Virtual and experimental device for hip revision prostheses testing

D L Popa¹, D C Calin² and G Buciu³

¹ Automotive, Transportation and Industrial Engineering Department, University of Craiova, Craiova, Romania
² Medicine Department, University of Medicine and Pharmacy of Craiova, Craiova, Romania
³ General Health Care Department, “Titu Maiorescu” University, Targu Jiu, Romania

E-mail: popadragoslaurentiu@yahoo.com; calindanielcosmin@yahoo.com

Abstract. Coxofemoral osteoarthritis is a common pathology, especially for the elderly or for certain social categories. One solution to this medical problem is the hip joint prosthesis. This, however, over time, can lead to joint damage, and obviously a revision of the prosthesis is required. The revision hip prosthesis presents additional mechanical elements and obviously different surgical techniques are required compared to the normal hip prosthesis. There are several methods of prosthesis and obviously more surgical techniques. We were interested to identify that revision method that retains as much bone material, and which has the same mechanical strength. A special device designed to test the revision hip prosthesis was designed and made using a cow’s pelvis as bone material. The orthopedic assembly is attached to the device and supposed to a progressive loading until it mechanically fails. The orthopedic system was analyzed with a microscope where the cracks that determined the mechanical failure of the orthopedic system were detected and measured. In parallel, the same device is loaded into a virtual test environment and subjected to a similar load. The actual results and those obtained in the virtual environment were compared and analyzed. Finally, interesting conclusions were drawn.

1. Introduction
The hip joint is of spheroidal type that allows rotations on the three axes of movement, having an important role in static and locomotion of the human body [1], [2], [3].

The pathology of the hip that requires a prosthesis shows a change in the joint morphology. Among the most common diseases in medical practice that have as a final therapeutic solution hip arthroplasty are primary and secondary coxarthrosis, followed by traumatic conditions, femoral fracture and pseudarthrosis of the femoral neck, the final stage of aseptic femoral head necrosis, as well as rheumatoid arthritis coxitis, or ankylosing spondylitis [2], [4], [5].

The treatment of hip pathology has evolved in the last two centuries, from rudimentary surgical procedures to modern hip arthroplasty, with a marked explosion in the last 30 years, considered one of the most successful operations to date. Hip arthroplasty is a permanent challenge due to the desire to discover the "supreme prosthesis", for which competition is still open [3], [6], [7], [8].

Endoprosthetic arthroplasty can be defined as reconstructive surgery with bone sacrifice and prosthetic replacement of the joint components [5], [9], [10].
Particular aspects appear, however, when restoring the hip prosthesis. In addition to the dislocation of the prosthesis, bone regressions in the acetabular area, enlargement of the contact area of the prosthesis cup, bone absences, cracking of the pelvic bone are noticeable [11]. For this reason, the surgical operation to restore the hip joint, in these special situations, is subject to additional challenges. Also, these interventions are usually performed in elderly patients, and for the restoration of the joint additional metallic elements of prosthesis are used [7], [12], [13], [14].

There are several methods of prosthesis and obviously more surgical techniques. We were interested to identify that revision method that retains as much bone material, and which has the same mechanical strength [9], [15], [16], [17].

In order to test an orthopedic system under the conditions of a laboratory of applied mechanics and of the resistance of the materials it was necessary to obtain such an assembly. For this, a fresh bone segment from a cow (5-6 hours) was used [18], [19]. This animal was used because the acetabular cavity is approximately the same size as the human pelvic bone and has similar mechanical properties. Also, revision prosthesis components were used for the hip prosthesis [20].

2. Performing a basic orthopedic assembly for revision of the hip prosthesis
A segment of the femur and pelvic bone was taken from the animal. Figure 1 shows these bone components that were analyzed and established the surgical procedures and performing the orthopedic assembly.

![Figure 1. Bone segments taken from the animal.](image1.jpg)

The prosthetic elements, but also the additional revision component that will form the assembly together with the bone component, are shown in Figure 2. The additional element will be fixed on the bone component by two orthopedic screws.

