Patient Specific Finite Element Modeling of Shoulder Implant Based on CT scan: The Concept of Modelling Explained

Kocsis György (✉ kocsisgyorgy@hotmail.com)

Semmelweis University

Gabor Henap

Budapest University of Technology and Economics

Research Article

Keywords: computer tomography (CT), Strain energy density (SED)

Posted Date: August 12th, 2021

DOI: <https://doi.org/10.21203/rs.3.rs-798715/v1>

License: © ⓘ This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Abstract

The following work introduces a method to carry out patient specific implant design based on computer tomography (CT) imaging technique. Point cloud bone model and the solid model creation process is described in detail. Using the radiodensity level of the CT-scan the continuous distribution of the elastic properties of the bone is also been recorded and used for the material model of the finite element procedure. For this purpose, a continuous mass density – elastic modulus curve is suggested, based on previous results in the literature. Stress shielding poses a serious issue regarding the survival of an implant. Strain energy density (SED) is a good indicator of the effects that drive the remodeling. Based on local SED difference caused by the implant this phenomenon can be quantified and visualized. This makes it possible to classify or redesign the implant in order to minimize the potential bone loss caused by the altered stress state.

Introduction

In silico models are becoming increasingly popular in orthopaedic endoprosthesis design. Advantages include speed, lack of hassle regarding ethics approval, lack of problems regarding tissue handling, relative ease of access, reproducibility, etc. Computer modeling allows for changing and controlling parameters easily. Creating accurate models is the essence of this method.

As many in silico models get published, it is our aim to explain the process of computer modeling to the orthopaedic public. This knowledge comes useful when critically appraising journal papers or communicating with engineers.

For the creation of a digital bone model, the most obvious source is a CT scan. The CT scan images consist of pixels of different radiodensity(1). The 3 dimensional pixels, or voxels, that represent bone need to be selected, preserving the bone density data as well. The point cloud is then converted into a solid body model by marking up the surfaces using a mesh of triangles. This model can then be fed into 3D printer, but if we want to do finite element analysis, some further steps are needed. The model needs to be transformed into a format that is suitable for further processing in a CAD system (3D solid bodies). Then, the geometrically realistic bone model can be used to perform a numerical analysis in order to assess its behavior under operating conditions, even with an implant added. The solid body model acquires physical material properties at this stage. Unlike most published models, the bone model presented here preserves the inhomogeneous nature of bone in a feasible way from the computational point of view.

The stress state of a body is complex, multiple stress components (normal and shear in three orthogonal planes) are present(2). In other words, stress has hydrostatic and deviatoric components. The majority of the finite element models in the orthopaedic context express results as Von Mises stress(3, 4). Von Mises stress is a useful scalar value. It combines the different stress components into one single “equivalent stress”, which represents the distortional energy density. Von Mises stress is widely applied for

assessment of ductile materials, such as metals for example, where the failure is connected to the deviatoric part of the stress only. This is an empirical fact that makes von Mises stress suitable for evaluating parts made from common engineering materials. There is no reason to hold this limitation in the case of bone (5, 6), especially if we want to characterize the stress shielding phenomenon.

Stress shielding occurs when an implant “removes” the typical stress from a section of bone and unloads it. Bone responds to mechanical stimuli. Cyclic stress induces bone production, lack of it tends to cause bone resorption. One widely accepted mechanism is that osteocytes have mechanoreceptors, which are stimulated by interstitial fluid flow or direct mechanical stimuli. In other words loading of the bone causes its microscopic segments to change shape (strain), creating the mechanical stimulus of the osteocytes(7). In this paper we are proposing the use of strain energy density to assess stress shielding, instead of Von Mises stress. Strain energy density functions represent the distribution of total stored elastic energy (both hydrostatic and deviatoric parts), as opposed to von Mises stress which represents the deviatoric part only.

We also discuss a proposed function between radiological bone density and bone elastic modulus.

