The Application of the Finite Element Method in the Biomechanics of the Human Upper Limb and of some Prosthetic Components

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 inclusion of analytical and experimental models in biomechanical studies leads to the obtainment of important data for the research concerning the human skeleton, its traumas and diseases. The paper showcases a number of results regarding the static and dynamic analysis of some biomechanical components by using the finite element method (FEM). Models, representing parts of the human upper limb, have been studied using static trials.

Considering the fact that we wish to emphasize the way in which such analyses can be done with a finite element method, we shall present only a few relevant examples for which we have experimental data, namely: analysis of the compression, bending and stretching of the humerus and bending of the radius and the ulna. Another study was carried out in order to determine the dynamic behavior of the model for total arm prosthesis, using FEM. The models were created with the aid of the SolidWorks program and the trials with the HyperMesh program.

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

1 Introduction

As in other sciences, in biomechanics, the integration of experimental and analytical models is critical in gaining knowledge and understanding the response of the various bones in 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 programs used in mechanical modelling and analysis of the various bodies, the three-dimensional one in particular, currently comprise a very great number of applications in different areas. Some of the software categories used in biomechanics are [12], [19]:

- **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;
- **Kinematical 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 modelling programs** which ensure the accuracy of the bone geometry representation, the possibility of parameter variation, the possibility of combining FE models with re-modelling algorithms, the possibility of using FE model in other types of analyses;
- **CAD programs** used in modelling and analysis from the viewpoint of the mechanical behaviour 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 ones, or of dividing complicated objects into simpler ones, 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 behaviour is entirely known [8].

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.

In order to use the finite element method in solving some problems, one must complete a few stages, detailed as follows [17]:

- The study of the structure

This study must be conducted for the correct determination of the model and of the appropriate finite elements (in accordance with the accuracy and quality of the expected results), which would model as accurately as possible the actual state of tension and deformation.

- The discretisation of the structure

This must be carried out so as, for the size of the finite elements to be as small as possible in the areas of interest in which a very accurate result is desired. The discrete structure may contain both areas with small-sized finite elements, and areas with large finite elements. The transition from one type of area to another must always be made with the aid of progressive transitional elements, in order to eliminate distortions that can lead to abrupt transitions.

A very important step in choosing the discretisation mode consists of checking the finite elements for distortions. In this respect, the recommendation is that the ratio of the edge lengths should be close in value to one and, in the case of finite elements shaped as a quadrangle or a hexahedron, the angles between the edges should be as close in value to 90° as possible. The distortions having emerged, for some reason, in the geometry of finite elements, can induce severe distortions of the obtained results.

- The study of finite element

This is necessary for establishing the equations for finite elements, which describe the behaviour of the medium within an element. In these equations, the variable is represented by the degrees of freedom imposed to the type of element employed and is determined depending on the category to which it belongs.

- The transformation of rigidity matrices

This transformation is necessary for the transfer of elements from the local frame of reference to the global frame of reference of the structure.

- The assembling of elemental equations

This step entails the assembling of elementary equations in the frame of reference attached to the structure (the assembling of finite elements). A condition is imposed for the unknown functions of the problem to have the same values in the common nodes.

- The solving of the system of equations

After obtaining the reduced system of equations, the latter will be solved through known numerical procedures. Thus, the values of the degrees of freedom in the nodes, representing the main variables in the system, will be determined.

- The calculus of secondary variables

After determining the main variables, the secondary variables will be ascertained, the latter being, for the structure of resistance, the specific deformations and the components of the tension tensor.

The objective of this paper is to presents 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.

The work also features a 3D total arm prosthesis model for which a dynamic study was conducted at proper frequencies, using FEM.

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 [3], [4], [5]. Using specific software, some of the steps mentioned in previous chapter are solved by the program.

HyperMesh, by Altair Engineering program [19], is a pre-post processor of high precision which allows a fast setup of the models and rapid interpreting of results. The technological advances facilitate an efficient differentiation with high accuracy which ensures quality control. **Optistruct Analysis** [20] is a fast and accurate solver for linear and non-linear analysis and it was used as well. For differentiation, we have used tetrahedral finite elements, processed automatically as well as manually on models created in **Solid Works** [18]. The models were converted into .igs files in order for the program to recognize them as 3D geometric models. In showing the bone structure as different, we have considered the static analysis. For this analysis, finite elements with different physical properties, densities, and modules of elasticity were used.

