



## MODELING BY FEM OF A TOTAL HIP PROSTHESIS

C. Druga<sup>1</sup>,

<sup>1</sup> Transilvania University, Brasov, ROMANIA, e-mail:druga@unitbv.ro

**Abstract:** This paper presents a modeling total hip prostheses using finite elements method FEM. In the first part of the paper is shown making the model CAD of the prosthesis of hip, in CATIA, a type of prosthesis exist in Biomechanical Laboratory of the Faculty of Design and Environment. The second part of the paper presents the propriu modeling of denture starting from the choice of the resulting force acting on the femoral head and the definition of titanium alloy used to the prosthesis.

**Keywords:** FEA, hip, prosthesis, CAD, CATIA

### 1. CAD MODEL OF THE TOTAL HIP PROSTHESIS

For finite element analysis (FEA) was performed CAD model of a total hip prosthesis Sulzer Ti-Al-Nb 2141674 (Figure 1) has been tested on the stand presented in reference [1]. This prosthesis is equipped Biomechanics Laboratory of Advanced Mechatronic Systems Department -D04 (Faculty of Product Design and Environment).



Figure 1: Image with Sulzer Ti-Al-Nb 2141674 prosthesis

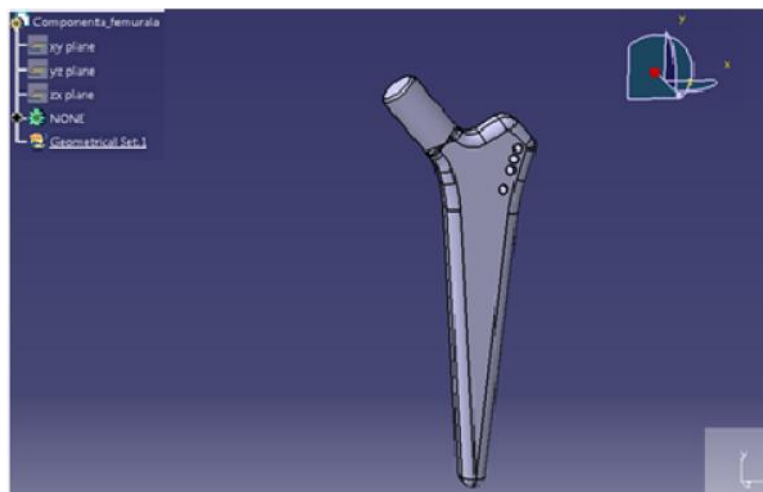
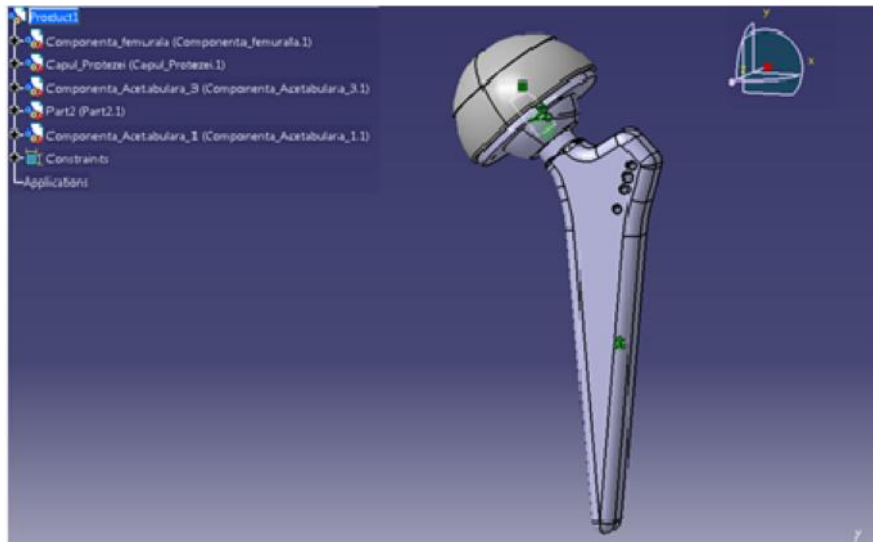


