acetabular cage, Bone graft adaptation, Finite element analysis, HyperMesh

ABSTRACT

Research significance: In the clinical practice, surgeons sometimes must deal with extended bone defects. Among others, bone grafts are used for filling the large absence.
After implantation, the structure of the graft can change, and the graft's load-bearing effect can be significant. This leads to the idea, that during the design of an implant this effect should be taken into account in the finite element simulations.
In this paper, the authors show the implementation of the bone graft adaptation.
Methodology: This programming task was done by using Python, Tcl and the HyperMesh interface. The bone remodeling algorithm and the related parameters were from the literature research. The results are shown with a finite element model prepared for the Optistruct solver, where the geometry models were based on a patient's CT data.
Results: Viewing the bone graft's elemental apparent density, the most loaded areas could be detected.
Conclusion: The model can predict qualitatively the bone graft's change, which can provide additional information for the implant design. Further analyses are required to investigate the sensitivity of the results.

INTRODUCTION

Clinical Overview

Total hip replacement is an effective way in the treating of osteoarthritis. However, after 10-20 years, a revision surgery has to be made, where the damaged prosthesis elements have to be changed.
In some cases the problem is not with the prosthesis, but the patient's bone. Due to some kind of infection or the stress-shielding effect, significant bone degradation can be observed (Figure 1). (Paprosky et al. 1994)

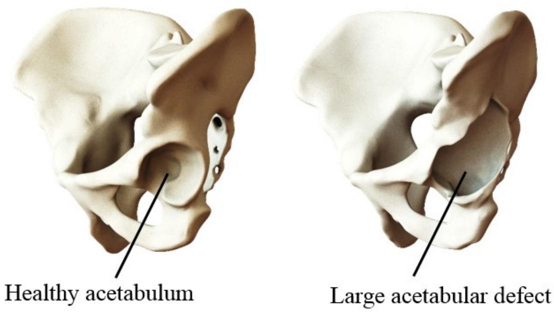

Figure 1. Comparison of healthy acetabulum and large acetabulum defect (Dóczi et al. 2020)

Another example is the tumorous mandible resections, where a large part of the mandible have to be replaced. (Chi Wu et al. 2020)
When the surgeons have to deal with these kind of large bone defects, it is a plausible way using void fillers, which can turn later into living bone tissue. (Sződy et al. 2018), (Ahmad and Schwarzkopf 2015)
It is evident, that in these situations the fixation system can not be so rigid that leads to bone degradation again. However, when the implant is not so stiff, it is usually a weaker construction as well, which can not withstand large forces. This is a trade-off problem.
There is another aspect, which should be considered. If the flexible implant can induce positive bone graft adaptation, it means, that the graft load-bearing effect can be significant later, the overall system can withstand the external forces more easily.
The authors had a different publication (Dóczi et al. 2020) where a simple cantilever beam model and an open-source solver were used to show this algorithm's qualitatively correct behavior. However, in this problem, more complex models required commercial software, which led to further improvements and changes in the process. These will be discussed in this paper.

Communications of the ECMS, Volume 35, Issue 1,
Proceedings, ©ECMS Khalid Al-Begain, Mauro Iacono,
Lelio Campanile, Andrzej Bargiela (Editors)
ISBN: 978-3-937436-72-2 / 78-3-937436-73-9(CD) ISSN 2522-2414

Bone Graft Remodeling

Bones and bone grafts can adapt to the loading environment.
There are models, which can describe this phenomenon. One of these claims, that the bone adaptation is related to the strain energy density (SED).
If the given volume part's SED value is divided by the part's density, a so-called stimulus can be obtained. (Chi Wu et al. 2020), (Sue 2016)
The bone's growth response (the density increment for the next step) for this stimulus can be separated into multiple zones, as shown in Figure 2.

