Creation of Patient Specific Finite Element Models of Bone-Prosthesis. Simulation of the Effect of Future Implants

J.J. Ródenas*, L. Giovannelli, E. Nadal, J.M. Navarro, M. Tur
Centro de Investigación en Tecnología de Vehículos
Universitat Politècnica de València, Spain
* *Corresponding author. E-mail: jjrodena@mcm.upv.es*

ABSTRACT: In this paper a computational technique to perform patient specific Finite Element analyses of the mechanical behaviour of future prosthetic devices is presented. The model is obtained from medical images and geometrical data; the former provide the information about the living tissues and the latter represent the prosthesis to be simulated. The strong point of this method is the simplicity in the creation of the FE model using a hierarchical structure of Cartesian grids and a non-conforming geometry independent analysis mesh.

1 INTRODUCTION

The mechanical behaviour of loaded bones changes in the presence of prosthetic devices which alter the stress distribution. This phenomenon is of primary importance in processes which affect the implant durability such as bone resorption and interface deterioration over the time. Intense research work has been done into the evaluation of the stress field in bone-prosthesis systems and in many cases the Finite Element Method (FEM) has been used for this purpose, Taylor et al. (1994).

The FEM is a computer based numerical technique for evaluating approximate solutions of problems commonly found in engineering environments whose behaviour can be expressed in terms of differential equations. It is the most commonly used method for numerical simulation in structural engineering, is a reliable tool for stress analysis and, hence, for supporting the geometry optimization of prosthetic devices during the design process, provided that good quality models are used.

The first step in Finite Element (FE) analysis is the generation of an appropriate mesh consisting in the subdivision of the geometry under consideration into the so-called elements. These are subdomains of simple geometry for which an approximate behaviour can be easily formulated.

Later, the equations used to characterize each of the elements of the mesh (or grid) are assembled into a system of equations that describes the overall behaviour of the problem under analysis. It must be noted that in the standard implementation of the FEM the geometry of the mesh conforms to the geometry of the domain to be analysed.

In this paper a method to perform patient specific stress analyses is proposed by means of a particular implementation of the FE method which makes use of a so-called Cartesian Grid consisting of a mesh of quadrilateral elements (cubes in the 3D case) that does not necessarily conform the geometry. Special techniques are used to take into account this lack of conformity between the FE mesh and the geometry. This leads to an implementation of the FEM whose effectiveness has been proved Nadal et al. (2013) in problems where an exact representation of the geometry is available through, for example, a CAD model. This implementation of the FEM is called *CG-FEM* which stands for Cartesian Grid FEM.

The work with 2D images, presented in this paper, represents a preliminary phase of the application of the method to MRI and CT scans and thus to more realistic 3D analyses. In this work the 2D geometry of a prosthesis has been superimposed on an x-ray image. Then the whole has been meshed and a stress analysis has been performed. The results are then shown and discussed.

The usual approach to this kind of problem is to use geometrical models both of the prosthesis and of the biological substratum it has to be implanted into. As opposite to our approach, in general, the geometrical model from the image is created by using segmentation tools and by defining geometrical boundaries. This is often the most time consuming step of the whole simulation process.

CG-FEM simplifies the creation of the model of both the prosthesis and the living tissue. In the case of the prosthesis, since the mesh is geometry independent, the quality of the element shape is guaranteed and the meshing process is very easy.

Concerning the biological part of the model, *CG-FEM* easily provides FE models from medical images in a direct way. With our technique no geometry is created from the medical image, hence segmentation is not required. Instead, the pixels are immersed in a Cartesian grid structure, as described in Giovannelli et al. (2013).

With *CG-FEM*, the independence between mesh and geometry is used to mix two different models into a single FE model: a FE model taken from a digital image with a FE model taken from a CAD model of an object like a prosthetic device. This model can therefore be used to predict the behavior of the system composed of bone and prosthetic device and could be used to select the best prosthetic device for each specific patient or even to redesign and optimize the geometry of the prosthesis according to the particular characteristics of the patient.