Figure 2: Stem prosthesis made in CATIA P3 V5R19

CAD model of this type of prosthesis and finite element analysis was performed using CATIA P3 V5R19 (figure 2 and 3). CATIA (Computer Aided Three-Dimensional Interactive Applications), product of the company Dassault Systems is currently one of the most used systems integrated CAD/CAM/CAE worldwide, with applications in various fields. At each new update (release) being introduced new modules and additional functionality, in parallel with the improvement of existing ones. CATIA provides a wide variety of integrated solutions to meet all aspects of

design and manufacturing. Of the many basic functions we can mention: Advanced concept of mechanical parts, making interactive assemblies, obtaining automatic projections or current ensemble piece, can design the set etc. Also, CATIA enables the design of parts and assemblies directly in three dimensions without draw first surfboards in two-dimensional representation. Starting with version 5, CATIA start to use increasingly more often the notion of virtual prototype.

This term designates all computerized data that allow the handling of a virtual object, created on the computer, in the same way with a real object. Can be so carry out the following activities: test of resistance to various requests, check if an assembly is removable or not, check the lack of collisions caused by mobility components, some over others [2].



**Figure 3:** CAD model of the total hip prosthesis made in CATIA P3 V5R19

In the case of virtual prototype made in CATIA, any changes that you apply Designer on it are easy to achieve, both due to the flexibility of working tools made available, as well as how to design completely set up. CATIA has a modular structure, which ensures a great versatility, switching from one mode to another is done quickly, with the possibility of continuous edit warring entity without loss of information and without the need for explicit conversion, by the user, the file format specific to each module. Of the modules implemented in CATIA [2]:

- *CATIA Sketcher* – creates an sketch profile in two dimensions, as a starting point in the process of obtaining a three-dimensional object;
- *CATIA Part Design* – using the concept of mechanical parts in three dimensions. It is recommended to use this module in conjunction with CATIA Sketcher;
- *CATIA Assembly Design* – allows the generation of a set of pieces using various mechanical constraints to position them and establishing contacts between surfaces;
- *CATIA Drafting* – contains tools to obtain execution drawings of parts and assemblies created.
- *CATIA Knowledge Advisor* – supports the user in the design set, using specific tools, such as formulas, parameters, rules and reactions, activated only after completion of a condition agreed upon in advance.

CATIA module Sketcher provides a complex set of functionalities and outlining tools and constraint, which allow creating and modifying elements (entities) an outline. Very important is the possibility to apply constraints to these elements. The module contains tools to rapidly obtain profiles used as support for a 3D body, but also the tools to permit the establishment of dimensions and constraints outline 2D, stating the user about a possible conflict between certain sizes.

*CATIA Part Design* module makes it possible to design 3D, with an intuitive interface and flexible from an outline representation to the concept of interactive product. The module provides a highly productive working environment in the design of mechanical products, being generally the other modules of the program CATIA V5R16. Each piece is designed with *CATIA Part Design* module contains a number of parameters, use both within or and how CATIA Knowledge Advisor, to set up and automatic creation of parts families, between an overall tracks dependencies etc. *CATIA Assembly Design* module offers many design tools that allow you to define and manage mechanical structures assembled. Some of the components and assemblies are created by the user and others are taken from the library program.

## 2. MODELING WITH THE FINITE ELEMENTS METHOD OF THE IMPLANT

*Geometric domain modeling* was done starting from the geometric dimensions of the implant; they were determined by direct measurements on existing implant in the laboratory. Their validation was done by comparison with

existing data in the catalogs offered by the manufacturer [2]. The second stage was the description of the flat profiles which formed the basis for the generation of 3D elements in Sketcher mode, through the succession of completion commands: *Start* → *Mechanical Part Design* → *Design* → *Sketcher*. Sketcher mode commands have been presented previously. The third stage consisted in the description of 3D elements using Pad module (Start → Design → Mechanical Part Design) [2]. Description of marks and marks was achieved in module Assembly Design (Start → Design → Mechanical Assembly Design) by using specific options (shown above). Thus was achieved CAD model of the hip implant shown in figures 3 and 4. Figure 4 shows the assemblies and the subassemblies of the implant with the specification of the materials used and the types of constraints used.

