Short Communication

BIOMEDICAL Journal of Scientific & Technical Research

ISSN: 2574 -1241

DOI: 10.26717/BJSTR.2022.43.006921

3-D Structural Simulation of a Hip Prosthesis

Trentadue B* and Ceddia M

Dipartimento di Meccanica, Matematicae Management, Politecnico di Bari, Italy

***Corresponding author:** Trentadue B, Dipartimento di Meccanica, Matematica e Management, Politecnico di Bari, Italy

ARTICLE INFO

Received: April 15, 2022

Published: May 02, 2022

Citation: Trentadue B, Ceddia M. 3-D Structural Simulation of a Hip Prosthesis. Biomed J Sci & Tech Res 43(4)-2022. BJSTR. MS.ID.006921.

ABSTRACT

This work aims to focus attention on partial or total prostheses obtained through digital imaging and EBM 3D printing, or on those prostheses made from the materials most used in current medicine. We will focus on the section of the acetabular cup, which represents the "heart" of the prosthesis itself, as it is the point where contact pressures arise, that is, those pressures and efforts that have led to failure and wear in the past. of prostheses. The research is performed for a representative case, in which the personalized hip prosthesis made using the 3D technique is used as an osteosarcoma treatment process. The article describes all the phases of the structural simulation by identifying a series of load scenarios for the biomechanical system consisting of the pelvis and endoprosthesis. The prototyping process is also the preparation of the finite element model using INVENTOR software of the pelvic bones and implant components will be explained later.

Introduction

In recent times [1] there has been an evolution in the study of pelvis prostheses, which has led to the latter being considered one of the main successes in the orthopedic sector. The reasons for the hip are for diseases such as osteoarthritis and other types of degenerative diseases from cartilage destruction and bone necrosis. Currently, a cause of the significant development of additive manufacturing technologies, based on metal powders such as titanium oxides, 3-D printing of endoprostheses has become very common. These technologies make it possible to use computed tomography data to generate the bone model, then the 3d model (Figure 1) of the model is created using software and then the behavior is simulated by means of FEM analysis. In the following article, the preparation process of the finite element model, the execution of the simulation and the execution of the analysis of the results will be explained, with any proposals for topological optimization of the prosthesis.

Analysis of Study

Rapid prototyping is a promising technique capable of reproducing three-dimensional physical structures with very high precision. Starting from CT images (Figure 1) it is possible to create, in a short time, perfect cutting masks and a custom-made prosthesis in trabecular titanium that can be perfectly superimposed on the area to be replaced. Preoperative planning is a process to be performed in a team, in which the surgeon must interface closely with the designer. Numerical simulations are then performed based on the patient's computed tomography and digital models of the implant. These simulations are widely used in modern orthopedics to analyze the stress-deformation state of large joint prostheses [2], allowing to obtain a distribution of stress and deformation in the implant and bone tissue and to evaluate their resistance. On the basis of the prepared model, the mechanical strength is evaluated, and conclusions are drawn relating to the possible topological

 DOI: 10.26717/BJSTR.2022.43.006921

optimization of the system. The physiological load condition considered is that in which the person is standing. It was decided to apply a load equal to 245 N [3] (half the patient's weight), applied to the center of gravity of the body located in the symmetry plane of the human being. But from the point of view of the calculation, this load is placed in the region of the sacrum. The solver used is Inventor, the relative solution was carried out starting from the hypothesis that the displacements are small in order to linearize the problem allowing a correct description of the contact behavior.