Materials And Methods

Finite element analysis

The analysis was based on an anonymised randomly selected anatomically normal CT-scan (Philips Ingenuity Core 64). The resolution of the images was 768x768, the pixel spacing and slice spacing were 0.4948mm and 0.75mm, respectively. 3D solid body models were built using Wolfram Mathematica (Wolfram Research, Hanborough, Oxfordshire, United Kingdom) and Ansys (Ansys Inc. Canonsburg, Pennsylvania, U.S. A.) and were modified with custom made implant models (own design). The finite element model can be seen in Fig. 1.

For the spatial discretization 237611 linear tetrahedron elements were used with a node number of 52106. Tetrahedrons are suitable to discretize complicated geometric features. The contact surfaces are locally refined in order to get better accuracy. The material properties of the different components are summarized in Table 1. The bone material properties were set as discussed in the material property mapping section, with a maximal assumed bone density $\rho_{bone} = 1.8g/cm^3$.

Table 1
Component names and material properties

Component name	Material	Young-modulus E [GPa]	Poisson ratio ν [-]
Humerus metaphyseal component	Titanium Ti-6Al-4V	96	0.36
Glenoid liner	HDPE	1.1	0.42
Glenoid tray	Titanium	96	0.36

Between the intended contact surfaces a frictional contact behavior was defined with a friction coefficient $\mu = 0.1$. The load was defined as follows. A coordinate system was attached to the guiding pin of the glenoid liner so that the x-axis is parallel to the axis of the pin (see Fig. 2). Displacement was applied to the distal humerus section relative to the proximal scapula section in the direction of the pin. Imposing prescribed displacement makes the analysis more stable in the presence of contact, especially when the model is constrained by a frictional contact surface only, as is the case here. In this analysis the required compression force is 750 N that was transferred between the humerus and the scapula (8). The prescribed displacement on the distal humerus was determined such that the reaction force on the constraint must be 750 N at the end of the analysis. This means that with force driven analysis exactly this force would be necessary in the direction of the displacement in order to cause the same displacement and stress field. The scapula was rigidly fixed as it is demonstrated in Fig. 2. Due to the highly nonlinear nature of the contact problem an incremental analysis was necessary and the effect of large deflection (i.e. geometrical nonlinearity) was also taken into account. Displacement was applied in 20 increments.

Results

There are many ways to express the results of the numerical analysis. The displacement field is the primary field variable of the displacement based FEA. From the displacement field several derived quantities can be evaluated, such as contact conditions of the touching surfaces. The displacement field of the models can be seen in Fig. 3. Please note that the magnitude of displacement is in tenth of millimeters or below. The distribution of the von Mises equivalent stress on the different components is shown in Figs. 4. Figure 5. presents the contact pressure distribution on the surface of the implants.

Strain energy density can be computed by Equation [19] using the principal stresses and strains. The distribution of this energy density alone is not as interesting as how an implant decreases strain energy density following implantation compared to native bone. For this reason stress shielding is expressed as:

$$u_{diff} = 100 \cdot \frac{u_{intact} - u_{implanted}}{u_{intact}}. \#(1)$$

where u stands for strain energy density.

For comparison an implant-bone model and a native bone model are necessary. In our case the native bone model was created using the implant-bone model, by giving bone properties to the plastic and metallic parts. The bone density data from the original CT scan were used for this. This step was necessary to keep both congruency and bone density mapping accuracy for both models, allowing for precise control of the loading conditions. The density mapping over the implant-bone and the native bone model can be seen in Fig. 6.

u_{diff} mathematically can be a positive or a negative value. Since energy can only have a positive value, the negative difference refers to an extra load that is not relevant in stress shielding. Therefore, only the positive values of u_{diff} were be considered. In Fig. 7. the distribution of the quantity (1) can be seen so that the colored domains cover those parts only where the strain energy difference is greater than zero. This refers to the extent and the possible intensity of the stress shielding phenomenon.

Discussion

Patient specific geometry

For a CT scan gamma radiation is used to map the relevant body part. The scanner records multi axial density data of tissues as small pixels. During the scan, each scanned pixel gets an address in a Cartesian frame that can be extracted with the pixel spacing information at hand. Along with marking the spatial position of the pixel, the density of the given pixel is also recorded. Although most scanner units are able to create output files in picture formats, it is more practical to choose the raw DICOM output data for further use.