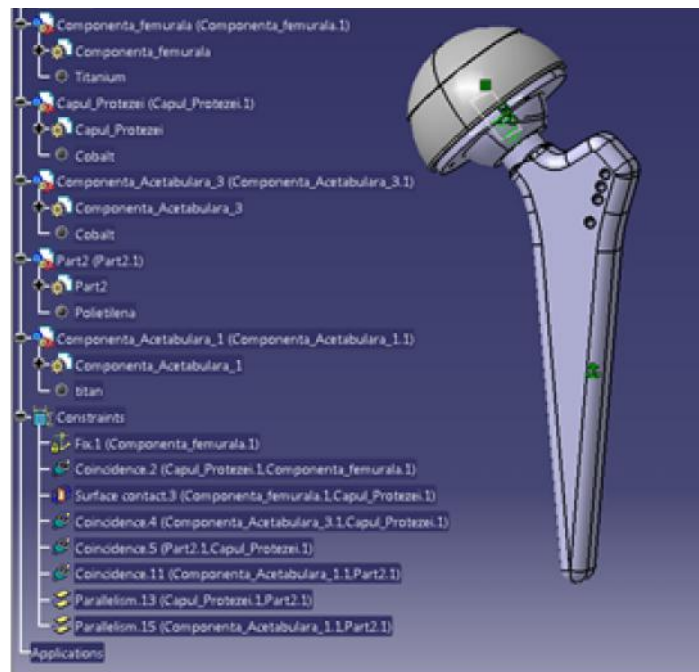


Figure 4

The first step in modeling of the material was done by identifying materials belonging to the implant. Thus, starting from the inscriptions applied by the manufacturer and data from the literature were identified the following materials:

- femoral stem of the prosthesis was made from an alloy of Ti-Al-Nb;
- the head of the prosthesis (diameter 28 mm) was made of alloy of Co-Cr-Mo (Protasul 21WF ISO 5832/12);
- acetabular cup, composed of three parts, as follows: inner lining (in contact with the femoral head) is made of alloy Co-Cr-Mo; intermediate component is from GUR UHMWPE 4150; hubcap acetabular is titanium alloy.

If the prosthesis stem, using the *Apply Material* has identified the type of material (metal section) and Titanium has been selected. Given that the program is not available to the CATIA user types of metal alloys, they must be defined by changing the mechanical and thermal properties. Thus, you enter in the dialog box of the option *Properties*: Young's modulus, Poisson's coefficient, density, coefficient of thermal expansion and yield. These data were obtained from the literature, and for the main bio-materials were presented in [1]. Similarly is defined and Co-Cr-Mo alloy used to manufacture the prosthesis head and acetabular cup inner liners and high density polyethylene (UHMWPE GUR 4150) has achieved intermediate component acetabular cup. Using the *Report* option, from the *Materials* you can get a report on the materials used in the analysis of the model as shown in Figure 5. Generation model with finite elements (static analysis), was achieved in module Generative Analysis by Structural sequence of completion commands: Start → Analysis & Simulation → Generative Structural Analysis → New Analysis Case → Static Analysis → OK.

Constraints modeling was done by: a)-modeling links with base using the Clamp, and b)-modeling links with adjacent elements with Rigid Connection (Figure 6).