For the analysis, a number of steps have been taken:

- 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 behaviour. 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 behaviour 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 [8], [10].

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) [11], [12].

Fig.1. 3D models: humerus, radius, ulna and total arm prosthesis

3D modeling was carried out with the aid of the SolidWorks programs starting from the real dimensions of the bones, based on the various models-moulds made of different materials, as well as with the aid of images from human anatomy atlases and expertise catalogues [2], [13], [14].

The bone system components of the human upper limb were created as solid, full geometric bodies (without considering the marrow). 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 [1], [12], [16], [18].

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 [6], [7], [11].

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

Based on the models 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) [11].

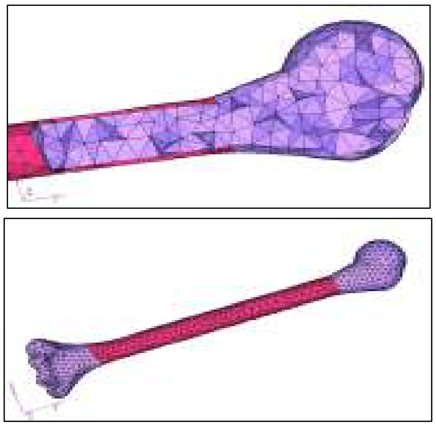

Fig.2. Meshing of the humerus into its 2 layers: spongy and compact

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) [11].

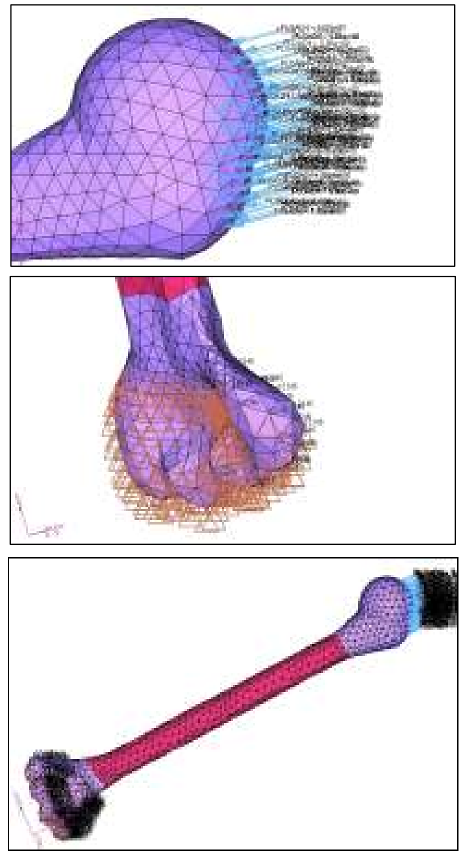

Fig.3. Application of forces and constraints compression of the proximal end on the humerus

The static analysis of compressions results by determining the equivalent Von Mises tensions for the proximal end, as shown in Fig. 4 .

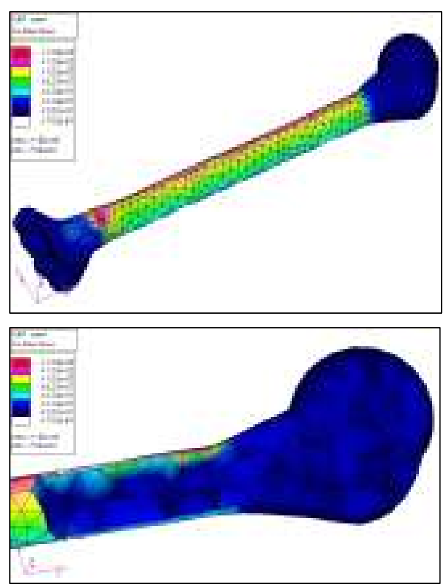

Fig.4. Results (Von Mises tensions) provided by the types of stresses on the compression of the humerus' proximal end

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 (Fig.5). The results of the static analysis for compression are also presented in Fig.5.

