Guilherme Coutinho

M00684745

Finite Element Analysis of Femur Prosthesis

Introduction

Hip replacement is one of the most well-known surgeries and a key issue in orthopedic biomechanics. The total hip prosthesis comprises of an acetabular part that replaces the hip bone socket and a femoral part that replaces the femoral head. The goal of this prothesis is to reestablish the typical capacity of the joint.

The prosthesis life can be anticipated by considering key attributes of undertaking activities such as the response to forces in complex geometric shape areas. The study of this give us the amount of stress exerted and the displacement that might be in cause. This is important because we can try to predict the response of the prosthesis to certain conditions that cause tiny displacement or even cracks in response to the biomechanical system. However, these calculations are complicated and very time consuming.

One powerful answer for this issue is direct finite element (FE) modelling. The technique is used in engineering and mathematics and it solves a problem dividing a large system into smaller, simpler parts that are called finite elements. The need to consider numerous equations for the computational area brings about a multimillion arrangement of numerical conditions, which must be taken care by a computer.

The Finite element analysis (FEA) is another computational method that has been effectively utilized since the 1970s. The basic mathematical principal behind Finite element analysis (FEA) is based on the finite elements (FE) that splits volume or an area into several smaller parts or elements. FEA then, by using Hookes's law, it calculates the stress, strain and reaction forces for each individual FE.

In this paper we will model three prothesis with different length stems and using FE and FEA methods, study and compare the different stress caused by the biomechanics of the body.

Explanation of the finite element analysis

To briefly explain this method, let's look at the FE in one dimension. Each FE consists of net of nodes that use a relationship between displacement and force. F1 and f2 represent a node in a spring were *u* is the displacements of each node[i].

f1 = K (u1 – u2)   f3 =K (u2 – u3)

f2 =K (u2 – u1)

If now two strings are interconnected the first element (k1) is bound by the nodes 1 and 2 and the second (k2) by nodes 2 and 3.

Now to measure the elasticity we apply Hookes's law and to be calculated in a computer we put it in a matrix form.

$$\frac{f1}{f2} = \begin{pmatrix} k1 & -k1 \\ -k1 & k1 \end{pmatrix} \frac{u1}{u2} \text{ and } \frac{f2}{f3} = \begin{pmatrix} k2 & -k2 \\ -k2 & k2 \end{pmatrix} \frac{u2}{u3}$$

Note that there is a static equilibrium at every internal node f2=f12+f23 = 0 (figure 2)
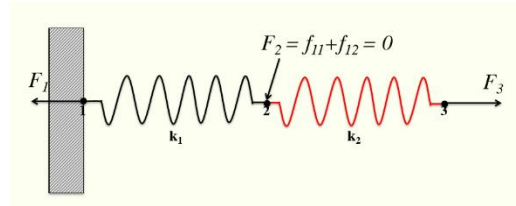
Figure 1: Static equilibrium

Integrate this into a larger matrix and now we can solve the displacement for multiple springs (figure 3)

$$\begin{pmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{pmatrix} = \begin{pmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & (k_1 + k_2) & -k_2 & 0 \\ 0 & -k_2 & (k_2 + k_3) & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{pmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{pmatrix}$$

Figure 2: Matrix of multiple springs

Most design work a component will be modelled by using two or three-dimensional elements.

To analyze stress the component must analyze the simultaneously complex system of stresses that are apply to each FE. This action is important, to investigate if the component is likely to fail and if at any point the stress exceeds the yield stress. There are different methods to analyze the maximum stresses, however the most accepted one is Von Mises.

$$(\sigma 1 - \sigma 3)^2 + (\sigma 2 - \sigma 3)^2 + (\sigma 3 - \sigma 1)^2 = 2\sigma^2$$

Von Mises – based on shear strain theory

However, Von Mises has some limitations. The main issues with Von Mises Criterion are that does not limit the hydrostatic, (same load in all directions in same time), compressions and tractions. This method assumes that the material used have symmetric behavior in traction and compression, making it difficult to study in case of anisotropic materials such as bones.

Materials and Methods

The modellings and studies were procedure in the software *Solidworks.* To achieve the design of the stems I imported a standard image of a stem sketch[ii] to the program and using the sketch tools, I draw the shape of the model giving parameters to establish a proper fit inside the femur. Figure 4 shows the parameters used for the model.
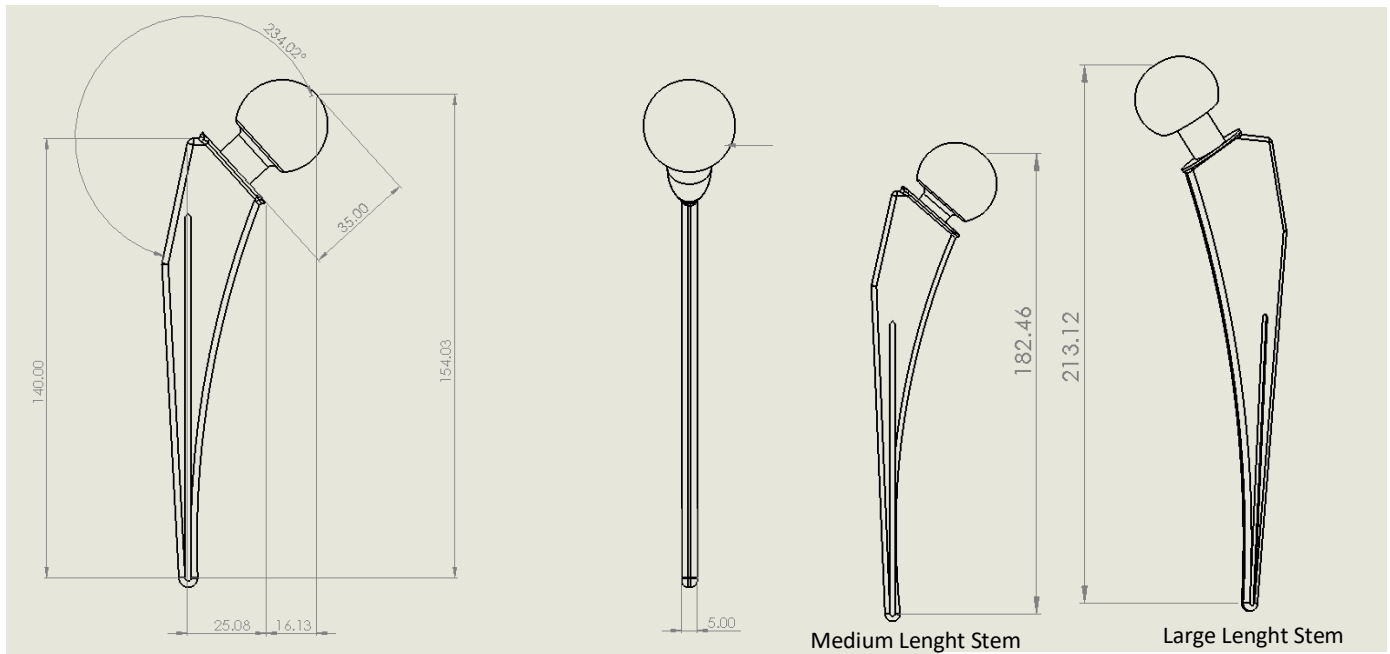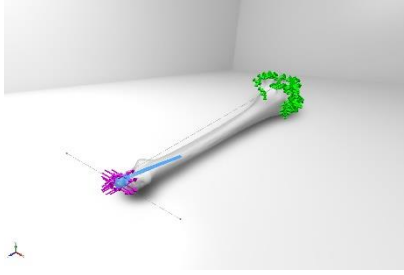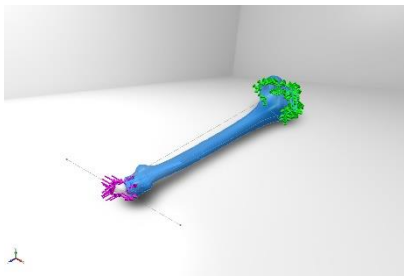

Figure 3:Sketch design of the control stem

On the left side the stem displayed is the Stem Control. It has 155 mm height by 41.20 mm length, and a thickness of 5 mm. The acetabular head is 15 mm of radius with an offset of 35 mm. On the right the stems have the same parameters, but different lengths of 185 mm and 215 mm, respectively.

To understand better the FEA study is important to first understand the material used, the forces and the different length that existed. For the Stem the material used was titanium taken from software data base. However, for the femur *Solidworks* does not included bone material, so a custom material had to be generated based on outside studies[iii]. Table 1 shows the properties of the materials used.
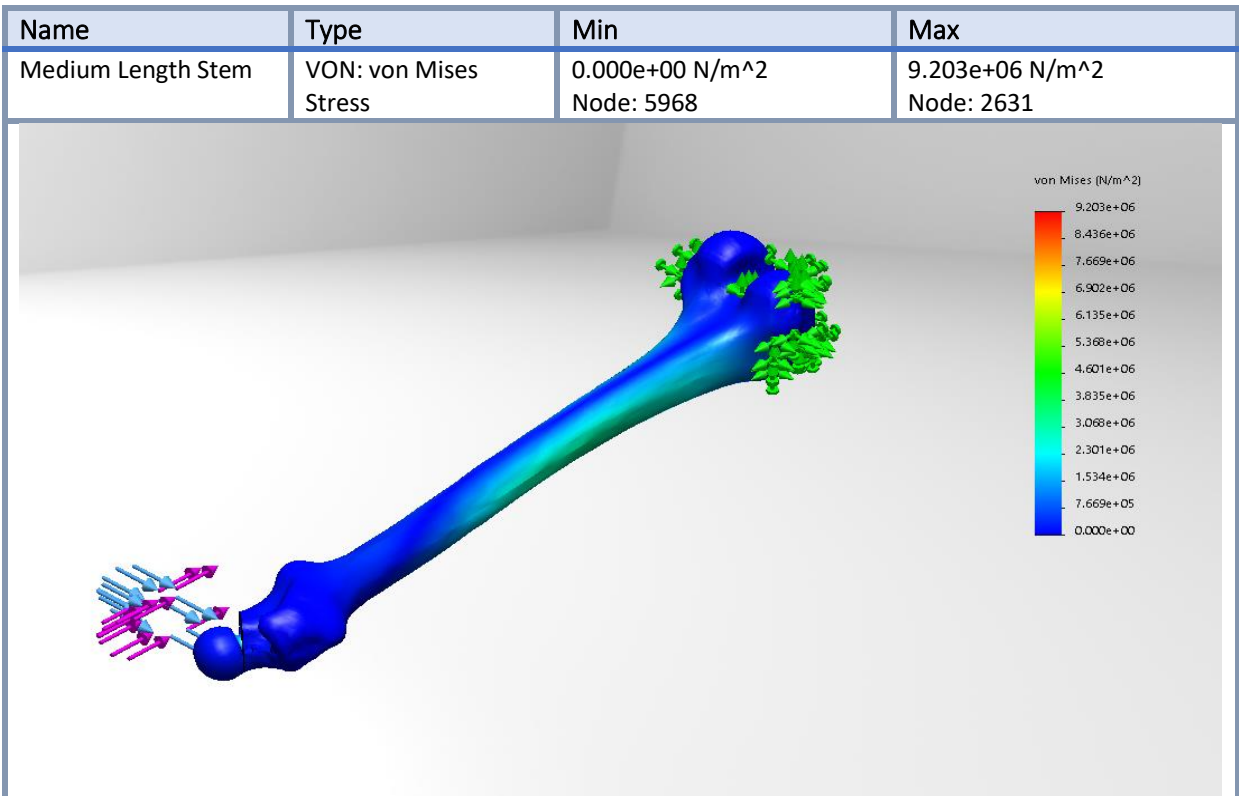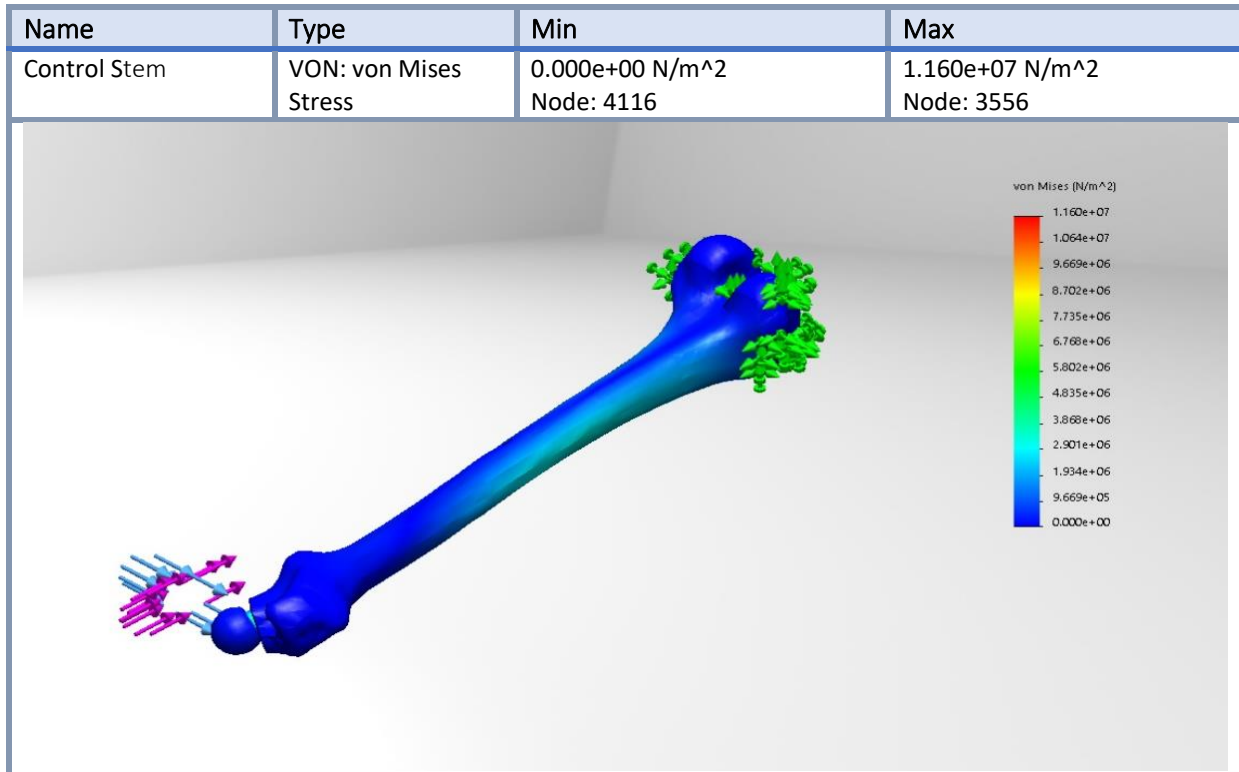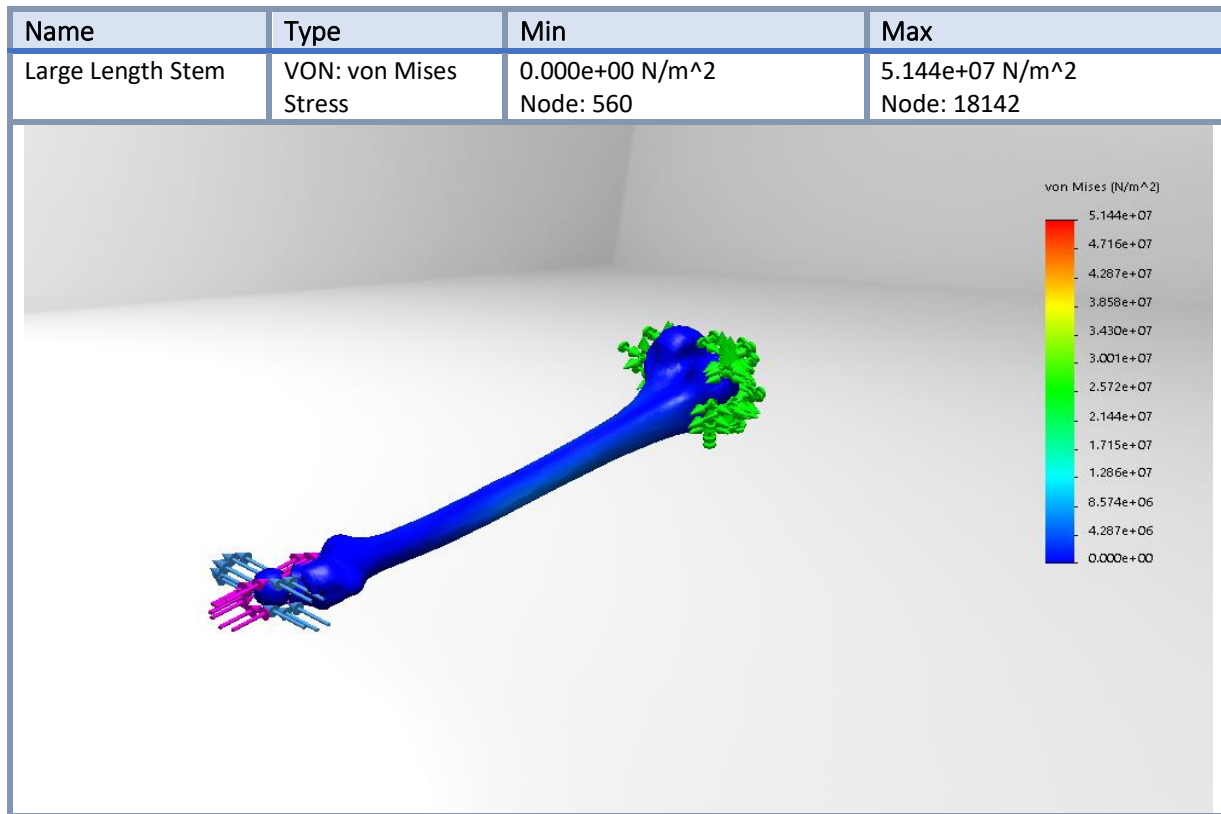
Table 1: Material Properties

Model Reference	Properties		Components
	Name:	**Ti-3Al-8V-6Cr-4Mo-4Zr**	Stem
	Model type:	**Linear Elastic Isotropic**	
	Default failure criterion:	**Max von Mises Stress**	
	Yield strength:	**1.03421e+09 N/m^2**	
	Tensile strength:	**1.22e+09 N/m^2**	
	Compressive strength:	**1.09e+09 N/m^2**	
	Elastic modulus:	**1.04e+11 N/m^2**	
	Poisson's ratio:	**0.33**	
	Mass density:	**4,820 kg/m^3**	
	Shear modulus:	**4e+10 N/m^2**	
	Thermal expansion:	**8e-06 /Kelvin**	
	Name:	**Femur Bone**	Bone
	Model type:	**Linear Elastic Isotropic**	
	Default failure criterion:	**Max von Mises Stress**	
	Yield strength:	**1.6e+08 N/m^2**	
	Elastic modulus:	**2.13e+09 N/m^2**	
	Poisson's ratio:	**0.3**	
	Mass density:	**0.002 kg/m^3**	
	Shear modulus:	**7e+07 N/m^2**	

To simulate the functions of biomechanical forces in the head of the femur the most accurate way, an establish the boundary condition an in-vivo video study was used. This video is available online[iv].

The force used were 175 N in the vertical direction and 45 N in the horizontal direction. These forces are the average forces collected from the video. Finally, boundary conditions were established by fixation the end cap of the femur. Once everything was set the study button was pressed, taking care of the mesh of prothesis and giving back of the stress, displacement and strain results.

Results

Name	Type	Min	Max
Control Stem	VON: von Mises Stress	0.000e+00 N/m^2 Node: 4116	1.160e+07 N/m^2 Node: 3556

Name	Type	Min	Max
Medium Length Stem	VON: von Mises Stress	0.000e+00 N/m^2 Node: 5968	9.203e+06 N/m^2 Node: 2631

Name	Type	Min	Max
Large Length Stem	VON: von Mises Stress	0.000e+00 N/m^2 Node: 560	5.144e+07 N/m^2 Node: 18142

The results show the FEA study of the three-length stem prothesis. On the right it is displayed the amount of stress in Von Mises scale. The Control Stem showed that it suffered a stress in the neck of the femur area of about $1.064x10^7 N/m^2$ and a reasonable stress towards the lower part of about $2.90x10^6 N/m^2$. On the Medium length stem is possible to find a concentrated stress also about the center region of the femur with a value of about $4.601x10^6 N/m^2$. Finally, in the larger stem length, the stress is generalized around the femur with about $4.287x10^6 N/m^2$.

For the number of elements adopted, it varies among the models. This is because a high-quality mesh in difficult shape areas is a complicated process, even for the computer, and it leads to a lot of errors. Although this is not the maximum capacity, the accuracy given is still high, and it is still considered a High-resolution mesh. Table 2 shows the number of elements, size and total nodes used in each study.

Table 2 Mesh Information

	Control Stem	Medium Length Stem	Large Length Stem
Total Nodes	18647	36931	18750
Total Elements	12195	24686	12286
Element Size (mm)	8.21492	9.90226	8.26507
Tolerance (mm)	0.410746	0.495113	0.413253

Discussion

This paper made the mechanical testing of the hip replacement prothesis based on tools given by the software *Solidworks* that allowed to model, mesh and study the biomechanical principals that affect three different length stems hip prothesis.

The basic biomechanical principal that governs the hip joint consists fundamentally in maintaining the equilibrium and balance during standing and performing activities. The description of forces that act around the hip joint during the daily activities is given by the sum of forces that our body weight establishes with the ground through the gravitational pull and its opposite force. Because our body consists of linked segments, the amount of force established is distributed across all joints. The hip joint generally acts as a lever, dividing the total force into one third conducting it to each leg and hip. These transmitted forces can come in different directions, depending on the activities perform, creating different types of stress. Given that the bone is anisotropic material the strength of the femur is different depending on the direction. Typically, compressive ultimate strength is of 205 MPa and tensile ultimate strength is of 131 MPa in the longitudinal direction.

Looking at the magnitude of this number and the results we can conclude that the length of the Stem does not cause a huge impact on the femur. However, variations among the model's designs are evident. On the control stem it was noticeable a more irregular stress distribution. This happens because of the short transmission area that existed between the femur and stem, while in the center zone we find a normal response of the bone elasticity. Next, on the medium length a lever force caused by the stem inside the femur caused the concentration of stress to happen around the middle. Finally, on the larger length stem we find a more even distribution of stress given the bigger contact area. In summary in this report we understood that the length of the stem is relatable to the distribution of stress in the femur bone.

References

[i] 2020. *Introduction To Stress Analysis (The Finite Element Method)*. Middlesex University.

[ii] *Phou, M., 2019. How To Make A Hip Implant On Solidworks Part 1_2*

[iii] Maharaja, S., 2020. *Numerical Analysis Of Fractured Femur Bone With*. ScienceDirect.

[iv] Li, J., 2020. *Biomechanics Of Total Hip Implant*. PDE2975, Middlesex.
https://orthoload.com

Borovkov, A., Maslov, L., Zhmaylo, M., Zelinskiy, I., Voinov, I., Keresten, I., Mamchits, D., Tikhilov, R., Kovalenko, A., Bilyk, S. and Denisov, A., 2018. *FINITE ELEMENT STRESS ANALYSIS OF A TOTAL HIP REPLACEMENT IN TWO-LEGGED STANDING*. Russian Journal of Biomechanics.

Javir, M. and Kirkire, D., 2017. *Finite Element Analysis Of Femur Prosthesis*. Ratnagiri, Maharashtra, India: International Conference on Nascent Technologies in the Engineering Field.

Lodygowski, T., Rakowski, J. and Litewka, P., 2014. *Recent Advances In Computational Mechanics*. CRC Press.

Singha, S. and Harsha, A., n.d. *Analysis Of Femoral Components Of Cemented Total Hip- Arthroplasty*. Varanasi 221005, Uttar Pradesh, India.: Department of Mechanical Engineering, Indian Institute of Technology (Banaras Hindu University.)