## Materials.1

Material	Titanium
Young's modulus	1,14e+011N_m2
Poisson's ratio	0,33
Density	4460kg_m3
Coefficient of thermal expansion	9,5e-006_Kdeg
Yield strength	8,25e+008N_m2

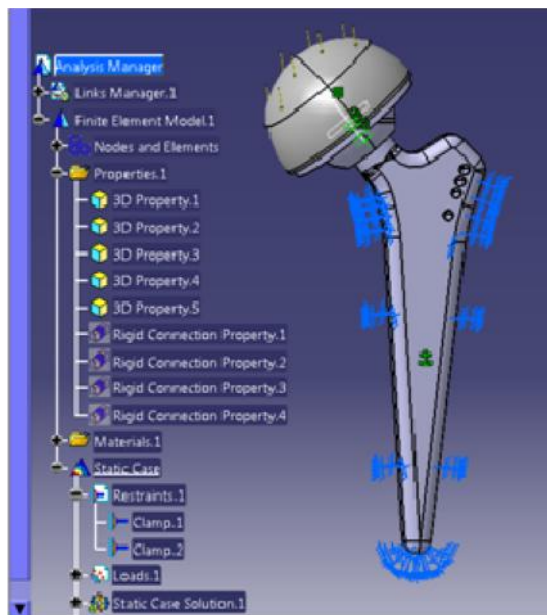
  

Material	Cobalt
Young's modulus	2,2e+011N_m2
Poisson's ratio	0,35
Density	4460kg_m3
Coefficient of thermal expansion	9,5e-006_Kdeg
Yield strength	8,25e+008N_m2

Material	Poietilena
Young's modulus	6e+008N_m2
Poisson's ratio	0,46
Density	1200kg_m3
Coefficient of thermal expansion	6,84e-005_Kdeg
Yield strength	0N_m2

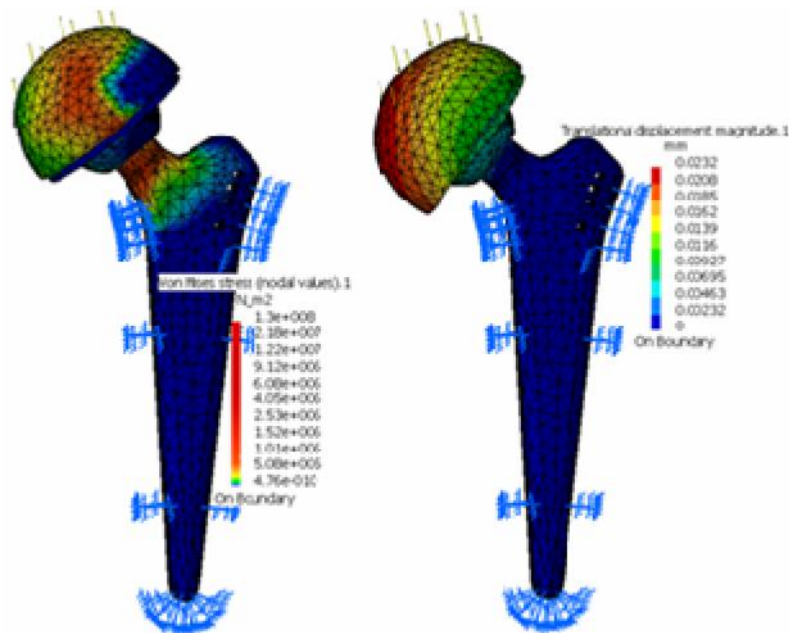
**Figure 5:** The main mechanical and thermal properties of implant materials



