Aspects Regarding the Validation of Mechanical Solutions through the Finite Element Method for the Human Upper Limb

ANTOANELA NAAJI
Faculty of Computer Science
“Vasile Goldis” Western University
Arad 310025, Bd. Revolutiei nr. 94-96
ROMANIA
anaaji@uvvg.ro, http://www.uvvg.ro

DANIELA GHERGHEL
Takata-Petri SRL
Arad 310375 Arad, Zona Industriala Vest Arad, Str. III, Nr. 9
ROMANIA
daniela.gherghel@eu.takata.com

Abstract: - The work presents a few results regarding the static and dynamic analysis of some biomechanical components through the finite element method. Models, representing parts of the human upper limb, have been studied using static trials. Another study was carried out in order to determine the dynamic behavior of the model for total arm prosthesis. The models have been created with the aid of the SolidWorks program and the trials with the HyperMesh program. Considering the fact that we wish to emphasize the way in which such analyses can be done with a finite element method, from meshing to the obtaining of results, as well as their comparison to the results obtained experimentally, we shall present only a few relevant examples for such studies.

Key-Words: - FEA, static analysis, dynamic analysis, biomechanics

1 Introduction
In biomechanics, as in other sciences, the integration of experimental and analytical models is critical in gaining knowledge and understanding the response of the various bones of the skeleton to the action of various mechanical factors. Experiments provide data that can subsequently be interpreted in various contexts. These investigations are influenced in particular by the achievements in the field of graphic processing and image processing techniques, computational mechanics, genetics and molecular biology. The integration of these techniques leads to obtaining important data for the research concerning the human skeleton, its traumas and its diseases.

The programs used in mechanical modeling and analysis of the various bodies, the three-dimensional one in particular, currently comprise a very great number of application in different areas. Some of the software categories used in biomechanics are [10], [14]:
• Motion analysis programs, which can modify, with the aid of some algorithms, the coordinates of an image in spatial data (3D) as well as the position of the rigid body and can carry out the motion analysis;
• Kinematic and dynamic simulation programs which create the general framework for the development of a wide range of models necessary to the study of human and animal motions;
• Imaging and morphology programs with the following functions: calculation of mass and inertia properties, bone structure analysis, imaging, etc.;
• Finite-element modeling programs which ensure the accuracy of the bone geometry representation, the possibility of parameter variation, the possibility of combining FE models with re-modeling algorithms, the possibility of using FE model in other types of analyses;
• CAD programs used in modeling and analysis from the viewpoint of the mechanical behavior of the various bone or muscular components of the human body.

The finite element method, or the finite element analysis, is based on the concept of building complicated objects out of simpler objects, or of dividing complicated objects into simpler objects for which known calculation schemes can be applied. In
many cases, for the solution of a practical problem, mathematics does not afford sufficiently powerful processes for finding the exact solution, and, most of the times, not even an approximate solution. Hence, the basic idea of the finite element method is to find the solution to a complicated problem by using individual components or *elements* whose behavior is entirely known [6].

The finite element analysis method is also apparent as a necessity of calculating and analyzing complex resistance structures, having more particular properties, such as the biomechanical ones, for which the analytical calculation methods are no longer operative.

The objective of this paper is to present a few results regarding the static and dynamic analysis of some biomechanical components through the finite element method.

The bone system components of the human upper limb will be analyze with the aid of this method, with a view to determining the state of stresses and deformations in propping and charging conditions as close to the real ones as possible.

2 Method

The finite element analysis solves several types of problems, among which: time-independent problems (analysis of tensions and deformations, static analysis of structures), proper value problems (natural frequencies and proper structure modes), necessary to the study proposed [2], [3], [4].

A number of steps have to be taken for the analysis:

- Creation of the geometric model, as presented;
- Meshing into different types of finite elements (in our case, tetrahedral three-dimensional elements);
- Defining of the charging conditions (forces applied as pressures to the surfaces of the elements);
- Application of nodal constraints (nodal blockages, in props etc);
- Running of the static or dynamic analysis;
- Generation of an analysis result.

Bones have certain characteristics determining their mechanical behavior. They are highly resistant as a result of their composite microstructure. At a macrostructure level, bones are featured as a composite material for both structural types: compact and spongy. The compact bone is very dense, strongly calcified and, although it has some hollows and channels in its structure, they are not visible. Its structure is created to resist to compression. It can also resist to twisting and bending, but to a lesser extent. The mechanical properties of the bone greatly depend on age, gender and state of health of the human subject to whom it belongs. Moreover, its behavior being elastic-plastic, the mechanical properties of the bone also vary according to time.

The mechanical properties of the spongy bone are significantly lower than those of the compact one and are influenced by bone density value. Thus, resistance to compression is proportional to the square of the density value, and the longitudinal elastic modulus is proportional to the cube of the density value [6], [8].

The bone structure being different, it has been taken into account also within the static analysis. For this reason, finite elements with different physical properties, densities and elastic modules have been used.

To begin with, three-dimensional geometric models were created for both biomechanical components (humerus, radius, ulna) and the model of a total arm prosthesis (Fig.1).

![Fig.1. 3D models: humerus, radius, ulna and total arm prosthesis](image-url)
Irregular shapes modeling required the usage of certain techniques of solid body construction, such as: reliefs, cuts, holes etc. Each component was modeled using the specific commands of the SolidWorks program, drawing several parallel sections through the bone, which were united and thus the final shape was obtained [10], [13].