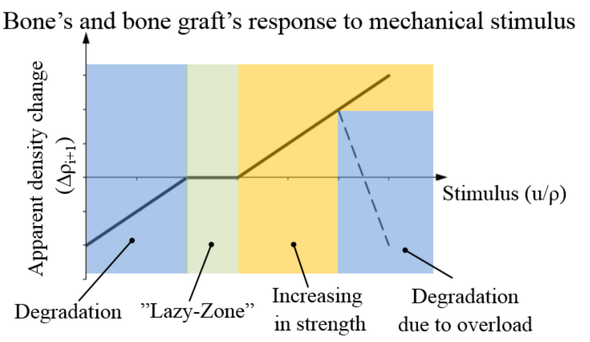

Figure 2. The Bone Graft Remodeling Model

If the mechanical stimulus is low, the bone's apparent density will be decreased, this is the bone degradation. This can happen for large overloads as well. There is a so-called "lazy-zone," a stimulus range, where changes can not be observed in the bone's apparent density. If the stimulus is higher than that, the bone's apparent density is increasing, which means the bone grafts become stiff.
This response is only a model for a pseudo load, which represents a recurring load for a given time period (week, month etc.). The slope of the function has effect of this (pseudo) time. If the slope is too high, this can lead to poor results, and if the slope is too low, the number of the required simulations to show the trends are increasing.
The elastic modulus is the relevant material property for the FE calculations. This can be calculated from an equation by the literature research using the density as shown in Equation (1). E is the elastic modulus, ρ is the density, *b* and *c* are constants from the literature. (Helgason et al. 2008)

$$E = b \cdot \rho^c \tag{1}$$

DATA AND METHOD

Software Environments

Altair HyperMesh is a powerful software for the preprocessing of FE models. Using Tcl, the user can write useful macros for automating tasks. Due to multiple user interfaces, models can be built even for Abaqus, ANSYS etc., solvers.
One of the in-built solvers is the Optistruct, which can be used for FE analysis and optimization as well. The FE input file can be modified as a text file, which can lead to profound customization possibilities. The input files can be separated into different parts, which is helpful during the overwriting because it is not necessary to read and rewrite the entire file.
FE results can be exported as text files as well.
Python is the most popular open-source programing language. Due to the communal improvements, many modules are available. For example, NumPy can be used for manipulating large multidimensional arrays, which is ideal for rewriting FE data.
For the post-processing of the results, pyNastran was utilized.

Implementation of the Graft Remodeling Algorithm

A flowchart can be seen in Figure 3 where the overall implementation is presented.