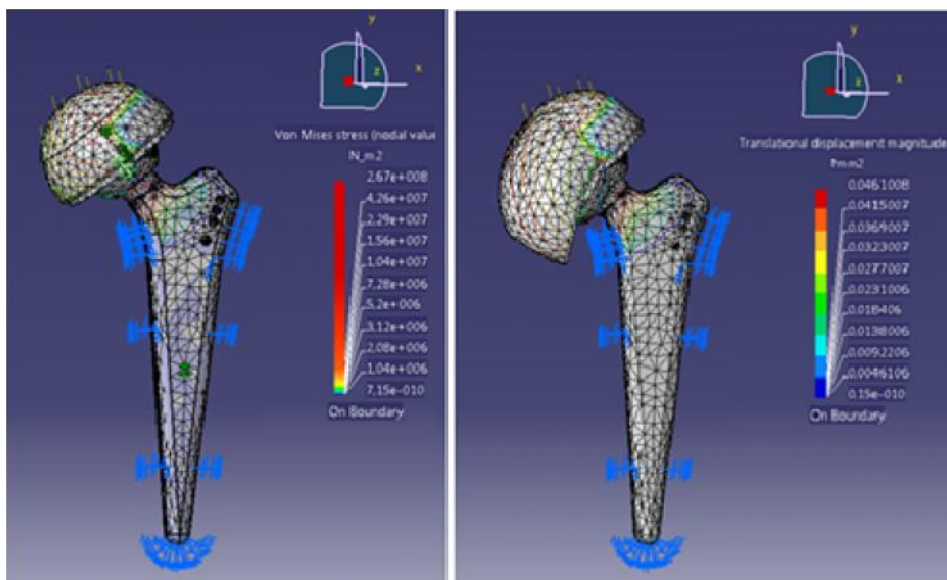
**Figure 6:** Modeling constraints [1]

Workload modeling can be done through one of the commands listed earlier, namely: *Pressure*, *Distributed Force*, *Bearing Load*, and *Moment* [2]. In this case it was opted for *Distributed Force*: a); in this dialog box, the values have been entered, the resultant force acting on the head of the femur, after the three axes [1]. The choice of the resulting force acting on the femoral head is described in detail in reference [1].

For example was chosen the case of two subjects of 98 and 86 kg, carrying out different activities such as: slowly walking, normal walking, fast walking, climbing stair, knee bend, standing, going down stairs and going up stairs. For starters it was taken into account (case with 98 kg) less than the value of the resulting force acting on the femoral head; this occurs in a situation where the subject make a knee bend [1]. In this case the resultant forces is 120% of B.W (body weight) and a value of 1177,3 N; It is considered that this makes an angle of 15 degrees to the vertical. This resultant force was not applied at a point but on a surface with a hotel features about 12 cm<sup>2</sup>; this is the average value of load-bearing surface of the femoral head in a natural healthy joint. Verification of the model provides information on: the type of finite elements in the model; material; the constraints of constituents and is achieved by the *Model Checker*. Solving the model with finite elements is done by the command *Compute*, which provides information about the hardware resources required to resolve the estimation model. The result is shown in Figure 7, where you can see the movements and distribution of von Mises stress.