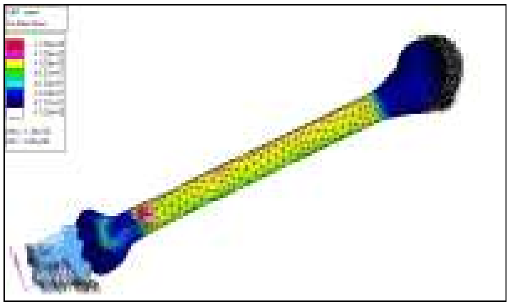

Fig.5. Charges / constraints, as well as results (Von Mises tensions) provided by the types of stresses on the compression of the humerus' distal end

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 colour diagram with the maximum stress values. At the same time, maximum stresses can also emerge 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. 6 and 7 show the results of displacement under static compression of the humerus at both proximal and distal epiphyses [11].

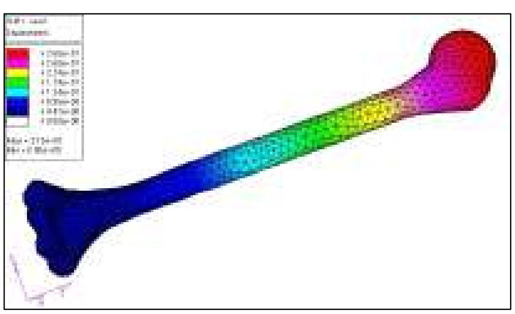

Fig.6. Maximum displacements at the compression of the humerus' proximal epiphyses

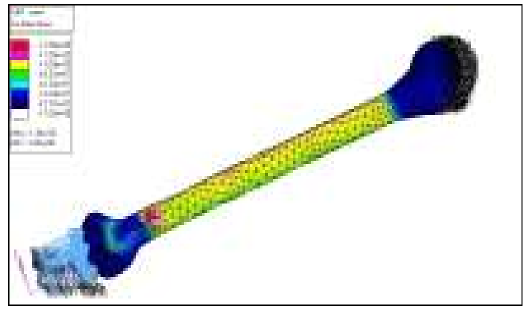

Fig.7. Maximum displacements at the compression of the humerus' distal epiphyses

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 (Fig 8).

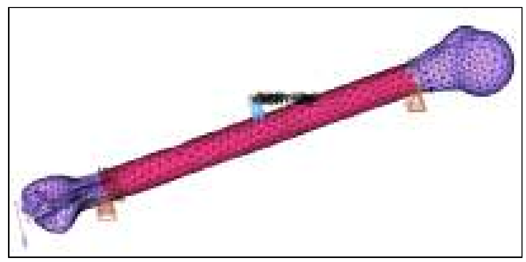

Fig.8. Constraints and bending forces applied to the humerus

The results obtained for static stress on bending the humerus, the equivalent Von Mises tensions, as well as the displacements, are shown in figures 9 and 10 [11].

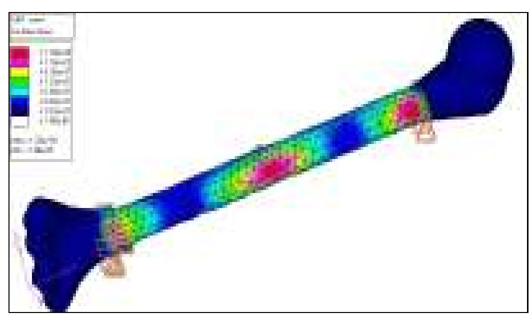

Fig. 9. Von Mises tension on bending the humerus

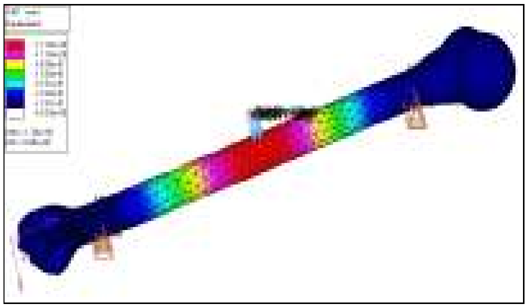

Fig.10. Displacements on bending the humerus

Fig. 11 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.