2 METHODOLOGY

Creating the FE model means, in the first place, calculating the global stiffness **K** matrix which describes the elastic behaviour of the system. Afterwards the boundary conditions will be imposed and the problem will be solved. The stiffness matrix **K** is obtained by assembling the stiffness matrices $\mathbf{k^e}$ of the elements of the mesh. Equation (1) shows the numerical integration necessary to calculate the matrix $\mathbf{k^e}$.

$$
\begin{aligned}
\mathbf{k^e} &= \int_{A^e} \mathbf{B^T D B} dA = \\
&= \sum_{i=1}^{IP} \mathbf{B}^T(\xi_i,\eta_i)\mathbf{D}(\xi_i,\eta_i))\mathbf{B}(\xi_i,\eta_i) \mid \mathbf{J}(\xi_i,\eta_i) \mid w_i
\end{aligned} \tag{1}
$$

where IP represents the number of integration points, (ξ_i, η_i) their positions in the element local coordinates, $\mid \mathbf{J}(\xi_i,\eta_i) \mid$ the corresponding Jacobian determinant, w_i the weights associated to the integration points depending on the quadrature rule chosen, **B** is the matrix which relates nodal displacements $\mathbf{u}^e$ and strain $\boldsymbol{\varepsilon}$ ($\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{u}^e$) and **D** is the Hook's matrix that relates strain and stress $\boldsymbol{\sigma}$ ($\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon}$).

2.1 *Material assignation*

Two different assignation procedures are used for the attribution of the material properties. In the first one a single material model is associated to the closed contour which represents the geometry of the prosthetic device. In the other, the user associates mechanical characteristics to a number of selected gray values. Since the bitmap is represented in a range between 0 and 255, the material properties corresponding to the remaining shades of gray are calculated by performing a linear interpolation of the values introduced by the user. In this way, the pixel values are associated to mechanical properties in advance (Helgason et al. 2008). Note that all the pixels in the area of interest are introduced in the numerical integration used to characterize the behaviour of each element of the mesh, so that in each element a kind of property homogenization is performed. This allows all the information in the medical image to be used without increasing the number of degrees of freedom (associated to the number of elements in the mesh) of the problem. This has the effect of reducing the computational cost of the numerical analysis with respect to other techniques.

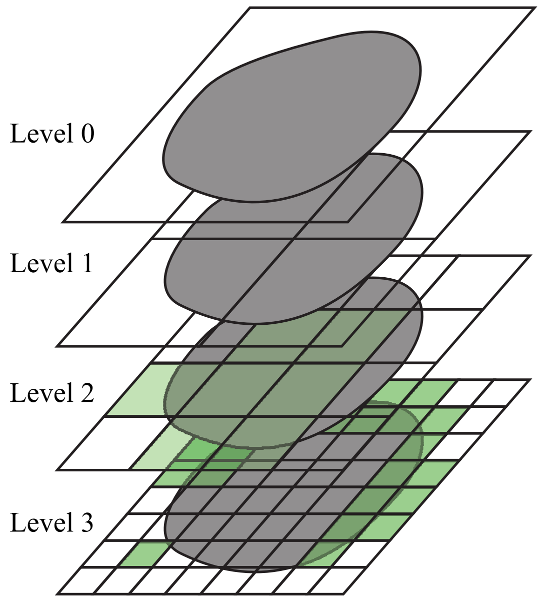

Figure 1: Hierarchical structure of Cartesian grids. Elements of different levels are used to create the analysis mesh, extracted from Nadal et al. (2013).

2.2 *The Finite Element discretization*

Firstly the bitmap dimensions are increased so that the image dimensions becomes powers of two. For this purpose *black* pixels are added at the borders of the bitmap. They do not change the material properties of the model because they are assigned the stiffness of air. The new bitmap perfectly fits the hierarchical Cartesian grid structure shown if Figure 1. It can be seen it consists of different mesh level, each one obtained by dividing the elements of the previous one into four identical rectangles. In this way the possibility of *h*-refining the mesh is guaranteed because at each mesh level an integer number of pixels is contained in each element, with the limit of one pixel per element. The model creation consists of two phases. In the first one the medical image without implant is meshed with a uniform Cartesian grid, the area of interest selected and the material properties univocally assigned to the gray level values. Then the implant geometry is defined (or imported) over the medical image. The pixels located inside the prosthesis boundary are deactivated because they represent the tissue that has to be removed to implant the prosthetic device. At this stage a first uniform mesh is intersected with the contour of the geometry and three classes of elements are distinguished. The three kinds of elements are represented in Figure 2 which shows the elements completely inside the geometric boundary (the dark gray ones), those totally on the bitmap (the white and light gray ones) and the elements on the boundary (those cut by the blue curve). Each class is integrated differently.