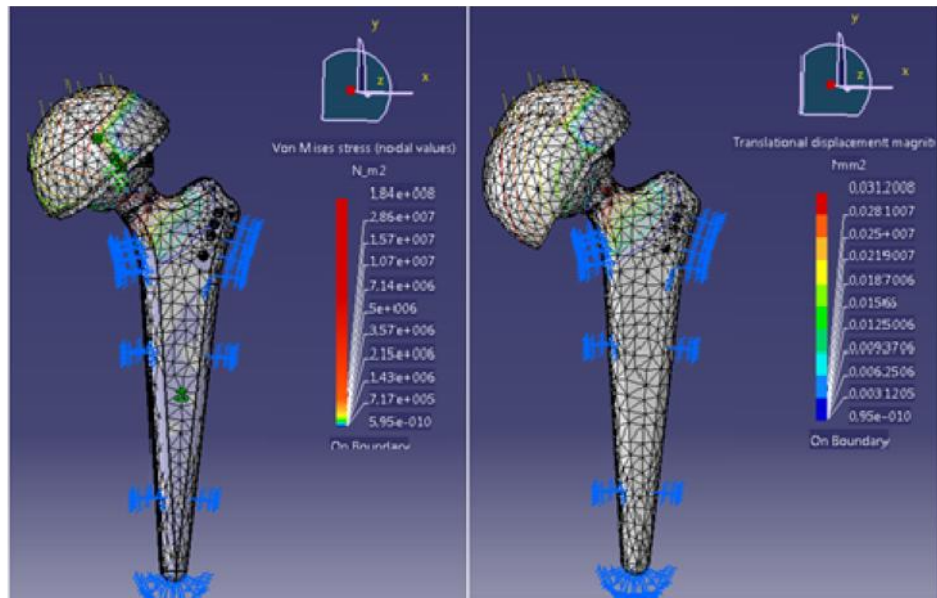


**Figure 7:** The results of FEA to the subject with 98 kg (resultant force 1177.3 N): (a) von Mises stresses fields and (b) movements [1]

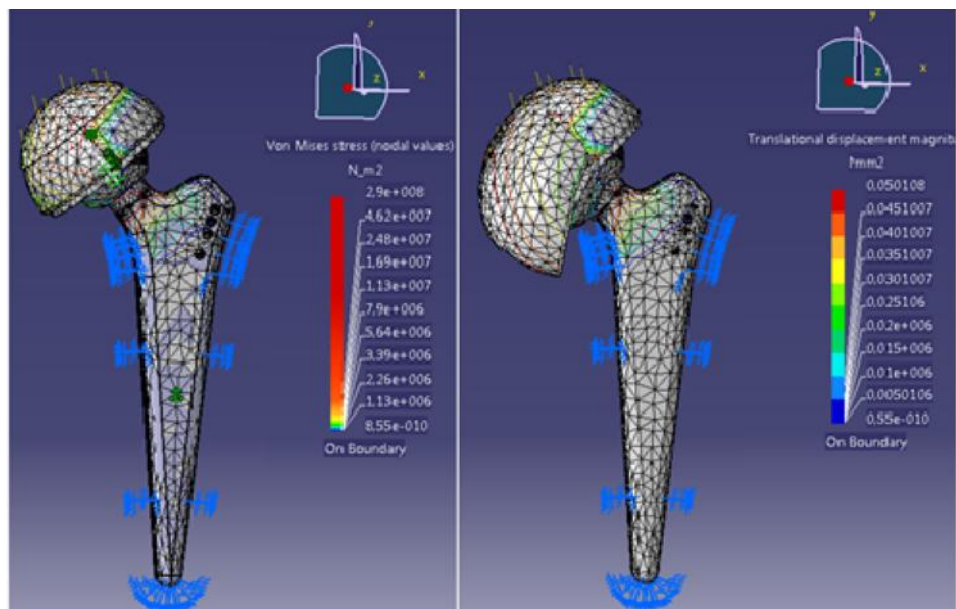


**Figure 8:** The results of FEA to the subject with 98 kg (resultant force 2371.6 N): von Mises stresses fields and movements [1]

The largest value of the resultant force acting on the femoral head, for subject with 98 kg during a slowly walking cycle is 242% of B.W (2371, 6N). By performing the same steps as before he got to the results in figure 8. Similarly it was analyzed the implant model of the subject with 86 kg for the minimum value of the resultant force (1601 N)-Figure 9 and for maximum value (2580 N)-Figure 10. Data obtained from FEA for the two subjects to which it took into account the minimum and maximum value of the resultant force acting on the femoral head along current activities.



**Figure 9:** The results of FEA to the subject with 86 kg (resultant force 1601 N): von Mises stresses fields and movements [1]



**Figure 10:** The results of FEA to the subject with 86 kg (resultant force 2580 N): von Mises stresses fields and movements

### 3. CONCLUSION

Method with finite elements analysis (FEA) has become a tool for working with a large use in all engineering fields. The main advantage of this method is to be removed from the potential barrier of the level of complexity of the problem being studied. Finite element analysis of implants is very important to the design and implementation of new types of prosthesis better than the classic ones.

### REFERENCES

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