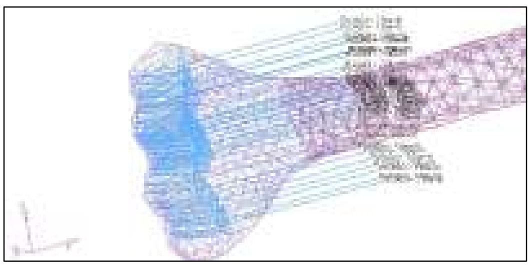

Fig.11. Application of forces, stress to stretching of the humerus

In the case of stress by stretching, the humerus was constrained at one of its ends, and the other was applied forces for the simulation of stretching (Fig.12).

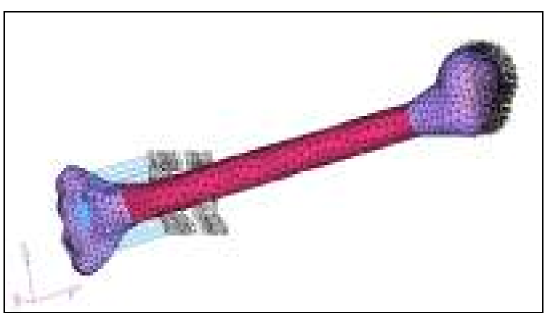

Fig.12. Constraints and the stretching force applied to the humerus

The results of this stress are shown by the Von Mises tensions and by the displacements in figures 13 and 14 [11].

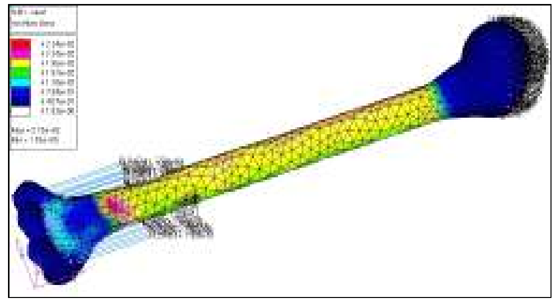

Fig.13. Von Mises tensions on stretching applied to the humerus

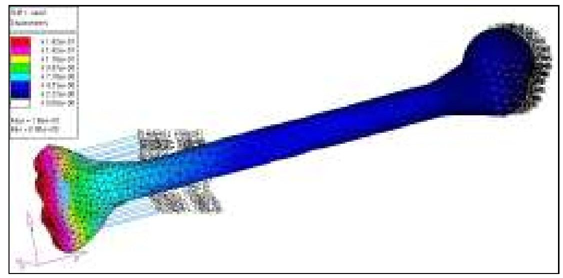

Fig.14. Displacements on stretching applied to the humerus

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.15, these components being only subject to bending [11].

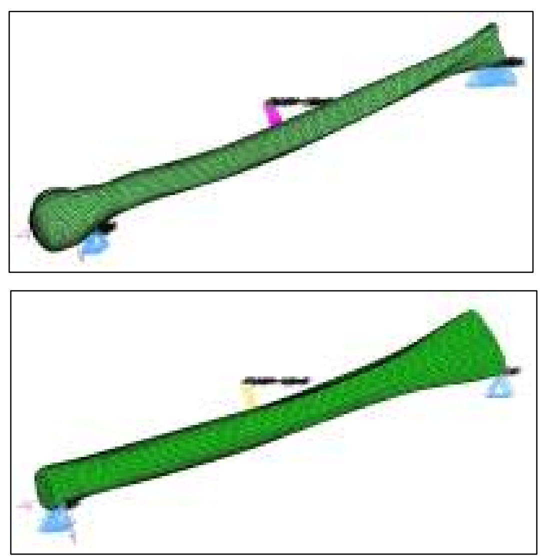

Fig.15. Application of charging forces and constraints to the ulna and radius

Following the analysis of the static stress on bending the ulna and the radius, we can view the equivalent Von Mises tensions, as well as well as the displacements, in the figures 16-19.