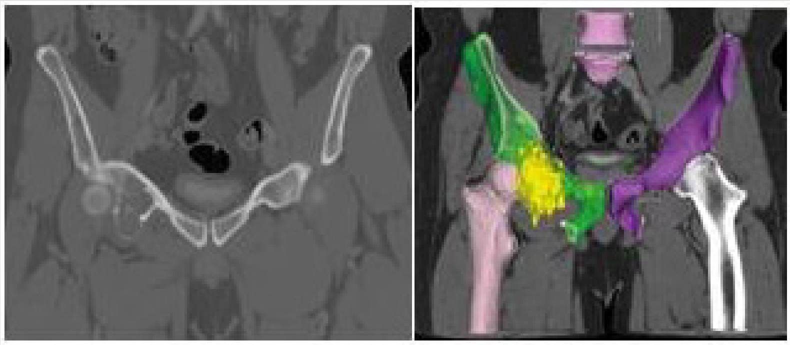

Figure 1: Coronal cut of CT and MRI in a patient suffering from a low-grade chondrosarcoma affecting the right acetabulum and the public branch. The 3D model of the pelvis and the neoplastic component (in yellow).

Preparing the Finite Element Model

The final goal of the project is to develop a rapid virtual test methodology for customized implants based on the finite element analysis of the endoprosthesis stress-deformation state [4]. The first step in performing the simulation is the preparation of the geometric model based on the tomographic data (Figure 2). The pelvic bone material is then modeled taking into account that it is composed of cortical and spongy layers with different properties. The external cortical layer is made up of a rigid material, the internal spongy one less resistant than the first, therefore more flexible and yielding. The mesh near the holes will be refined to increase the accuracy of the results, as well as the Fem analysis will be performed with the INVENTOR software.

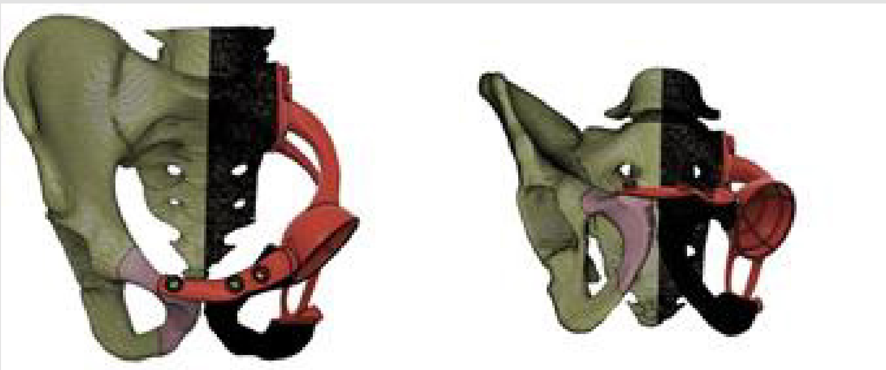

Figure 2: 3-D rendering of the model.

Constraint and Load Conditions

Once the model was built, we moved on to the mesh phase and the introduction of its boundary conditions. As regards the latter, the constraint conditions and the load conditions have been defined. Due to the constraints, it was decided to ensure that the prosthetic implant remained locked in the initial position, with no more possibility of translation or rotation. The exchange of stresses between prosthesis and bone can lead to favorable or unfavorable events (success or failure). The mechanical stresses exchanged are due to the fact that the prosthesis is inserted in the lower limbs, naturally subjected to high loads. The prosthesis must transmit the loads to the bone and, to avoid resorption, a correct range of load and therefore of deformation on the bone is required. If the deformation transmitted is below the range, because the prosthesis is too rigid, the bone undergoes resorption; if it is too high, the bone can break because it is too stressed. To do this, the prosthesis was constrained

DOI: 10.26717/BJSTR.2022.43.006921

by selecting the external area of the anchor points and inserting the constraints that prevent the movement of the nodes along all three reference axes (Figure 3). After inserting the constraints, we moved on to the "creation" of the load conditions (Figures 4-6), using the specifications obtained from the bibliography and applying the loads with intensity and direction as follows. Once the boundary conditions were completed, the mesh was created (Figure 7).

Mesh size = 0.100

Below is the model with mesh.

Figure 3: constraint on the top face.