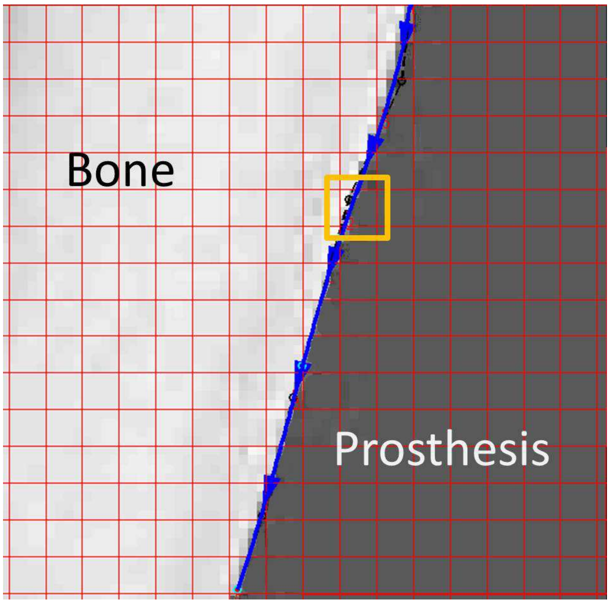

Figure 2: Homogeneous mesh of the area at the boundary between bone an prosthesis.

2.3 *The Element Integration*

For the integration, the elements inside the geometry are treated in the usual way and take into account the elastic properties of the material of the prosthetic device. For these elements the nature of the CG-FEM allows efficient calculation and storage of the stiffness matrix Nadal et al. (2013). The elements which are completely located on the bitmap use the pixels as integration points. Each pixel has its own material properties, depending on the gray level, and the corresponding weight in the numerical integration is equal to its square area in local coordinates. The elements which lie on the boundary between the living tissue and the implant, that is between the bitmap and the geometry, are divided into triangular integration subdomains on the geometry side and into square ones which coincide with the pixels on the side of the bitmap, see Figure 3.

In general, the triangular subdomains and the square ones can be overlapping on some areas. There can also be parts of the element which belong neither to the triangular domains nor to the square ones.

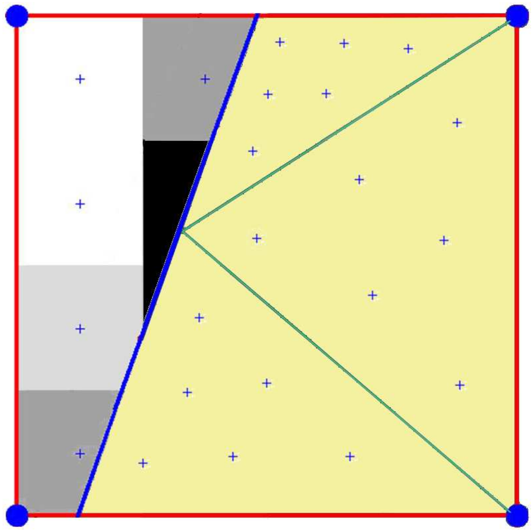

Figure 3: Integration domains and points of an element on the contour.

The involved error has been considered negligible. Note that it cannot be reduced by refining the mesh (because this does not change the integration domains corresponding to the pixels), but only by enhancing the resolution of the image. The elements on the boundary between bone and prosthetic device have overall elastic properties which come from both the heterogeneous material defined in the bitmap and the homogeneous one assigned to the geometry. This means the final properties of these elements homogenize the mechanical characteristics of both bone and prosthesis.

2.4 *The refinement process*

The homogenization due to the integration can lead to lacks of accuracy in the description of the elastic behaviour of the model. This would increase with the level of the heterogeneity. This is particularly important along the interfaces both explicitly defined by the geometry or implicitly by the gray field. To reduce this error a *h*-adaptive process is performed to decrease the element size according to the change of material. This also enhances the solution in the areas of the bitmap that show a fast change of material, for example at the interface between cortical and trabecular bone. The refinement process is also guided by the geometrical singularities in order to enhance the stress representation around them, where the stress gradient is expected to be large.

Once the bitmap and the geometry have been introduced in the structure of the Cartesian grid, it is very simple to substitute an element from a level with the corresponding ones of a more refined level. The necessary information is calculated by a number of routines in a very efficient way thanks to the hierarchical nature of the mesh structure. It makes the calculation fast on one hand but, on the other hand, it also allows memory to be saved, because it makes unnecessary to store most of the information about the mesh. The result is a structured non-conforming analysis mesh in which Multi-Point Constraints are imposed to guarantee the C_0 continuity of the displacement field along the sides of the adjacent elements of different refinement levels.