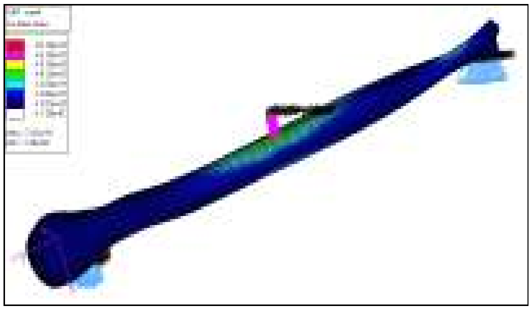

Fig.16. Von Mises tension on bending the ulna

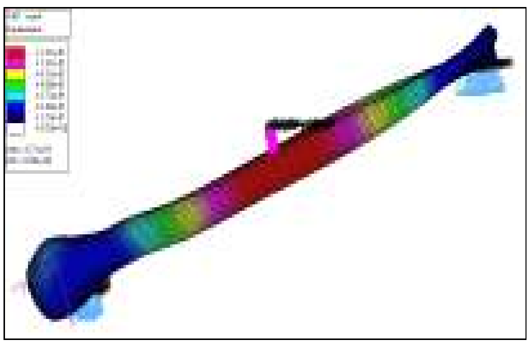

Fig.17. Displacements on bending the ulna

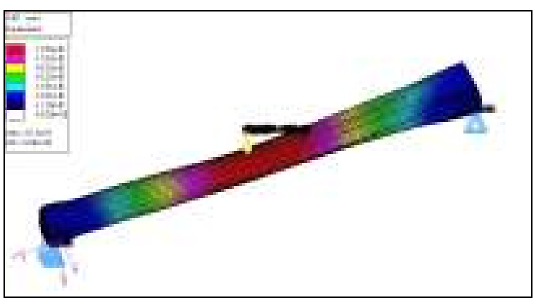

Fig.18. Von Mises tension on bending the radius

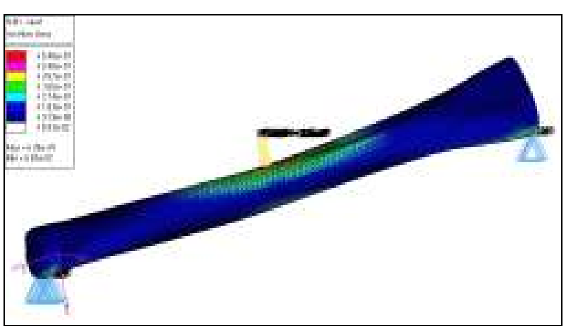

Fig.19. Displacements on bending the radius

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 Experimental results

Experimental research regarding quasi-static stresses has led to various results [9],[15]. Test tubes were utilized, having previously processed, by measuring the breakage forces for each type of stress and the surfaces of the transversal sections were calculated, being considered as trapezoidal or triangular. The epiphyses of the bones were debited and for compression the maximum height of 20 cm was used. The tensions were calculated by dividing the forces occurring during the stress by the surfaces of the sections. For the humerus, stressed by compression, average tension values of 1.2 MPa were obtained on the proximal end and of 1.5 MPa on the distal one. In the case of stress on bending, average tension values of 2 MPa were obtained for the humerus, 2.3 MPa for the radius and 4.4 MPa for the ulna. For the stress on stretching of the humerus, average tension values of 2 Mpa were obtained, according to the data from Table 1.

Bone type	**Static trial**	**No. of samples**	**Average value [MPa]**	**Standard deviation [MPa]**
Humerus	*Compression-proximal end*	11	1.2348	0.6284
	Compression-distal end	11	1.4593	0.7369
	Stretching	8	3.0035	1.6932
	Bending	10	2.0971	2.5682
Radius	*Bending*	11	2.2988	1.0205
Ulna	*Bending*	12	4.4310	2.3989

Tab. 1.Experimental values for the bones of the human upper limb

Long bones are the most resistant and the most complexly stressed, and after comparing the results, we arrived to the conclusion that, at the distal end of the humerus, the values obtained are higher than those at the proximal end. By comparing the values of tensions at breakage by bending and stretching to those obtained on compression, significant differences were found.

This conclusion is also due to the fact that the test tubes for compression trials contained samples from the distal ends of bones where the structure is trabecular, and for the stretching and bending ones the samples were taken from the central area of the bones where the structure is cortical.