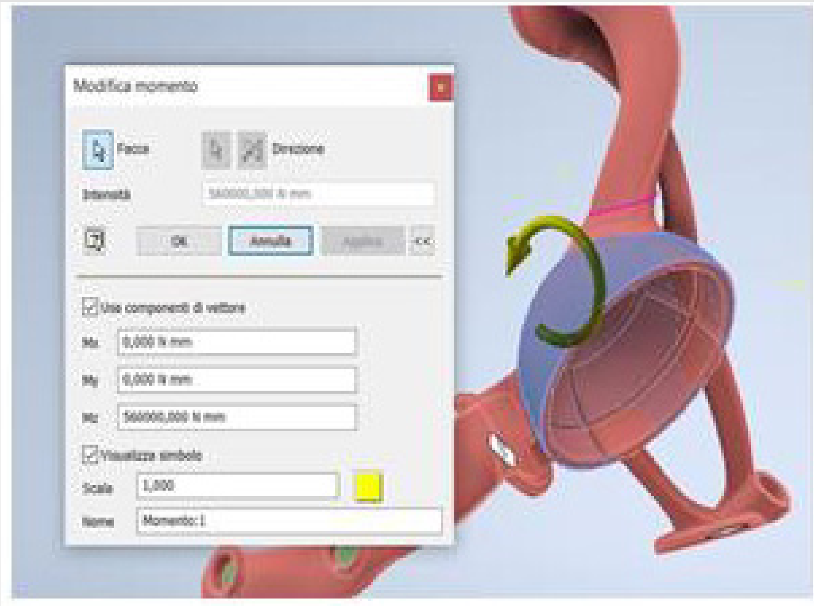

Figure 4: Moment on acetabular cup for 56mm offset.

Figure 5: Load on the acetabular cup.

Figure 6: Overall loads.

Figure 7: Mesh of model.

Materials

In order to make the choice of the most suitable material for the construction of the prosthesis, we were based on 3 parameters:

1. Biocompatibility
2. Mechanical response very similar to bones.
3. Costs

From the point of view of biocompatibility, it is satisfied when the material is an inert substance towards the organism (Table

 DOI: 10.26717/BJSTR.2022.43.006921

1). Considering the pharmacological situation incorporated into a living system, also a mechanical response must be compared with the mechanical characteristics of the bones. Bone is a ductile and fragile material [5] at the same time: the graph obtained experimentally shows an initial linear elastic region, in which small forces applied are sufficient to determine large deformations of the bone, followed by a plateau of the curve with almost constant stress up to a fracture (ductile behavior). The implant screws and the prosthesis itself will be made of Ti6Al4 titanium alloy [6]. The following table summarizes the data relating to the materials analyzed in the following work.

Table 1: Mechanical properties of the materials.

Material name	Density, kg/m^3	Elastic modulus, MPa	Poisson ratio	Critical stress, MPa
Spongy bone	1 188	500	0.2	10
Cortical bone	1 470	10 000	0.3	160
Titanium alloy	4 410	110 000	0.3	1065

Results

Particular attention was paid to estimating the correct pretension force of the screw between implant and bone, since a high pretension force can cause bone breakage. Subsequently, through the simulations, the material that better respected the mechanical conditions of the acetabular bone was chosen, obtaining from the first simulations shown in the Figure 8 that, the most suitable material for implantology is grade 23 titanium, since there are deformations and higher safety factors and as close as possible to bone tissue. Despite the use of this material, there are serious shortcomings in resistance and limitation to deformation which will be subject to subsequent amendments.

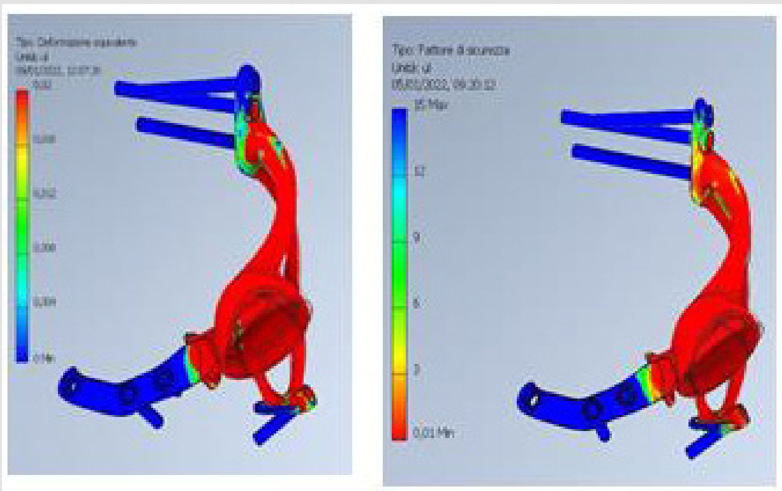

Figure 8: Deformation and stress.