![Figure 2. Prosthetic metal elements.](image2.jpg)
In order to obtain the orthopedic fitting, the specific tools for orthopedic surgery were used. Some of these are shown in Figure 3 and are used for milling the acetabular cavity and adapting to the components of the prosthesis, but also for cutting and removal.

![Figure 3. Tools used to adapt the bone component to the prosthetic elements.](image)

Also, materials needed to obtain the prosthetic assembly, such as orthopedic cement and surgical gloves, as shown in Figure 4, were used.

![Figure 4. Additional materials used to obtain the orthopedic assembly.](image)

A first step was that in which the acetabular cavity was adapted to the size of the polyethylene cup of the main prosthesis. This operation was performed with a set of special mills, and the steps of this operation are shown in Figure 5.
Figure 5. Stages of adaptation of the acetabular cavity to the polyethylene cup of the prosthesis.

For the use of the additional prosthesis element, an additional spherical cavity was milled into the bone component. The stages of this operation are shown in Figure 6. Finally, the polyethylene cup and the additional revision element were positioned on the bone component.

Figure 6. Steps to obtain the additional cavity for the revision element.

3. Design and manufacturing of a fixing device for testing the orthopedic assembly
Prior to the completion of the orthopedic assembly, a fixing device was designed and modeled to allow experimental testing. This device will achieve the fixation of the orthopedic assembly on a universal testing machine. For this operation the SolidWorks CAD software was used, and the components are shown in Figure 7.
Figure 7. The virtual components of the fixing device.

These components, the elements of the prosthesis, the bone component, but also the components of the test machine (in blue) were assembled in SolidWorks and analyzed (Figure 8).

Figure 8. The final assembly of the virtual fixing device.

Finally, these components were manufactured from an aluminum alloy and the fixing device components as shown in Figure 9 were obtained.
4. Finalizing the orthopedic assembly and testing on the universal machine

In order to obtain the orthopedic assembly, was used an orthopedic cement which has two components, one basic and one hardener. The two components were mixed then the additional element was glued, then the polyethylene cup of the prosthesis. These steps are shown in Figure 10.
Because the working height of the test machine is limited to 27 cm, the orthopedic assembly had to be adjusted (cut), and the steps of this operation are shown in Figure 11.

For testing, the universal EDZ 20 test machine consisting of the main machine and the control panel as shown in Figure 12 was used.

Figure 13 shows the universal EDZ 20 test machine with the orthopedic assembly fixed on the lower table. The upper table is the fixed, and through the lower table the force is applied in the anatomical direction of application. The force was applied gradually, and the orthopedic assembly gave up the value of 450 kgf, ie 4414.5 N.
5. Testing the orthopedic assembly using the finite element method

To verify the orthopedic assembly in a virtual environment, using the finite element method, a simplified model was used that could be validated by the experimental results. Figure 14 shows the simplified model in SolidWorks and after exporting it to Ansys Workbench.

The materials that were attached to the geometry imported into Ansys have the properties of the real ones that were used in the experiment. Table 1 presents these materials for each component, as well as their mechanical properties [21], [22], [23]. Most were selected from the Ansys database known as Engineering Data.

| Component              | Material            | Density (kg/m3) | Young's Modulus (Pa) | Poisson's Ratio | Shear Modulus (Pa) |
|------------------------|---------------------|-----------------|----------------------|----------------|-------------------|
| Pelvic bone            | Bone                | 1400            | 1 E+10               | 0.3            | 8.3331 E+9        |
| Additional revision component | Titanium Alloy | 4620            | 9.6 E+10             | 0.36           | 1.142 E+11        |
| Polyethylene cup       | Polyethylene        | 950             | 1.1 E+9              | 0.42           | 2.291 E+9         |
| Femoral stem           | Stainless Steel     | 7750            | 1.93 E+11            | 0.31           | 1.693 E+11        |
| Revision metal cup     | Titanium Alloy      | 4620            | 9.6 E+10             | 0.36           | 1.142 E+11        |
| Spherical couple       | Stainless Steel     | 7750            | 1.93 E+11            | 0.31           | 1.693 E+11        |
The next step in defining the simulation parameters consists in the division into finite elements. The finite element used was the tetrahedron using the Relevance parameter at 100. A finite element structure was obtained with 138902 nodes and 772381 elements as shown in Figure 15.