5 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 [6], [11].

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.20).

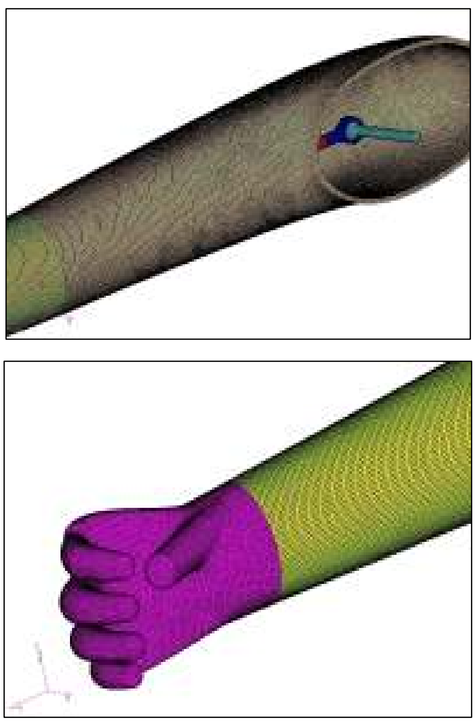

Fig.20. Meshing of prosthetic components (cosmetic part, joints, bars)

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. 21).

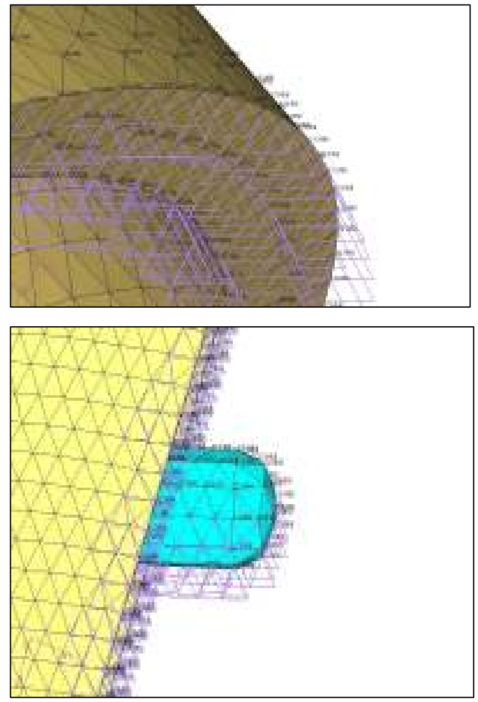

Fig.21. Nodal constraints applied to the prosthetic component and the inner mechanisms of the prosthesis

The results of the study at proper frequencies are shown in Fig. 22, pointing out that two proper frequencies were found, on the two directions of study [11

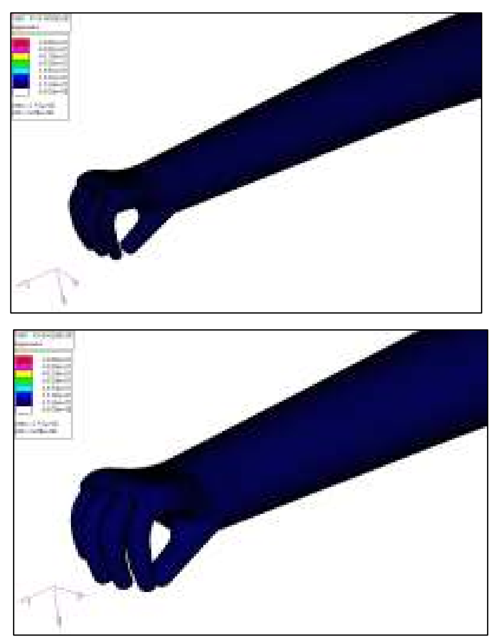

Fig.22. Proper frequencies on the two directions of the study

One can draw the conclusion that the behaviour of the cosmetic part did not influence the dynamic analysis results (Fig. 23).