Conclusion

Engineering tools such as finite elements (FE) offer enormous potential, allowing studies on inaccessible locations and encouraging further reflections to direct the production and clinical management of increasingly customizable prostheses. However, it should be borne in mind that the results depend on the input parameters and that caution is therefore mandatory for interpretation, because these models are a simplification of the physiological conditions. In this work we started by analyzing what are the causes that lead to surgery to replace the hemibacin and we saw how they could depend on oncological situations or traumatic events. Subsequently, the state of the art on prostheses and prosthetic interventions was analyzed, in order to identify the parameters that could influence the design of a possible prosthesis. At first, the first total prosthesis model was created by creating the anchor points, the various branches and the acetabular cup section.

Once the model was obtained, the first simulations were carried out with the parameters mentioned above, the results (Figure 8) of which were compared with those taken from the state of the art and also with each other, to verify which material could provide an answer as close as possible to the natural one of the human bodies. From the analysis of the results, it was noted that materials with lower stiffness led to higher stresses and lower safety coefficients. The analysis of these led to the exclusion of high-density polyethylene as a possible solution to reduce costs and guided subsequent changes [7]. Finally, a second model was produced working on the basis of the first and intervening where the first had deficiencies without however affecting the reciprocal position between the acetabular cup and the fastening elements. Consequently, new simulations were carried out on the second model, without changing the load conditions and the constraints, which showed how the model reacted in an optimal way, providing new margins for improvement from a structural and economic point of view. Finally, an initial optimization process of the sections was carried out to provide an idea of the possible changes to be made in conjunction with the clinical evaluation of the latter. In conclusion, this study is believed to have inaugurated new foundations for further room for maneuver for subsequent studies. Possible growth trajectories, for example, could involve the addition of new parameters to be considered, such as an analysis of the contact pressures between the femoral head and the acetabulum in order to improve the loads on the model. In this regard, a challenge for this type of study could be to build a model that is no longer static but dynamic that can simulate and perform contact analysis under dynamic conditions.

References

1. Steven M Kurtz, Kevin L Ong, Edmund Lau, Marcel Widmer, Milka Maravic, et al. (2011) International survey of primary and revision total knee replacement. International Orthopedics (SICOT) 35(12): 1783-1789.
2. Kluess D (2010) Orthopaedic Biomechanics 2010. Finite Element Analysis, pp. 151-170.
3. Xishi Wang, Tianying Wang, Fuchuan Jiang, Yixiang Duan (2005) The hip stress level analysis for human routine activities. Biomedical Engineering Journal, p. 17.
4. Liang H, Ji T, Zhang Y, Wang Y, Guo W (2017) Reconstruction with 3D-printed pelvic endoprostheses after resection of a pelvic tumour. Bone Joint J 99 -B (2): 267-275.
5. Zhou Y, Min L, Liu Y, Shi R, Zhang W, et al. (2013) Finite element analysis of the pelvis after modular hemipelvic endoprosthesis reconstruction", International Orthopaedics (SICOT) 37(4): 653-658.
6. Shah F A, Snis A, Matic A, Thomsen P, Palmquist A (2016) 3D printed Ti6Al4V implant surface promotes bone maturation and retains a higher density of less aged osteocytes at the bone-implant interface. Acta Biomater 30: 357-367.
7. Borovkov AI, Maslov LB, Zhmaylo MA, Zelinskiy IA, Voinov IB (2018) Russian Journal of Biomechanics 24(4): 382-400.

ISSN: 2574-1241

DOI: 10.26717/BJSTR.2022.43.006921

Trentadue B. Biomed J Sci & Tech Res