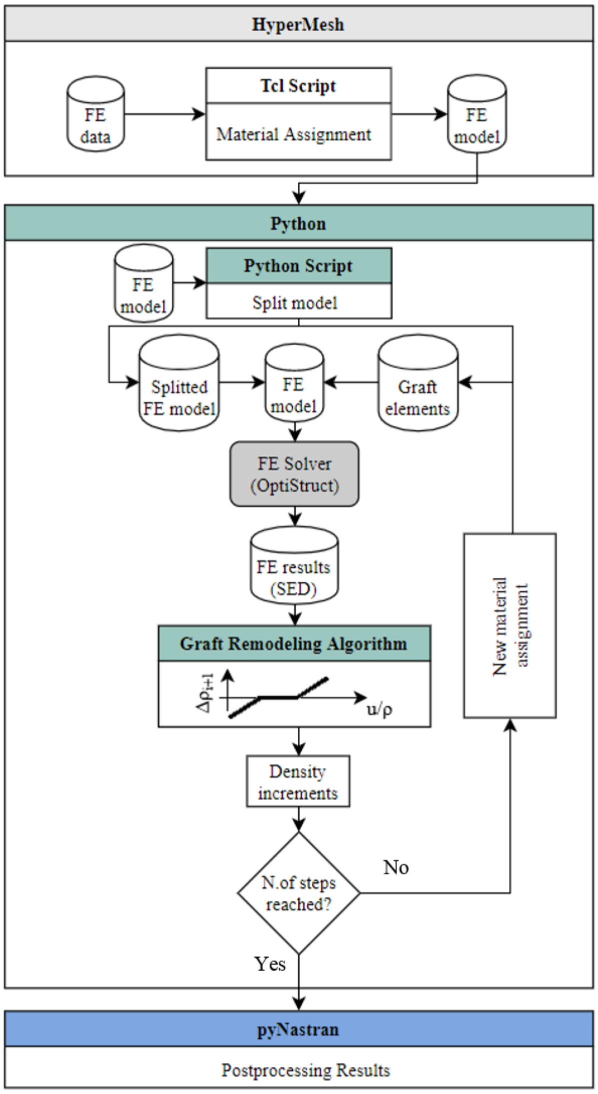

Figure 3. The Flowchart of the Implementation

The workflow is the following. The FE model should be prepared in HyperMesh as usual. A set for the elements of the graft with ID 1 must be created.
For the material assignment, the graft's potential elastic modulus range should be discretized to a large number of ranges; in other words a lot of material data and properties have to enter. Obviously, this should be done by a Tcl script.
After the material assignment, the input file must be saved.
The structure of the saved OptiStruct input file is the following. First, the coordinates of the nodes can be seen. After that, the sets and the elements are presented.
The property ID of an element, which consist the element's material data as well is written after the ID of the element. This number have to be changed during the remodeling process.
In order to make fast changes in the input file, it should be split into parts. The graft data can be separated in another text file with a python script due to the aforementioned set definition.
The required FE results are the strain energy densities of the graft elements. These can be exported to a Nastran ".pch" file, which is easily readable.
The graft remodeling script can calculate the stimulus array from the FE result file and the growth increments of the density for every graft element.
In the next step, the separated input file of the graft elements is rewritten so the elements have new density and elastic modulus value. The input file is solvable again to the pre-defined calculation steps.
The end-results are the graft elements and their density-elastic modulus distribution. For the effective visualization of these plots, a freely available program, pyNastran is used.

Finite Element Model

In this paper, the implementation of the algorithm is the main focus, and the authors investigate the benefits and the disadvantages of the discussed modeling process.
In order to get detailed observations, a pelvis model with a large acetabular defect was used. The geometry model was from a patient's CT data. After the segmentation and the CAD work, a hemipelvis model was generated.
The surgeon prescribed the center of the acetabular cage. (Sződy et al. 2018) The graft's geometry model was mainly in the direction of the maximum amplitude force vector from the gait cycle. This is the most common loading of the implant and the graft. (Bergmann et al. 2001)
The geometry models can be seen in Figure 4.
The FE preprocessing was done in the HyperMesh preprocessing software. HyperMesh. The hemipelvis and the graft were meshed with 10 node tetrahedral elements. Near the acetabular defect, a homogenous bone model was used. In the healthy areas of the pelvis, the material model was separated into a spongiosus and cortical parts. The cortical parts were represented as 6-node triangular shell elements with a 1 mm thickness. (Plessers and Mau 2016)

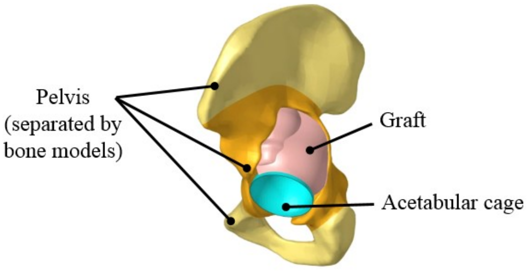

Figure 4. The Parts of the Geometry Model

The acetabular cage has steel properties and it is modeled using 6-node triangular shell elements as well, but with 1.5 mm thickness.
Information about the FE mesh can be seen in Table 1.