2.5 *The boundary conditions*

Enforcing the boundary conditions requires geometrical entities to be defined. Parts of the implant contour or new curves defined by the user can serve the purpose. Regarding the Neumann conditions, only require the evaluation of the integrals along internal boundaries in the elements. The Lagrange multiplier technique is used for the imposition of the Dirichlet condition (prescribed displacements), Nadal et al. (2013).

3 NUMERICAL RESULTS

We recall that the application of the method to 2D problems is only an initial stage before its implementation for the 3D data sets provided, for example, by CT-scans. A FE calculation has been performed which simulates the effect of a hip prosthetic device in a femur. A x-ray image of a femur has been used for this purpose, see Figure 4. It has been cut out in order to delete the joint.

Figure 4: X-ray of a femur and the same image with the joint removed.

Then, straight lines, a B-spline and a NURBS (non uniform rational B-splines) have been used to define the contour of a prosthesis. An additional line was introduced to enforce homogeneous Dirichlet boundary conditions at the bottom of the image and a parabolic pressure was applied to the joint of the prosthesis, see Figure 5.

The pressure has a maximum value of 10 MPa at the centre of the loaded arc and goes to zero at the edges.

Concerning the material properties, the pixels which have their center inside the geometrical contour have been deactivated and homogeneous material properties have been assigned to the closed domain.

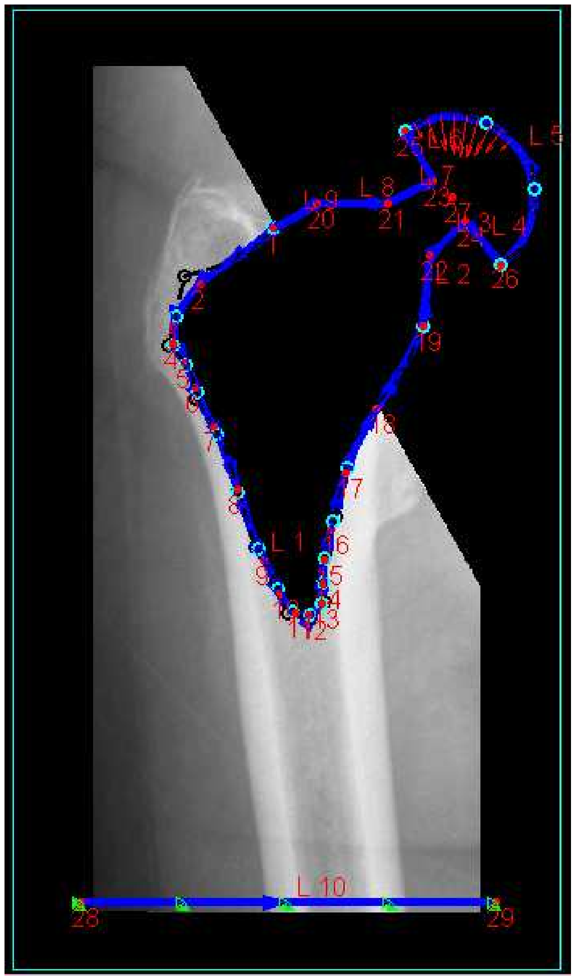

Figure 5: X-ray with the geometry of a hip prosthetic device.

The material properties of bone, muscle and titanioum have been assigned according to Table 1. The interpolation over the gray scale, between the gray levels where material properties have been defined, is shown in Figure 6 and in Figure 7.

Material	**E [GPa]**	ν	**Gray level**
Titanium	116.000	0.32	–
Bone	14.200	0.30	255
Muscle	0.645	0.43	150
Air	0.000	0.00	0

Table 1: Material Properties referred to Figure 4.

The material properties have been taken from literature (Viceconti et al. 1998) and (Kim et al. 2010).

For the creation of the FE model, the bitmap and the geomety have been immersed in an initial uniform mesh of level 5 and then an *h*-adaptive refinement was performed by imposing that the maximum ratio between the standard deviation and the mean value of the gray field in each element cannot be greater than 3 and the maximum refinement level allowed was 9. The resulting analysis mesh is shown in Figure 8. Note that although the different living tissues have not been explicitly segmented, the mesh refinement process automatically identifies the boundaries of the tissues and refine the mesh to properly capture their geometry.