The next step is to get this data out of the CT scanner output file into a three dimensional point cloud. The individual pixels' density data can be characterized by a numeric value between 0 and 1. As CT scans record density for all tissues, it is also necessary to decide which density values represent bone. As bone is significantly more radiopaque than soft tissues, this task is relatively easy, but may require human input. If the cutoff value is selected correctly, the "noise" from the soft tissues is minimal, but minimal number of actual bone pixels are lost by not being selected. This step is more difficult if the cortical bone is not dense due to osteoporosis, as is the case for the scapula of an elderly patient, for example. Another problematic area is the cartilage covered articular surfaces. In the latter cases bone density values on the surface of the bone fall closer to those of the surrounding soft tissues. For the sake of simplicity, any pixels denser than the determined cutoff density value will be considered bone.

A typical image from a typical CT scanner is 768x768 pixels per slice, but can be a lot more (9). This is a lot of data, and it is practical to get rid of all the unnecessary points and only keep the pixels of each CT slice that are relevant to us. The relevant area of each CT slice can be marked, creating a field or region of interest (ROI). Human input may come handy in this step.

Once the data package from the CT output file is read, the density cutoff value for bone is selected and the appropriate ROIs are marked, the computer is able to create a three dimensional point cloud that represents bone.

In this study Wolfram Mathematica software tool was used for these steps, because it allows good control of each step, bone density values can be preserved, and the output file can be customized.

An algorithm was created that stores the point cloud representation of the bone as a matrix for each slice. The amount of data can further be reduced using a sparse array storage technique – this means that within each ROI, only the points that represent bone will be stored.

Before this point cloud is converted into a virtual three dimensional bone, more data is needed: pixel spacing and slice spacing. This step determines how far the points of the point cloud need to be from each other when building a size correct virtual model. Slice- and pixel spacing is stored amongst the metadata of the DICOM file. The 3D model is built following a mathematical algorithm by creating a mesh of triangles.

STL is a file format created by 3D Systems(10). One backronym for STL is “Standard Triangle Language”. This format is widely used for 3D printing, rapid prototyping and computer-aided manufacturing. In this format the surface of a solid body is represented by a mesh of triangles and their normal vectors. STL file can be created in the Mathematica environment.

The finishing step is the conversion of the STL model into a solid object model. This can be done using the SpaceClaim module of Ansys finite element software package. The reverse engineering tool is capable of covering the surface mesh of the object with NURBS-patches (Non-Uniform Rational B-Spline, mathematical representations of 3D geometry that can accurately describe any shape from a simple 2D line, circle, arc, or curve to the most complex 3D organic free-form surface or solid). With this conversion a solid body model is created, generally referred to as a CAD model.

The most challenging part of the presented method is the conversion of a point cloud model to a standard CAD solid geometry. This is not automated yet in commercial applications. The CAD model of the bone is necessary so that the CAD model of an orthopaedic implant could be “implanted” into the bone model. Creating an STL model is a necessary in-between step.

When the NURBS-patches are applied, the medullary cavity (which is present in the STL model) is typically lost from the model. This is because applying the NURBS patches is like wrapping the outer surface of the model in cling film – anything within the wrapping becomes a solid body and the medullary cavity is lost. The traditional method to recreate the medullary cavity is to build a solid body (CAD) model of the medullary cavity itself and then subtract it from the bone solid body model. This method is very impractical. This is why an alternative method is proposed here, in the material property mapping section.

For finite element analysis the solid body model implant is merged into the bone solid body model, using Ansys. Then a finite element mesh is created, the quality of which is of paramount importance. The finite element mesh breaks up the solid geometry into small domains that are connected by the nodes. Mathematical equations are solved over these elements, and a solution field is approximated with specific interpolation functions. The mesh should be fine enough to capture all of the relevant geometric features but the number of the applicable nodes is limited by the computational capabilities of the device.