![Figure 15. The structure of finite elements.](image)

The Explicit Dynamics module was used, the safe module that allows the analysis of some systems until the mechanical failure, as well as after this phenomenon. Due to the very short periods analyzed, a maximum analysis period of 0.1 sec was chosen. Also, an evolution of the force from 0 to $1.65 \times 10^8$ N was chosen, which is located at the base of the bone component as in Figure 16. The fixed element was considered to be the stem of the prosthesis, as it happened in the experiment.

![Figure 16. The displacement of the force and its evolution.](image)

At the analysis settings, 10,000 calculation cycles were chosen for the total duration of 0.1 sec. Also, the Erosion Controls parameter was chosen for the analysis to be performed On Material Failure. Such analyzes are large consumers of hardware and software resources, but also of time. Thus, using computer with i5 processor and 16 GB of RAM, the calculation took approximately 28 hours. The obtained results consisted of maps of stress, displacement and strain (Figure 17). Also, the simulation stopped at time $t = 0.00000273$ sec. when material failure occurred.
6. Discussions and conclusions

From the analysis of the cracks and the stress map for the bone component, it is found that this yield of the material occurs in the acetabular area upon contact with the additional revision element. The cracked area was analyzed with digital stereo microscope INSIZE ISM-PM200SB. Figure 18 shows two images obtained on the stress map, but also with the microscope on the bone component.

Figure 17. The stress, displacement and strain maps.

Figure 18. The cracks appeared on the virtual model and on the real model.
Given the evolution of the force in the virtual model, but also the moment at which the system yields, i.e. \( t = 0.00000273 \) sec., the force value can be calculated very quickly at the moment of virtual failure and \( F_v = 4504.5 \) N, compared to the value of the experimentally determined force \( F_e = 4414.5 \) N. Comparing the experimental force with the one obtained on the virtual model, we find an error \( e = 0.102\% \), which validates the virtual model and the simulation performed using the finite element method.

Also, the following conclusions were underlined:
- using CAD and FEM methods, very complicated biological systems can be modeled and simulated;
- the virtual model proposed by this work has been experimentally validated;
- methods of virtual prototyping coupled with virtual reconstruction from CT or MRI images and with rapid prototyping methods, open the way of innovation of customized orthopedic systems for each patient.

7. References

[1] Bîzdoacă N G, Tarniţă D N, Tarniţă D, Popa D L, Bîzdoacă E, 2008 Shape memory alloy based modular adaptive orthopedic implants Proceedings of the 1st WSEAS international conference on Biomedical electronics and biomedical informatics World Scientific and Engineering Academy and Society (WSEAS) pp 188-195

[2] Calin D C, Popa D L, Grecu A F 2020 Virtual Experimental Analyzes of the Normal and Arthrotic Hip Applied Mechanics and Materials 896 pp 3-14

[3] Ciunel S, Popa D L, Gherghina G, Bogdan M L, Tutunea D 2014 Human Head-Neck System Behavior During Virtual Impact Automotive Simulations Applied Mechanics and Materials 659 pp 177-182

[4] Jeli Z, Stojićević M, Cvetkovic I, Duta A, Popa D L 2017 A 3D analysis of geometrical factors and their influence on air flow around a satellite dish, FME Transactions 45(2) pp. 262-267

[5] Tarnita D, Berceanu C, Tarnita, C 2010 The three-dimensional printing–a modern technology used for biomedical prototypes Mater. Plast. 47(3) pp 328-334

[6] Popa D L, Duţă A, Tutunea D, Gherghina G, Buciu G, Calin D C 2016 Virtual Methods Applied to Human Bones and Joints Re-Construction Used for Orthopedic Systems Applied Mechanics and Materials 822 pp 160-165

[7] Tarnita D, Catana M, Tarnita D N 2016 Design and Simulation of an Orthotic Device for Patients with Osteoarthritis New Trends in Medical and Service Robots Springer Publishing House pp 61-77