The von Mises stress field induced on the system

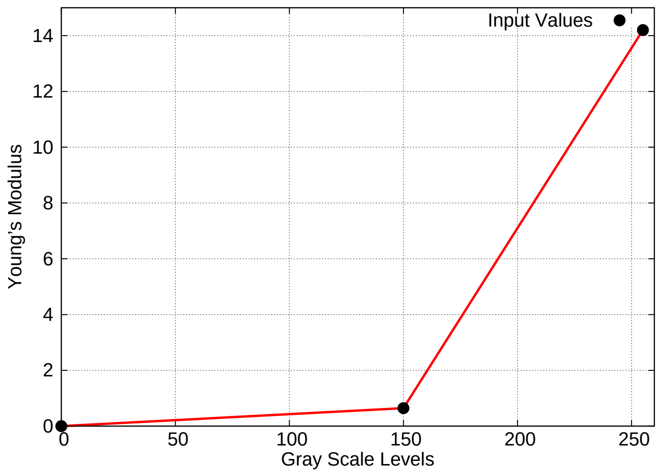

Figure 6: Interpolation of E over the gray scale.

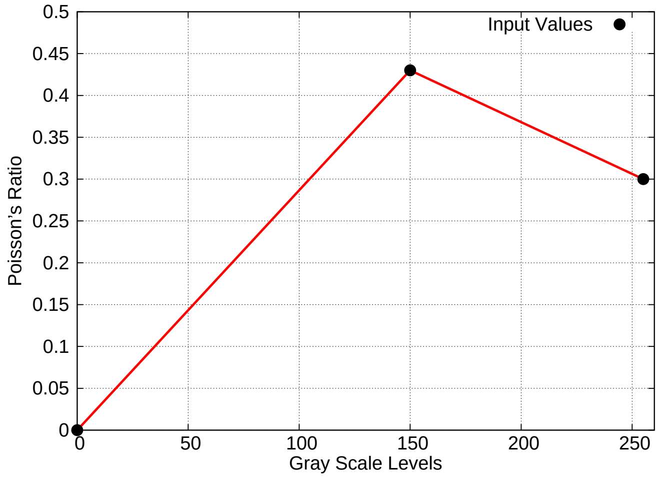

Figure 7: Interpolation of ν over the gray scale.

prosthesis-bone by the load was calculated and it is shown in Figure 9.

Considering the limitations of the image, the von Mises stress field obtained is reasonable. The compression-bending appears in the stress distribution with higher values near the edges of the prosthetic device and of the bone and the transfer of momentum from the device to the human tissue. The stress concentration at the neck of the prosthesis has been captured.

4 CONCLUSIONS

A strategy has been proposed for modelling the effect of the interaction of implants with bones on the basis of preoperative medical images. The strong points of the method are: a) the lack of the segmentation procedure, with the exception of the creation of the areas for the imposition of the boundary conditions, and the simplicity of introducing the prosthetic device by geometrical models. The methodology has been successfully implemented in 2D. The numerical result obtained in this 2D implementation is promising regarding a future implementation of the method for 3D problems with CT-scans.

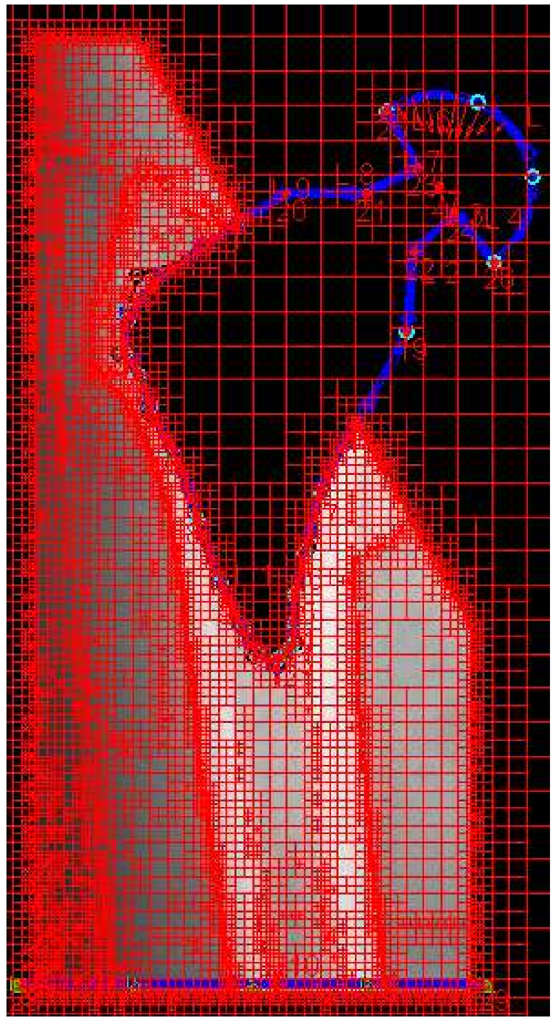
Figure 8: FE mesh of the system femur-prosthesis.