Material properties are also used as a parameter for finite element analysis. From an orthopaedic surgeon's perspective, it must be noted that good software for finite element analysis requires a massive investment to develop, and the development follows the needs of the potential market. For this reason, there is no designated method in Ansys to tell the computer that bone is a non-homogenous material and that different density values from the CT scan represent different material properties. For this reason many publications use a homogeneous model to describe bone physical properties, accepting a potentially large error(11–13). However, there is no inherent obstacle to utilize continuously distributed material properties as it demonstrated below. The workflow of the whole analysis process can be seen in Fig. 8.

Material Property Mapping

The radiodensity of the CT-image pixel allows to create a spatial map of bone density that is proportional to the stiffness of it. For this it has to be assumed that the pixel radiodensity is linearly proportional of the mass density and the mass density is proportional to the Young modulus of elasticity. Nobakhti et al. reviewed the available models (5). Several functions have been published to describe the mass density-Young modulus functions but these are very different from each other. They are continuous but only piecewise smooth based on measurement result possibly relating to yet undescribed properties of the bone segments tested. Based on these previously published models a smooth function can be created through the entire density domain that is more convenient in the numerical modelling point of view. Based on the models of Nobakhti et al. the

$$E^{KH}(\rho) = \frac{1}{a + e^{b + c\rho}} \#(2)$$

function is proposed to establish the connection between the bone density and the elastic modulus.

The parameters of the function are determined as follows. The radiodensity values and the mass density of the CT-image are scaled between 0 and 1. The radiodensity of the minimal threshold value for bone is assumed to be 1% of the maximal density. Between 1 and 100%, at least one additional point needs to be selected. This can be determined with the averaged values that can be found in Fig. 9. In practice, with the maximal estimated bone density and the associated Young modulus at hand, the parameters of the E^{KH} curve can be determined. For a maximal bone density $\rho_{max} = 1.8g/cm^3$ and a maximal Young

modulus $E_{max} = 20 \text{ GPa}$ the function has the shape shown in Fig. 9. The additional point are determined as the averaged values of the presented functions at density values 0.5 and 1. Thus, the parameters of Equation [2] are: $a = -2.62$, $b = 1.245$, $c = 0.019$.

The value of the computed Young modulus can then be assigned to the spatial “points” of our bone solid model, and be directly used for numerical structural analysis as input parameter. This makes the automation of the mapping procedure of the Young modulus to the model more convenient, but creating a “continuous” model has other advantages when it comes to modelling the medullary canal.

The medullary cavity can be built as a separate model which then can be subtracted from the bone model. This makes the geometry of the model complicated and makes the finite element meshing more problematic and computationally expensive. Presence of cancellous bone complicates things further. Instead of modelling the cavities directly, using the above mentioned material property mapping allows us to use a solid bone model without cavities. The function in Fig. 9 eliminates the stiffness automatically in the domains where a cavity is present (i.e. the radiodensity level - therefore, the density is small). This makes the mathematical interpretation of the cancellous domains a lot more straightforward, especially in the context of patient specific implant design.

Strain Energy Density (Sed)

In the following chapter a brief introduction about the interpretation of the stress field and SED is given. In general, stress is a tensor quantity with 6 independent components (for non-polar materials). In an arbitrary Cartesian system, the matrix of the stress tensor can be express as

$$[\sigma] = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix}. \#(3)$$

Working with stress tensors can be cumbersome, therefore simplifications are used wherever possible. The von Mises equivalent stress – frequently used in finite element orthopaedic publications) is a scalar invariant quantity (does not depend on the applied coordinate system) that characterizes the stress state of the given point of a body. The stress state of a given body can be additively decomposed into two parts, the hydrostatic and the deviatoric components. Von Mises stress is very useful in engineering for metal structures, because in metals the hydrostatic component plays no empirically important role in failure, and therefore it can be omitted. Von Mises stress is only related to the deviatoric part of the strain energy, and is considered to be an accurate predictor of metal failure above a material dependent limit value (Mises yield surface). Von Mises stress for this reason is only accurate in ductile materials, and bone is not one of those. The applicability of von Mises stress for stress shielding for this reason is also

debatable. Strain energy density is more appropriate. The elastic strain energy density can be computed as the double contraction of the stress- and strain tensors,

$$u = \frac{1}{2} \sigma : \epsilon = \frac{1}{2} \sigma_{ij} \epsilon_{ij}. \#(4)$$