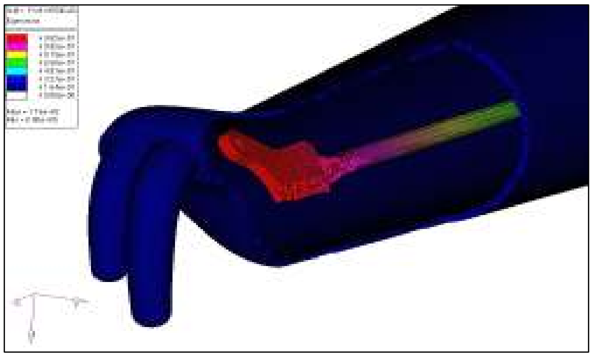

Fig.23. Dynamic analysis results in the structure of prosthetic components

6 Conclusion

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

The static analysis of CAD models on compression trials at the two ends of the humerus; the stretching of the humerus; as well as the bending of the humerus and of the ulna / radius was materialized in the colour diagram of the values resulted.

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.

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

The FEA applied to the dynamic study of the total prosthesis components for the human upper limb was also materialized in the colour diagram with the values obtained for the study frequencies in the two directions.

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:

[1] Abduallah, H., Tarry, C.; Abderrahim, M., Therapeutic Robot for the Upper Limb Rehabilitation, *Wseas Transactions On Systems,* Vol. 6, Issue 1, pp. 88-95, 2007

[2] Abrahams P.H., Hutchings, R.T., Marks S.C. Jr., *McMinns Colour Atlas of Human Anatomy*, Mosby, London, 4-th ed., 1998

[3] Antonescu D.& all., *Metode de calcul şi tehnici experimentale de analiza tensiunilor în biomecanică,* Timişoara, 1986

[4] Baciu C., *Anatomia funcțională şi biomecanica aparatului locomotor,* Bucureşti, 1983

[5] Brickmann P., Frobin, W., Leivseth, G., *Musculoskeletal Biomechanics*, Thieme, Stuttgart – New York, 2000

[6] Chivu, Catrina.; Chivu, C.; Georgescu, C., Design and fuzzy control of hand prosthesis or anthropomorphic robotic hand, *WSEAS Transactions on Systems and Control*, Vol. 3, Issue 5, pp. 333-342, 2008

[7] Dandy D.J., Edwards D.J., *Essential Orthopaedics and Trauma*, 3-rd edition, Churrchill Livingstone, USA, 1998

[8] Dragulescu D., *Modelarea în Biomecanică, Editura Didactică şi Pedagogică*, Bucureşti, 2005

[9] Drăgulescu D., Rusu L., Morcovescu V., Precup C., Comparative study of mechanical stresses in human limbs bones, *Applied Bionic and Biomechanics, vol.1(2), no.1*, Ed. Openmindjournals - New Zeeland, 2004

[10] Fung Y.C., *Biomechanics: Mechanical Proprieties of Living Tissues*, Springer – Verlag, Berlin, 1993

[11] Gherghel D., *Contributii privind recuperarea, prin protezare, a functiilor mecanice ale membrului superior uman*, Ed. Politehnica Timisoara, 2008

[12] Náaji, A., *Tehnici de modelare. Sistem de diagnoza pentru membrul superior uman*, Ed. Orizonturi universitare, 2007

[13] Netter F.H., *Atlas of Human Anatomy*, Second Edition, Novartis, New Jersey, 1990

[14] Papilian V., *Anatomia omului - Aparatul locomotor,* Bucureşti, 1974

[15] Precup, C. – *Studiu anatomo-experimental asupra rezistentei oaselor lungi ale membrelor superior si inferior cu aplicabilitate in diagnosticul etiologic al traumatismelor mecanice produse la nivelul femurului (PhD Thesis)*, Timsoara, 2008

[16] Sirbiladze, G;, Sikharulidze, A.; Sirbiladze, N., Fuzzy programming problem in the weakly structurable dynamic system and choice of

decisions, *Wseas Transactions on System and Control*, vol. 3, issue 11, p. 937-953, 2008
[17] Ungureanu L., *Modele de reconstructie a mâinii umane si a functiilor sale*, Ed. Politehnica, Timisoara, 2007
[18] *** *SolidWorks*, Educational Edition, 2004-2005
[19] *** *CAE Seminare*, Altair Engineering, 2005
[20] *** www.altair.com - Altair Engineering web page