[8] Kosić B, Stojićević M, Jeli Z, Popkonstantinović B, Duta A, Dragićević A 2019 3D analysis of different metamaterial geometry and simulation of metamaterial usage, FME Transactions 47(2) pp 349-354

[9] Tarnita D, Psla D, Geonea I, Vaida C, Catana M, Tarnita D N 2019 Static and Dynamic Analysis of Osteoarthritic and Orthotic Human Knee J Bionic Eng 16(3) pp 514-525

[10] Petrovici I L, Tenovici M C, Vaduva R C, Tarnita D N, Vintila G, Popa D L 2019 About Three-Dimensional Models of Osteosynthesis Systems Journal of Industrial Design and Engineering Graphics 14(1) pp 159-162

[11] Tarnita D, Popa D, Boborelu C, Dumitru N, Calafeteanu D, Tarnita D N 2015 Experimental bench used to test human elbow endoprosthesis New Trends in Mechanism and Machine Science Springer Cham pp 669-677

[12] Buciu G, Popa D L, Grecu D, Niculescu D, Nemes R 2012 Comparative analysis of the three new designs of tibial nails which eliminate the use of orthopedic screws Proceedings of The 4th International Conference “Advanced Composite Materials Engineering” COMAT 2012, Lux Libris Publishing House, Brasov, Romania pp 387-392

[13] Tarniţă D, Popa D, Tarniţă D N, Grecu D, Negru M 2006 The virtual model of the prosthetic tibial components Rom J Morphol Embryol 47(4) pp 339-344

[14] Tarnita D, Tarnita D N, Bizdoaca N, Popa D 2009 Contributions on the dynamic simulation of
the virtual model of the human knee joint Materialwissenschaft und Werkstofftechnik, Materials Science and Engineering Technology Special Edition Biomaterials Willey-Vch. 40(1-2) pp 73-81

[15] Vatu M, Vintila D, Mercut V, Popescu S M, Popa D L, Petrovici I L, Vintila G, Pitru A 2019 Three-dimensional modeling of the dental-maxillary system Journal of Industrial Design and Engineering Graphics 14(1) pp 207-210

[16] Tarnita D, Tarnita D N, Bizdoaca N, Tarnita C, Berceanu C, Boborelu C 2009 Modular adaptive bone plate for humerus bone osteosynthesis Rom J Morphol Embryol 50(3) pp 447-452

[17] Vatu M, Vintilă D, Popa D L, Mercut R, Popescu S M, Vintila G 2020 Determining Mechanical Causes that Produce Dental Wear Using Finite Element Method Applied Mechanics and Materials 896 pp 15-22

[18] Tarnita D, Tarnita D N, Hacman L, Copilusi C, Berceanu C, Cismaru F 2010 In vitro experiment of the modular orthopedic plate based on Nitinol, used for human radius bone fractures Rom J Morphol Embryol 51(2) pp 315-320

[19] Vatu M, Vintilă D, Popa D L, Mercut V, Popescu S M, Vintila G 2019 Simulations Using Finite Element Method Made on a Personalized Dental System Advanced Engineering Forum Trans Tech Publications Ltd 34 pp 175-182

[20] Tarnita D, Tarnita D N, Popa D, Grecu D, Niculescu D 2010 Numerical simulations of human tibia osteosynthesis using modular plates based on Nitinol staples Rom J Morphol Embryol 51(1) pp 145-150

[21] Laflamme M, Lamontagne J, Guidoin R 2013 Anterior cruciate ligament pros theses using biotextiles, Biotextiles as Medical Implants, M.W.K.S.G. Guidoin, Editor 2013, Woodhead Publishing pp 590-639

[22] Ambrosio L, Gloria A, Causa F, 2010 Composite materials for replacement of ligaments and tendons, Biomedical Composites L. Ambrosio Editor Woo head Publishing pp 234-254

[23] Woo S L, Abramowitch Y, Loh S D, Musahl J C, Wang J H 2003, Ligament healing: Present status and the future of functional tissue engineering, Functional Tissue Engineering, Springer-Verlag New York pp 17-20