For the sake of simplicity, the latter can be expressed with principal values,

$$[\sigma] = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix}, [\epsilon] = \begin{bmatrix} \epsilon_1 & 0 & 0 \\ 0 & \epsilon_2 & 0 \\ 0 & 0 & \epsilon_3 \end{bmatrix}. \#(5)$$

Thus, the strain energy density has the form

$$u = \frac{1}{2} (\sigma_1 \epsilon_1 + \sigma_2 \epsilon_2 + \sigma_3 \epsilon_3) \#(7)$$

The stress tensor can be decomposed into the sum of a volumetric and a deviatoric part. The invariant notation of these two components are

$$\sigma_{vol} = \frac{1}{3} tr \sigma \delta, \#(8)$$

$$\sigma_{dev} = \sigma - \frac{1}{3} tr \sigma \delta. \#(9)$$

In matrix form

$$[\sigma_{vol}] = \begin{bmatrix} \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) & 0 & 0 \\ 0 & \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) & 0 \\ 0 & 0 & \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) \end{bmatrix}, \#(10)$$

$$[\sigma_{dev}] = \begin{bmatrix} \frac{1}{3}(2\sigma_1 - \sigma_2 - \sigma_3) & 0 & 0 \\ 0 & \frac{1}{3}(-\sigma_1 + 2\sigma_2 - \sigma_3) & 0 \\ 0 & 0 & \frac{1}{3}(-\sigma_1 - \sigma_2 + 2\sigma_3) \end{bmatrix}. \#(11)$$

The material law of the isotropic small strain elasticity (Hooke's law) is

$$\epsilon = \frac{1+\nu}{E} \left[\sigma - \frac{\nu}{1+\nu} \text{tr}\sigma \delta \right], \#(12)$$

thus, the three nonzero strain components are

$$\epsilon_1 = \frac{1}{E}(\sigma_1 - \nu(\sigma_2 + \sigma_3)), \#(13)$$

$$\epsilon_2 = \frac{1}{E}(\sigma_2 - \nu(\sigma_1 + \sigma_3)), \#(14)$$

$$\epsilon_3 = \frac{1}{E}(\sigma_3 - \nu(\sigma_1 + \sigma_2)). \#(15)$$

Substituting back, the volumetric part of the strain energy density has the form

$$u_{vol} = \frac{1}{2} \sigma_{vol} : \epsilon = \frac{(1-2\nu)(\text{tr}\sigma)^2}{6E}, \#(16)$$

and the deviatoric component will be

$$u_{dev} = \frac{1}{2} \sigma_{dev} : \epsilon = \frac{1+\nu}{6E} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]. \#(17)$$

The so called von Mises stress (Mises equivalent stress) is defined as

$$\sigma_{eqv} = \sqrt{\frac{3}{2} \sigma_d : \sigma_d} = \sqrt{\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]}. \#(18)$$

Using the equation above, the total elastic strain energy density can be expressed as

$$u_{tot} = u_{vol} + u_{dev} = \frac{(1-2\nu)(\text{tr}\sigma)^2}{6E} + \frac{1+\nu}{3E} \sigma_{eqv}^2. \#(19)$$

In short, the Mises equivalent stress is only present in the deviatoric part and in the case of bone it is not reasonable to omit the hydrostatic term. SED is also used in the context of bone remodeling(14).

In simple terms our proposed method expresses stress shielding as a percentage of the original strain energy density before inserting an implant. No clear cutoff values have been established for notable bone loss in the context of prosthetic stress shielding so far. This can be a promising field of future research.