The total arm prosthesis, as a simplified model, was created by using the same modeling technique as for the upper limb components. The following were modeled: on the inside, metallic bars with the corresponding joints for shoulder, elbow and hand; a hand resistance component; on the outside, the cosmetic part which takes the shape of a human arm [5], [9].

3 Results regarding the static mechanical trials of the human upper limb components

Based on the model shown in Fig. 1, a section through the meshed model of the humerus was obtained, with an inner layer that is different from the outer one in its material characteristics (Fig. 2).

The meshing of the humerus was effectuated with the aid of over 2,700 elements and over 12,000 nodes, by using tetrahedral three-dimensional finite elements. These elements were chosen with a view to obtaining results as close to the real one as possible.

The mechanical trials that are object to the static analysis were: compression, stretching and bending. Compression was applied to the humerus (distal / proximal end), ulna and radius; stretching to the humerus and bending to all of the three components.

The static analysis results are represented through the color diagram of the equivalent stresses: Von Mises, Maximum Principal Stress and Major stress, as well of the maximum deformations.

In the case of compression of the proximal end of the humerus, the forces acting upon it were applied in the form of a pressure on the finite element surface. In this case, at the distal end was applied constrains (node blockages) (Fig. 3).

Similarly, in the case of compression of the distal end of the humerus, the application of forces and constraints was reversed, in other words, the forces of pressure were applied to the distal end and the blockages to the proximal one.

The static analysis results, by determining equivalent stresses for the proximal and/or distal end, are shown in Fig. 4.

Fig. 2. Meshing of the humerus into its 2 layers: spongy and compact [9]

Fig. 3. Application of forces and constraints to compression to the proximal end of the humerus

Fig. 4. Equivalent stresses for the proximal (a) and distal (b) ends of the humerus
Fig. 4. Results provided through the types of stresses on the compression of the humerus’ epiphyses.

One can notice that in the case of compression, the charge is taken over by the compact layer of the diaphysis, as emphasized in the color diagram with the maximum stress values. At the same time, maximum stresses can emerge also in the areas of passage from the diaphysis to the epiphyses, materialize in real life, for example, by the frequent emergence of humeral neck fractures.

Following the applications of the same method for the static analysis of compression of the humerus, the results show that in the area of the epiphyses, maximum displacements emerge in distal or proximal epiphyses, depending of the application of compression, to the distal or the proximal end (Fig. 5).

Fig. 5. Maximum displacements at the compression of the humerus’ proximal epiphyses [9]

Fig. 6 illustrates an example of application of forces to the finite elements’ surfaces, in the case of stress to the stretching of the humerus, with a wire frame view.

Fig.6. Application of forces, stress to stretching of the humerus.

In the case of stress to the bending of the humerus, the maximum stresses are found in the area of passage from the diaphysis to the distal epiphysis, while the maximum deformations are also located in the area of the distal epiphysis.

For the meshing of the ulna and the radius over 60,000 elements, and 13,000 respectively, were used, and over 12,000 nodes. Here, too, tetrahedral three-dimensional elements were employed. Charges and constraints are emphasized in Fig. 7, these components being only subject to bending.

Fig. 7. Application of charging forces and constraints to the ulna and radius [9]

The results of the study regarding the bending of the ulna and the radius are: maximum stresses in the charging application areas and maximum displacements in the same areas. Considerable stresses can emerge in the propping areas, owing to nodal blockages.
4 Results regarding the mechanical trials at proper frequencies of the total arm prosthesis

We considered the simplified 3D model of the total arm prosthesis, designed in Solid Works, on which the dynamic study at proper frequencies was conducted [9].

The work methodology is the same as the one presented in the static analysis section. The characteristics of the composing materials were introduced in correspondence with the parts of the prosthesis. The composing elements of the prosthesis were meshed into tetrahedral three-dimensional elements and over 500,000 elements, 140,000 nodes, respectively, were obtained (Fig.8).

Fig.8. Meshing of prosthetic components (cosmetic part, joints, bars)

The constraints were applied to the outline surface of the cosmetic part and to the elements of the inner mechanism through which the prosthesis is attached (Fig. 9).

Fig.9. Nodal constraints applied to the prosthetic component and the inner mechanisms of the prosthesis

The results of the study at proper frequencies are shown in Fig. 10, pointing out that two proper frequencies were found, on the two directions of study.

Fig.10. Proper frequencies on the two directions of the study [9]
One can draw the conclusion that the behavior of the cosmetic part did not influence the dynamic analysis results (Fig. 11).

![Fig.11. Dynamic analysis results in the structure of prosthetic components](image)

5 Conclusion
This work has presented the 3D models of biomechanical and prosthetic components, the study of their mechanical behavior being conducted by using the finite element method.

The bone system components of the human upper limb, analyzed with the aid of this method, with a view to determining the state of stresses and deformations in propping and charging conditions as close to the real ones as possible, have led comparative results to those obtained by way of experimentation [7].

Propping and charging conditions of the composing models of the human upper limb can be found on the real-life human skeleton.

The dynamic study of proper frequencies in the case of total arm prosthesis has led to obtaining results whose values did not surpass the limits of the proper frequencies.

References: