Short Communication

BIOMEDICAL
Journal of Scientific & Technical Research

ISSN: 2574 -1241

DOI: 10.26717/BJSTR.2023.48.007598

Design and Optimization of an Artificial Hip Joint by Finite Element Analysis

Bartolomeo Trentadue*, Mario Ceddia and Caterina Callea

DMMM – Politecnico Di Bari, Italy

***Corresponding author:** Bartolomeo Trentadue, DMMM - Politecnico Di Bari, Italy

ARTICLE INFO

Received: December 13, 2022

Published: January 13, 2023

Citation: Bartolomeo Trentadue, Mario Ceddia and Caterina Callea. Design and Optimization of an Artificial Hip Joint by Finite Element Analysis. Biomed J Sci & Tech Res 48(1)-2023. BJSTR. MS.ID.007598.

ABSTRACT

Titanium is one of the widely used hip joint implant materials. Even with excellent properties for hip joint implantation, this material is likely to fail after 12-15 years of implantation due to excessive wear and stress. Computational analysis using the finite element method can be used to analyze stress in the hip joint implant. The study aims to evaluate stresses and analysis of safety factors. The analysis concludes by introducing an optimized prosthesis, evaluating the stress, and comparing these results with the classical prosthesis.

Introduction

One of the most critical joints in the human body is the hip joints, which connect the femur to the pelvis. This joint can deteriorate with time for many problems; The main problem is osteoarthritis [1,2]. Other reasons for hip joint impairment are atrophic arthritis and avascular necrosis [3]. Hip joint replacement called total hip arthroplasty is the main orthopedic surgery with over 350,000 and 60,000 surgeries performed each year in the United States and the United Kingdom [4,5]. It is not surprising that many people need this surgery because the hip joint has the function of supporting the weight of the upper body during many activities that can reach up to 4 times the body weight. The artificial hip joint consists of two main parts. The first is the acetabular component, which replaces the acetabulum in the pelvis.

The second is the femoral component, such as the stem and head of the sphere placed in place of the femoral head. Many widely used hip joint implant materials can be divided into several pairs: metal to metal, ceramic to ceramic, polymer to ceramic, and metal to polymer. Titanium has been widely used as a metal implant material because of its mechanical properties, corrosion resistance and non-magnetic properties that meet the minimum criteria as an implant material. 15 Even with its good properties, researchers have stated that hip implantation would degrade its function and fail within 12 to 15 years [4]. There is a computational analysis that uses the finite element method that can be used to calculate stress in the stem of the hip joint implant. Using Inventor Professional static structural software, this analysis can be performed from the previous study for different areas of hip replacement cross-section.

Materials and Methods

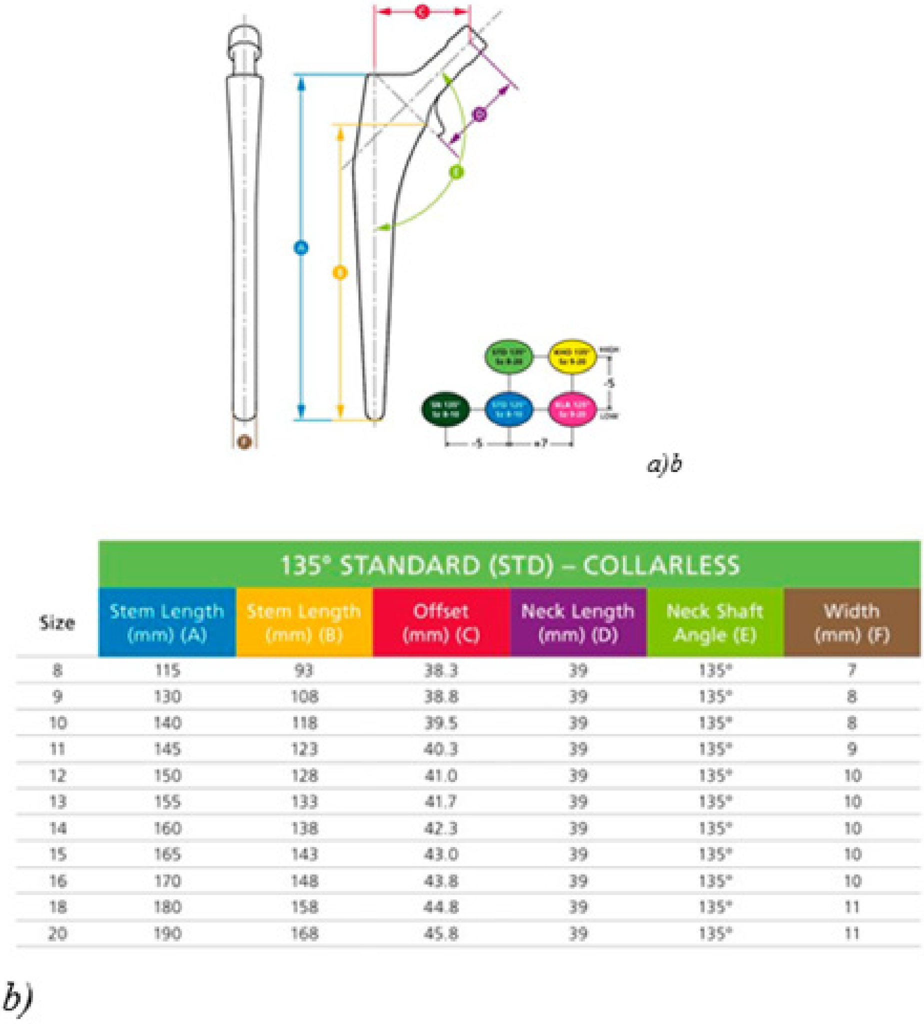
The geometric analysis in this study uses the design of the standardized Corail system, referring to a “collarless” model, with an angle between the axis of the stem and the axis of the neck of 135° (Figure 1). A part file “Standard.ipt” was created, using the command “2D Sketch” the longitudinal section of the workpiece was realized. Once the sketch was completed, a symmetrical extrusion was performed with respect to the work surface. Finally, the three most important transverse edges were connected so that the mechanical response was partially improved. It is important to specify that the end part of the package, i.e., the part that is inserted inside

DOI: 10.26717/BJSTR.2023.48.007598

the head, was given a variable rectangular section, because in the face of numerous load tests on the assembly, it was found that a coupling of this type was the most optimal (Figure 2). The material used in this study is Titanium grade 4 [6] in which the mechanical characteristics are shown in Table 1.

Table 1: Mechanical properties of Titanium grade 4.

Composition		Yield Strength (MPa)	UTS (MPa)	Elongation (%)
Ti	Grade 1	170	240	24
	Grade 2	275	345	20
	Grade 3	380	450	18
	Grade 4	485	550	15



Size	135° STANDARD (STD) – COLLARLESS					
	Stem Length (mm) (A)	Stem Length (mm) (B)	Offset (mm) (C)	Neck Length (mm) (D)	Neck Shaft Angle (E)	Width (mm) (F)
8	115	93	38.3	39	135°	7
9	130	108	38.8	39	135°	8
10	140	118	39.5	39	135°	8
11	145	123	40.3	39	135°	9
12	150	128	41.0	39	135°	10
13	155	133	41.7	39	135°	10
14	160	138	42.3	39	135°	10
15	165	143	43.0	39	135°	10
16	170	148	43.8	39	135°	10
18	180	158	44.8	39	135°	11
20	190	168	45.8	39	135°	11

b)

Figure 1:
(a) Standardized geometries and
(b) values of the "collarless" Corail prosthesis.

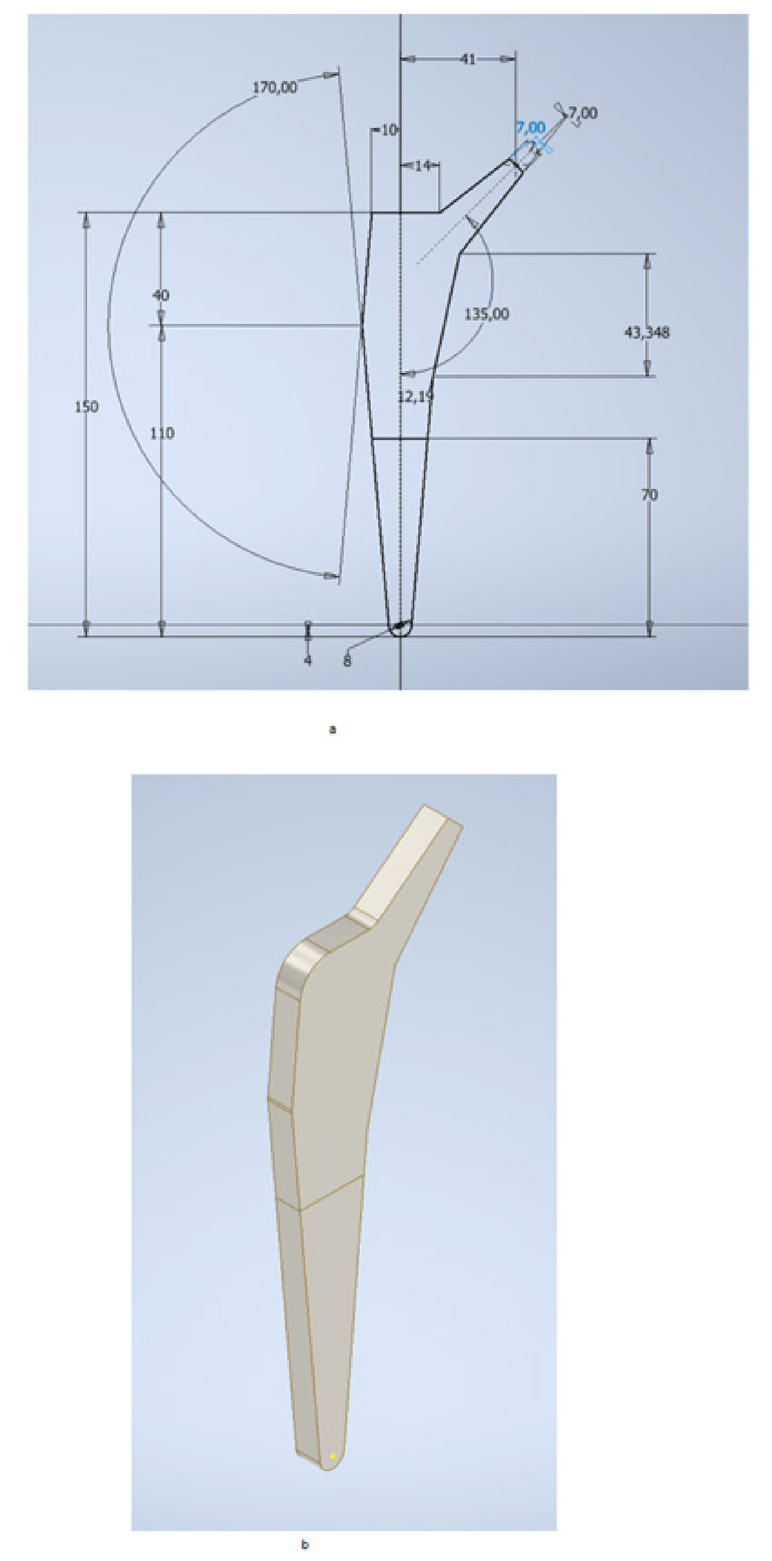


Figure 2:
(a) 2D model and
(b) 3D model of Hip Joint.

Loads

The loads used in the analysis, based on what is written in ISO 7206, on the distribution of forces of Charnley's theory and on what is available in the literature are:

Constraints

For the constraints we followed what is written in the ISO 7206 standard (Figure 3). For simulation convergence, the optimal mesh size of the stem is achieved by varying from 5 mm to 1 mm mesh size. He concludes that from 5 mm to 3 mm mesh size, the maximum Von-Mises stresses are likely to increase significantly. So, the mesh size from 3 mm to 1 mm shows that the maximum von-Mises stresses are stable. Therefore, the mesh size of 3 mm is used in this study for a stable, accurate and fast simulation. The total elements and knots of the mesh size of 3 mm are 10,917 and 6,652 respectively. The size of the element at most of the von-Mises stress variation graph can be seen in Figure 4.

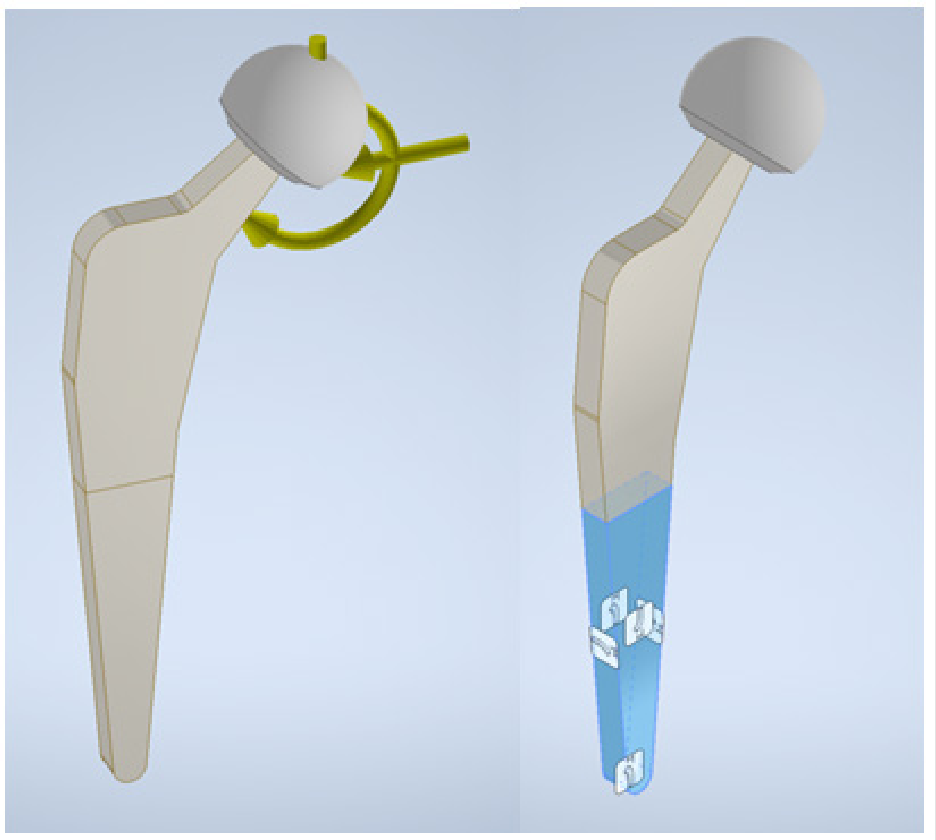


Figure 3: Configuration of loads and constraints.

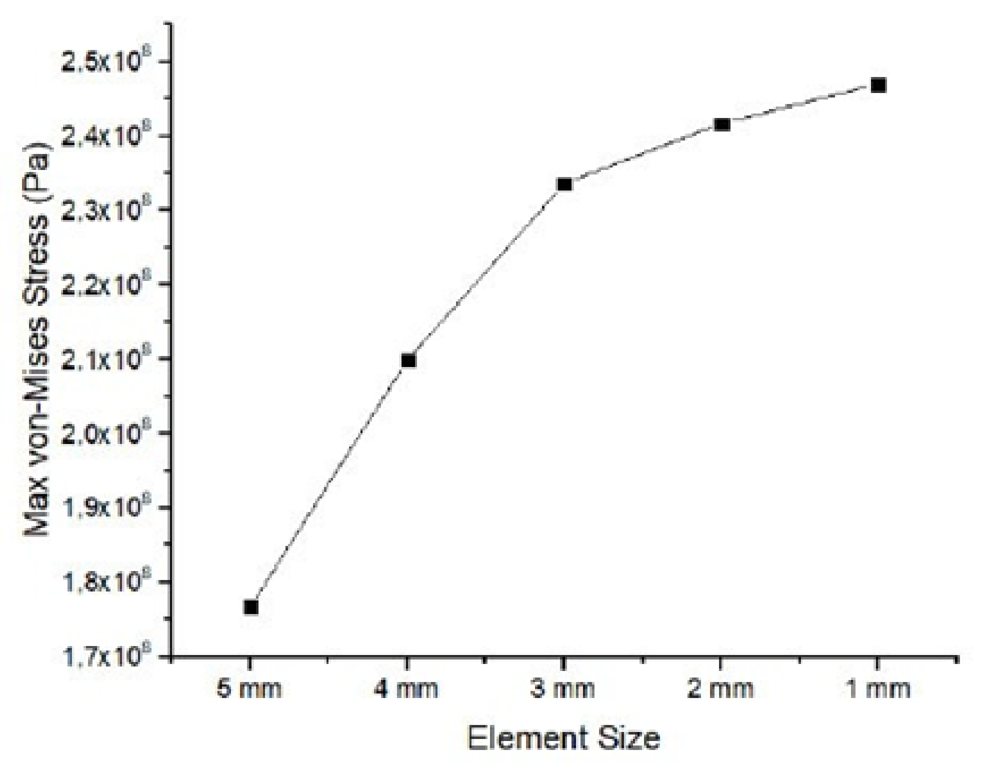


Figure 4: Stress variation with element size.

Results

After defining all the boundary conditions, it was possible to proceed with FEM analysis [7]. In line with what emerged from the qualitative analysis and the verification of the sections, the most stressed areas are those at the fixing and neck. The software allows you to identify the point where there is the greatest Von Mises stress, in this case it is near the constraint and with a value. This value is compared with the yield strength of Titanium grade 5, or (Figure 5) It is evident that the maximum stress is far lower than the elastic limit of the material, this is very important because for higher values it would enter the plastic field causing permanent deformations on the stem. These, also considering the fatigue stresses to which a hip prosthesis is subjected, over time could propagate to cause a definitive rupture, and consequently significant damage in patients. The simulation also shows that the internal areas on the side faces of the stem and the head are subject to very low tensions, the optimization will take place based on these results. From the verification of the sections, however, it emerged that the neck area is also a critical area, for completeness therefore, through the command “Investigation point” we wanted to verify the Von Mises stress, which shows high values but always much lower than the yield strength (Figure 6).

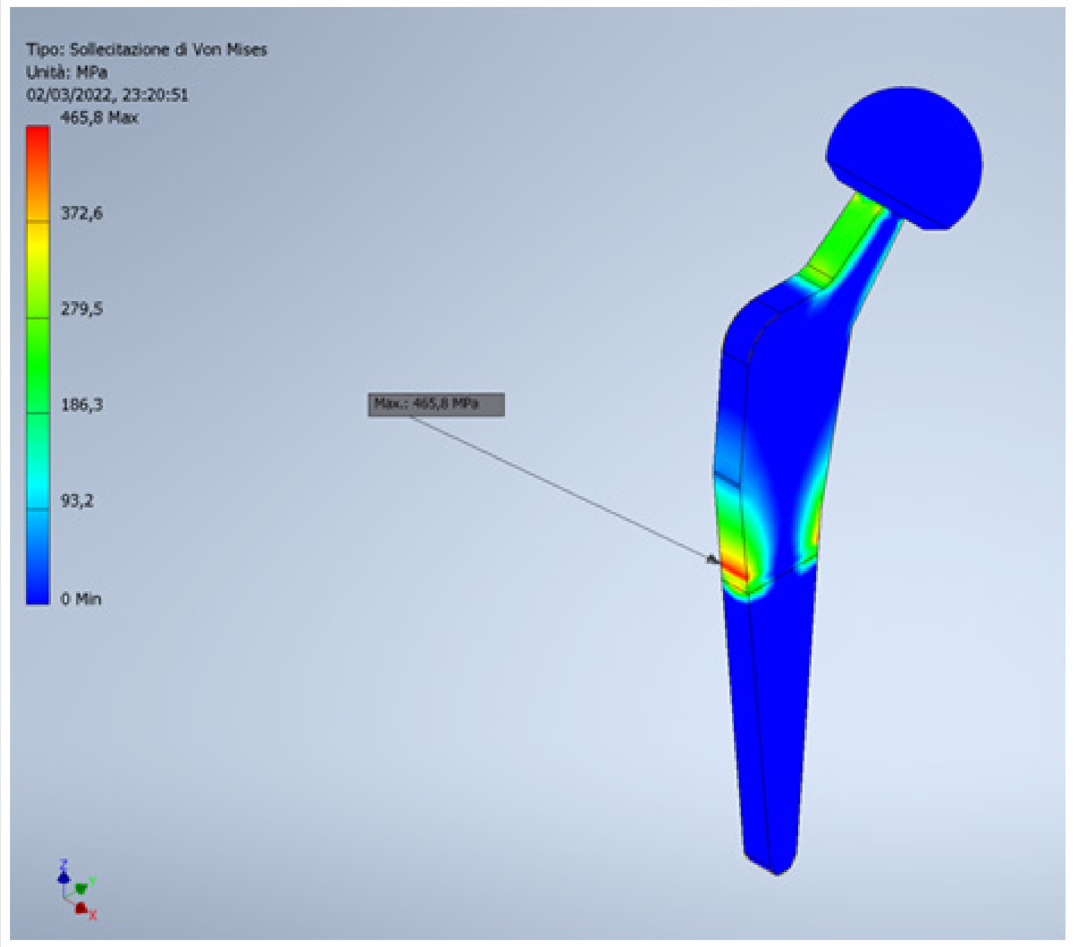


Figure 5: Von Mises stress on the stem.

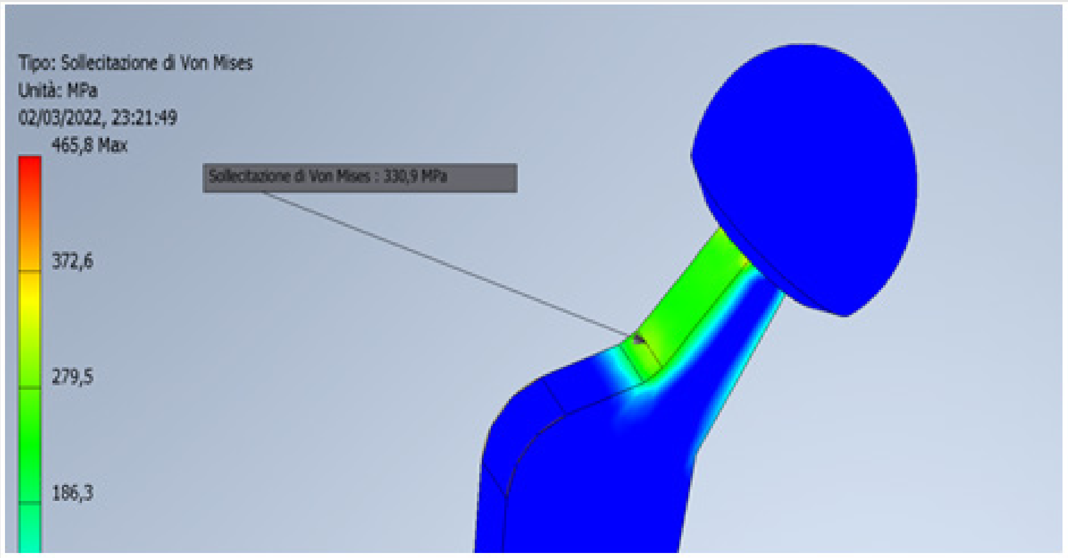


Figure 6: Von Mises stress on the neck.

DOI: 10.26717/BJSTR.2023.48.007598

Optimization

Since the beginning of the twenty-first century, increasingly innovative systems have been developed in terms of design and increasingly biocompatible with the body. Below is the stem following a first optimization Figure 7 At this point the stress analysis was repeated, keeping the boundary conditions constant, to verify that the optimization is advantageous both in terms of costs and in terms of safety. In this case the maximum Von Mises stress has a value of 406.6 MPa compared to 465.8 MPa of the non-optimized prosthesis. Considering these results, optimization is beneficial both in terms of cost and safety. The stress at the windows was also analyzed to verify that there were no risks and here too the voltage values are very low. In addition, the total mass of the prosthesis increased from 0.265 kg to 0.222 kg.

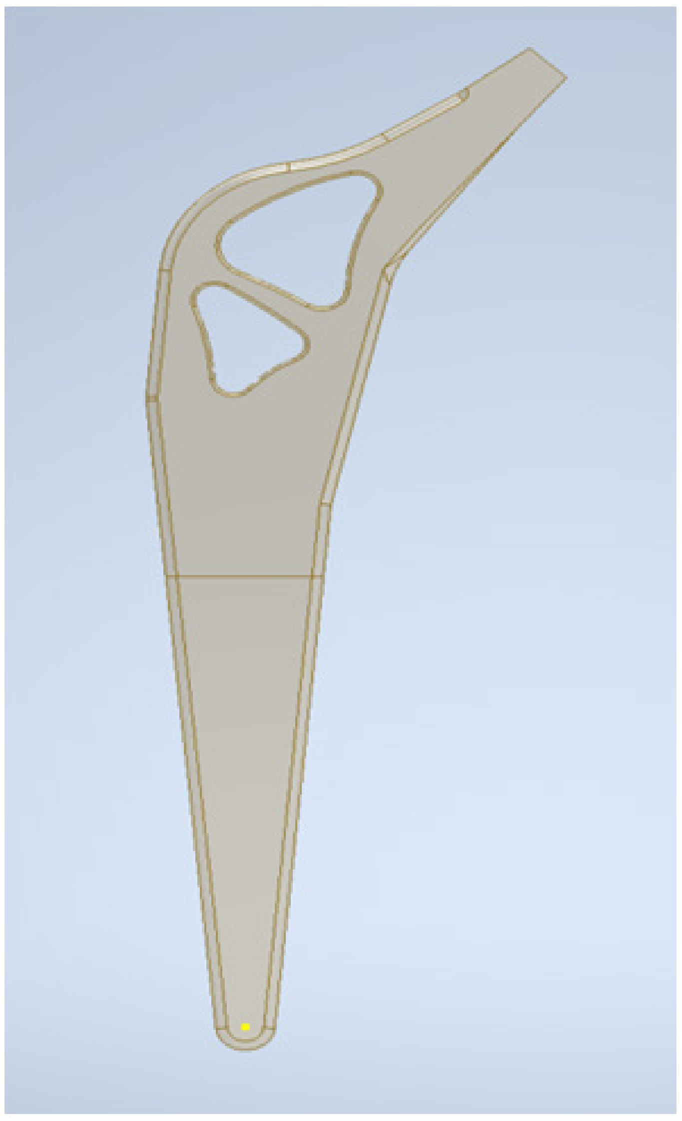


Figure 7: Optimized 3D stem model.