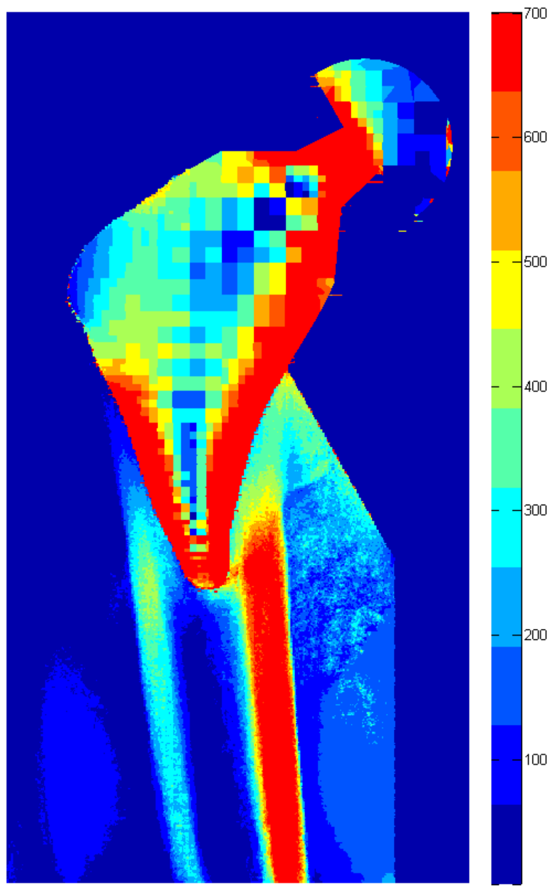

Figure 9: Von Mises stress field of the problem in Figure 8.

ACKNOWLEDGEMENTS

With the support of Framework Programme 7 Initial Training Network Funding under Grant No. 289361 ‘Integrating Numerical Simulation and Geometric Design Technology (INSIST)’, Ministerio de Economía y Competitividad of Spain (DPI2010-20542) FPU program (AP2008-01086) and Generalitat Valenciana (PROMETEO/2012/023).

REFERENCES

Giovannelli, L., O. Marco, J. Navarro, E. Giner, & J. Ródenas (accepted 2013). Ditrct creation of finite element models from medical images using cartesian grids. *Proceedings of the IV ECCOMAS Thematic Conference on Computational Vision and Medical Image Processing: VipIMAGE 2013*.

Helgason, B., E. Perilli, E. Schileo, F. Taddei, S. Brynólfsson, & M. Viceconti (2008). Mathematical relationships between bone density and mechanical properties: A literature review. *Clinical Biomechanics 23*, 135–146.

Kim, H., P. Jürgens, S. Weber, L. Nolte, & M. Reyes (2010). A new soft-tissue simulation strategy for cranio-maxillofacial surgery using facial muscle template model. *Progress in Biophysics and Molecular Biology 103*, 284–291.

Nadal, E., J. Ródenas, J. Albelda, M. Tur, J. Tarancón, & F. Fuenmayor (2013). Efficient finite element methodology based on cartesian grids: Application to structural shape optimization. *Abstract and Applied Analysis 2013*, Article ID 953786, 19 pages.

Taylor, M., K. Tanner, M. Freeman, & A. Yettram (1994). Cancellous bone stresses surrounding the femoral component of a hip prosthesis: an elastic-plastic finite element analysis. *Medical engineering & physics 17*, 544–550.

Viceconti, M., L. Bellingeri, L. Cristofolini, & A. Toni (1998). A comparative study on different methods of automatic mesh generation of human femurs. *Medical engineering & physics 20*, 1–10.