Table 1: The Data of the FE Mesh

Number of nodes	210 296
Number of elements	151 139
Number of 10 node tetra elements	140 470
Number of 6 node trias	10669

All of the materials had homogenous, linear elastic, and isotropic properties.
The material properties can be seen in Table 2. The bone's material properties are from the literature research. (Anderson et al. 2005), (Ravera et al. 2016)

Table 2: Elastic Material Properties

	Young's modulus [MPa]	Poisson's ratio [-]
Steel (AISI 316L)	192000	0.3
Cortical bone	17000	0.3
Trabecular bone	100	0.3
Homogenous bone	7000	0.3

In order to investigate the trends, a simplified model was used with bonded connections everywhere.
The modeled acetabular cage has no flange, so it can not connect to the pelvis. After the initial graft density definition, the load was a prescribed displacement by the authors' choice (-0.1 mm; 0.1mm; 0.5mm in the X, Y, Z directions, respectively), at the center of the acetabular cage, transferred with rigid bars. In this simulation, the reaction forces were calculated, the graft's initial density was the minimum density, 382 kg/m^3. The resultant reaction force will be used as a pseudo load, for the graft remodeling calculations. The authors think this approach can be used for eliminating the effect of other irrelevant contacts and the graft's changes can be separately viewed, because from the perspective of the bone graft, just the connecting parts are important.

Fix boundary conditions were used at the sacroiliac joint and the pubic symphysis, according to the literature research. (Plessers and Mau 2016)
The resultant force vector's components and the overall FE model with the boundary conditions can be seen in Figure 5.

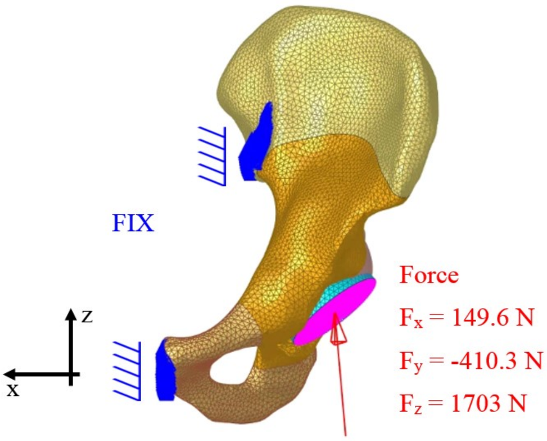

Figure 5. The Finite Element Model

Parameters of the Graft Remodeling Algorithm

The parameters used for the graft remodeling algorithm can be seen in Table 3. The presented graft remodeling parameters only have demonstration goals. Further investigations required to define these numbers.

Table 3: Parameters for the graft remodeling algorithm

	Value	Dimension
Density coefficient (*b*)	1,8	m^2/s^2
Density exponent (*c*)	3	-
"Lazy zone" lower	0,05	m^2/s^2
"Lazy zone" upper	0,1	m^2/s^2
Min density	382	kg/m^3
Max density	2322	kg/m^3
Slope	10	$kg \cdot s^2/m^5$

The bone graft resorption due to possible overload was not examined.
Another value was defined, which name was apparent mass. It is the summarized value of graft element's volumes (V_i) multiplied by their densities (ρ_i). It represents the evolving new bone structure quantitatively, and further comparisons can be made with it. The equation can be seen in Equation (2), where '*i*' is the index of a graft element.

$$m_{app} = \Sigma \rho_i \cdot V_i \quad (2)$$

Different simulations were made to investigate the effect of the graft's initial density, which means different initial elastic modulus as well. The graft's initial densities were 382 kg/m^3; 500 kg/m^3, and 618 kg/m^3, respectively.
The number of calculation steps was set to 20.

RESULTS

The new density distribution of the graft can be seen in Figure 6. In this model, the graft's initial density was 618 kg/m^3.