Conclusion

In this paper a method is presented relating to a possible workflow of the design and analysis of patient specific bone implant. The survival of the implant can be increased if the device is perfectly fitted to the individual parameters of the patient. In order to achieved this a reliable method is needed to affordably develop such a device before the surgery. This work presents the workflow of this process from the CT-scan of the patient to the strength analysis of the customized implant. It involves the extraction of the distribution of material properties that is used in the FEA analysis in order to produce the most realistic environment for the implant. For the description of stress shielding we propose a method is that is capable to quantify this effect that makes it possible to consider this effect in the design process of the implant. The methods are demonstrated in a shoulder implant but the applicability is not limited to this area.

Declarations

Compliance with Ethical Standards:

Funding: This study did not receive any funding.

Conflict of Interest: The authors of this article, Gyorgy Kocsis and Gabor Henap, declare no conflict of interest.

Ethical approval: This article does not contain any studies with human participants or animals performed by any of the authors.

Author contributions:

GK: Writing the paper, providing orthopaedic background

GH: Model building and finite element analysis, providing mechanics background

References

1. Goldman, L. W. Principles of CT and CT Technology. *Journal of Nuclear Medicine Technology*, **35**, 115–128 (2007).

2. Gross, D., Bonet, J., Hauger, W. & Schröder, J. W. A. Wall. (Springer Berlin Heidelberg: Imprint: Springer, Berlin, Heidelberg 2018), pp. 1 online resource (XIII, 308 pages 159 illustrations, 156 illustrations in color).
3. Christensen, R. M. A Comprehensive Theory of Yielding and Failure for Isotropic Materials. *Journal of Engineering Materials and Technology*, **129**, 173–181 (2007).
4. Mises, R. Mechanik der festen Körper im plastisch-deformablen Zustand. *Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen, Mathematisch-Physikalische Klasse*, **1913**, 582–592 (1913).
5. Nobakhti, S. & Shefelbine, S. J. On the Relation of Bone Mineral Density and the Elastic Modulus in Healthy and Pathologic Bone. *Curr Osteoporos Rep*, **16**, 404–410 (2018).
6. Tuncer, M., Hansen, U. N. & Amis, A. A. Prediction of structural failure of tibial bone models under physiological loads: effect of CT density-modulus relationships. *Med Eng Phys* 36, 991–997; discussion 991(2014).
7. Rochefort, G. Y. & Benhamou, C. L. Osteocytes are not only mechanoreceptive cells. *Int J Numer Method Biomed Eng*, **29**, 1082–1088 (2013).
8. Bergmann, G. *et al.* In vivo glenohumeral contact forces–measurements in the first patient 7 months postoperatively. *J Biomech*, **40**, 2139–2149 (2007).
9. Hata, A. *et al.* Effect of Matrix Size on the Image Quality of Ultra-high-resolution CT of the Lung: Comparison of 512 × 512, 1024 × 1024, and 2048 × 2048. *Acad Radiol*, **25**, 869–876 (2018).
10. Szilvsi-Nagy, M. & Mátyási, G. Analysis of STL files. *Mathematical and Computer Modelling*, **38**, 945–960 (2003).
11. Patel, R. J., Wright, T. M. & Gao, Y. Load transfer after cemented total shoulder arthroplasty. *J Shoulder Elbow Surg*, **23**, 1553–1562 (2014).
12. Denard, P. J., Lederman, E., Parsons, B. O. & Romeo, A. A. Finite element analysis of glenoid-sided lateralization in reverse shoulder arthroplasty. *J Orthop Res*, **35**, 1548–1555 (2017).
13. Geraldès, D. M., Hansen, U. & Amis, A. A. Parametric analysis of glenoid implant design and fixation type. *J Orthop Res*, **35**, 775–784 (2017).
14. Peyroteo, M. M. A., Belinha, J., Natal, R. M. & Jorge Load adaptation through bone remodeling: a mechanobiological model coupled with the finite element method. *Biomech Model Mechanobiol*, (2021).

Figures

Figure 1

Finite element mesh

Figure 2

Boundary conditions

Figure 3

Displacement field

Figure 4

Distribution of the Von Mises equivalent stress

Figure 5

Contact pressure distribution

Figure 6

Density mapping for the native- and the implant-bone

Figure 7

Strain energy density difference distribution

Figure 8

Patient specific analysis workflow

Figure 9

Young modulus as the function of the mass density.