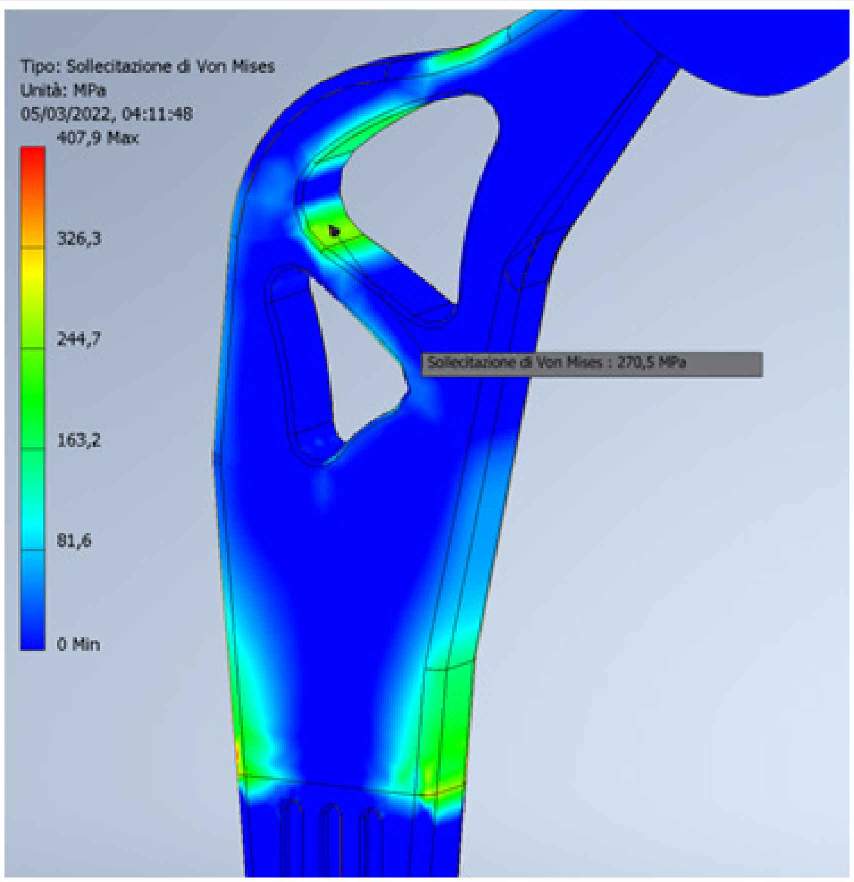


Figure 8: Stress analysis of the optimized prosthesis.

Figure 8 The present thesis work aimed at the topological optimization of a prosthetic stem through FEM analysis, performed on the Autodesk Inventor 3D CAD modeling software. We started from a stem without fittings and windows, an analysis of the stresses on the prosthesis was carried out and the results were analyzed. Based on the value of the Von Mises stresses at various points, a custom prosthesis was designed. The stem was connected along the side edges, resized and then two windows were made. To verify the correct success of the optimization, an FEM [8] analysis was again carried out on the optimized prosthesis that showed how the new model is more advantageous in terms of costs and mechanical response. For any future developments, the study could be conducted in a similar way to what has been done in this paper by studying the mechanical response due to other more biocompatible and less expensive materials, or even continuing the optimization of this stem looking for more advantageous configurations for both the manufacturer and the patient.

References

1. Merola M, Affatato S (2019) Materials for Hip Prostheses: A Review of Wear and Loading Considerations. Materials (Basel) 12(3): 495.
2. Ali M, Brogren E, Atroshi I (2020) Assessment of a novel computer software in diagnosing radiocarpal teoarthritis on plain radiographs of patients with previous distal radius fracture. Osteoarthr Cartil Open 2(4): 100112.
3. Al-Sanea A, Eltayeb M, Kumar NN (2018) Simulation and Analysis of Artificia Hip Joint Using Software Modeling. Int Conf Comput Control Electr Electron Eng (ICCCEEE).
4. Catauro M, Papale F, Sapio L, Naviglio S (2016) Biological influence of Ca/P ratio on calcium phosphate coatings by sol-gel processing. Mater Sci Eng C 65: 188-193.
5. Taqriban RB, Ismail R, Jamari J, Bayuseno AP (2020) Computational Analysis of Differen Designed Hip Joint Prostheses Using Finite Element Method. in 2020 7th International Conference on Information Technology, Computer, and Electrical Engineering (ICITACEE). 2020: 164-168.
6. Nardone I (2018-2019) Characterization and optimization of heat treatments on the Ti-6Al-4V alloy produced for Electron Beam Melting (EBM) Thesis. Master's degree (Politecnico di Torino).
7. Finite element modeling of hip implant static loading. International Conference on Manuufacturing Engineering and Materials.
8. Hughes TJR (1987) The finite element method. Linear static and dynamic finite element analysis. Thomas JR Hughes CMAM E, 65(2).

ISSN: 2574-1241

DOI: 10.26717/BJSTR.2023.48.007598

Bartolomeo Trentadue. Biomed J Sci & Tech Res