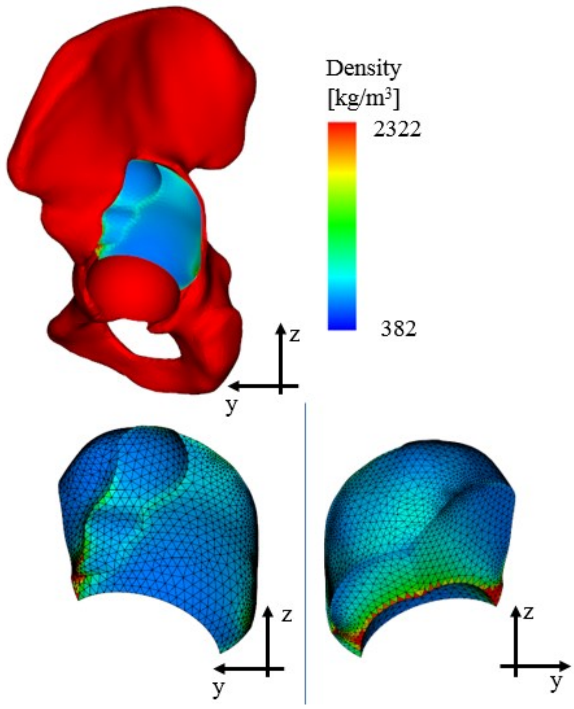

Figure 6. The Density Distribution of the Graft

Using Equation (2), and the apparent mass approach, the different models can be compared. In Figure 7, it can be seen the changing of the apparent mass during calculation steps, with different initial graft's densities.

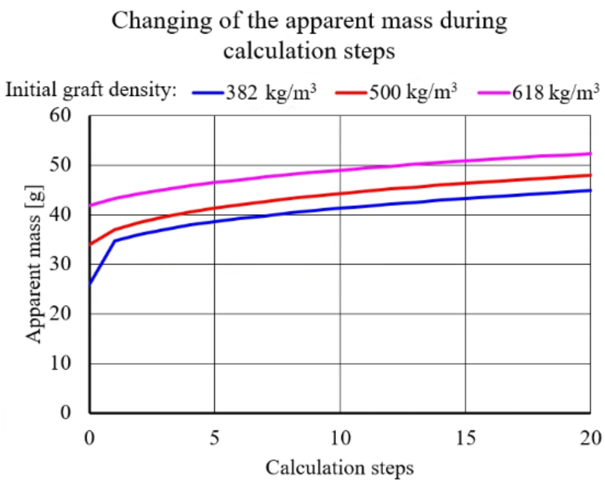

Figure 7. Changing of the Apparent Mass During Calculation Steps

DISCUSSION

The implementation of the graft remodeling algorithm into the HyperMesh-Optistruct interface was successful. The results in Figure 6 shows that the most loaded areas of the graft, where the elements have the largest density value. These are the areas where shear strain occurs. Further investigations are required to analyze this phenomenon; the authors want to implement the overload-degradation effect into the model, which possible eliminates these results. The other important aspect is the other loaded area on the back of the graft, in the force vector's direction, which is a qualitatively correct result. With the use of second-order elements, the checkerboard problem is not revealed. (Bendsoe 2003), (Rahman et al. 2013)

The results in Figure 7 shows that bigger initial density means a bigger apparent mass. At first glance, it is evident, but it can be seen that the difference between the apparent masses becomes a constant value as the number of the calculation steps is increasing. This information can be used for further implant development because it suggests that the same implant design can produce the same apparent mass growth after a given time, regardless of the graft's initial density and elastic modulus in the investigated range.

Further sensitivity analyses have to be done. Including the effect of the slope of the algorithm, the boundary values of the "lazy-zone", and the effect of the overload and the X,Y,Z values of the prescribed displacement.

The authors want a validation for the model, so actual X-ray images or CT data will be investigated.

It is obvious, that there are many parameters that have effects on the results. However, implant development's main task is to find the trends, which have a significant impact on the design.

ACKNOWLEDGMENT

The research reported in this paper and carried out at BME has been supported by the NRDI Fund (TKP2020 NC, Grant No. BME-NCS) based on the charter of bolster issued by the NRDI Office under the auspices of the Ministry for Innovation and Technology.

REFERENCES

Ahmad, A. and Schwarzkopf, R. 2015. "Clinical evaluation and surgical options in acetabular reconstruction: A literature review." *Journal of Orthopaedics* 12 (2): S238-S243

Anderson, A. et al. 2005. "Subject-Specific Finite Element Model of the Pelvis: Development, Validation and Sensitivity Studies." *Journal of Biomechanical Engineering* 27 (3): 364-373

Bendsoe, M. 2003. "Aspects of topology optimization and bone remodeling schemes." http://biopt.ippt.gov.pl/Minipapers/Bendsoe.pdf 2020.10.15. 15:56

Bergmann, G. et al. 2001. "Hip contact forces and gait patterns from routine activities." *Journal of Biomechanics* 34 (7): 859-891Chi Wu et al. 2020. "Time-dependent topology optimization of bone plates considering bone remodeling."

Chi Wu et al. 2020. "Time-dependent topology optimization of bone plates considering bone remodeling." *Computer Methods in Applied Mechanics and Engineering*. 359: 112702

Dóczi, M., Sződy, R., Zwierczyk, P. 2020. "Finite element modeling of the changing of bone grafts using HyperMesh-Calculix interface." *GÉP* LXXIV: 15-18

Helgason, B. et al. (2008): "Mathematical relationships between bone density and mechanical properties: A literature review". *Clinical Biomechanics* 23 (2): 135-146

Paprosky, W., Perona, P. and Lawrence, J.1994. "Acetabular defect classification and surgical reconstruction in revision arthroplasty: A 6-year follow-up evaluation." *The Journal of Arthroplasty* 9 (1): 33-44

Plessers, K. and Mau, H. 2016. "Stress Analysis of a Burch-Schneider Cage in an Acetabular Bone Defect: A Case Study." Reconstructive review. 6 (1): 37-42

Rahman, K. et al. 2013. "Structural topology optimization method based on bone remodeling." *Applied Mechanics and Materials* 432-426: 1813-1818

Ravera, E. et al. 2015. "Combined finite element and musculoskeletal models for analysis of pelvis throughout the gait cycle." *Conference: 1st Pan-American Congress on Computational Mechanics and XI Argentine Congress on Computational Mechanics*

Sue, A. 2016. Bone remodeling. http://web.aeromech.usyd.edu.au/AMME5981/Course_Documents/files/Lecture%208%20-%20Bone%20Remodelling.pdf 2021.03.17. 12:10

Sződy, R. et al. 2017. (in hungarian) “Csípőprotézis revízióikor alkalmazott „custom made” vápakosár tervezése és készítése, három esetben alkalmazott eljárás.” In *7. Hungarian Conference of Biomechanics* (Szeged, HU, okt 6-7) *Biomechanica Hungarica* 10(2): 20

AUTHOR BIOGRAPHIES

MARTIN O. DÓCZI is a Ph.D. student at the Budapest University of Technology and Economics Department of Machine and Product Design, where he studied mechanical engineering and obtained his degree in 2019. His research area is numerical biomechanics and implant development. His e-mail address is: doczi.martin@gt3.bme.hu and his web-page can be found at http://www.gt3.bme.hu.

RÓBERT SZŐDY is a orthopeadic and traumatology physician. He got his degree at the Semmelweis University in 1995. He made a traumatology professional examination in 2000 and an orthopeadics professional examination in 2005. He works as a surgeon at Péterfy Hospital and Manninger Jenő National Institute of Traumatology. His e-mail address is: robert.szody@gmail.com.

PÉTER T. ZWIERCZYK is an assistant professor at Budapest University of Technology and Economics Department of Machine and Product Design where he received his M.Sc. degree and then completed his Ph.D. in mechanical engineering. His main research field is the railway wheel-rail connection. He is a member of the finite element modelling (FEM) research group. His e-mail address is: z.peter@gt3.bme.hu and his web-page can be found at: http://gt3.bme.hu