Proceedings IRF2018: 6th International Conference Integrity-Reliability-Failure
Lisbon/Portugal 22-26 July 2018. Editors J.F. Silva Gomes and S.A. Meguid
Publ. INEGI/FEUP (2018); ISBN: 978-989-20-8313-1

PAPER REF: 7150

NUMERICAL ANALYSIS OF A COMPOSITE LEG PROSTHESIS

João Victor Gomes dos Santos[1], Vincent Wong[2], Vicente Gerlin Neto[3(*)], Luis Carlos Paschoarelli[1], Marcos Antônio dos Reis Pereira[2], Carlos Alberto Fortulan[4], Cesar Renato Foschini[3]

[1]Department of Design, São Paulo State University - Unesp, Bauru, Brazil
[2]Department of Mechanical Engineering, São Paulo State University - Unesp, Bauru, Brazil
[3]Federal Institute for Education, Science and Technology of São Paulo at Birigui, Birigui, Brazil
[4]Department of Mechanical Engineering, University of São Paulo - USP, São Carlos, Brazil
(*)*Email:* gerlinneto@gmail.com

ABSTRACT

This work consists on analysing the behaviour of a transtibial prosthesis developed with a new composite material of polymer matrix and bamboo reinforcement. The analysis developed was a static analysis using the finite element method, considering the tensions generated by the weight of the patient using an FEA software (NASTRAN IN CAD 2015 and AUTODESK INVENTOR 2015). The development of this study will provide information on the elements that suffer the highest concentration of stresses.

Keywords: biocomposites, Finite Element Method, prosthesis, bamboo.

INTRODUCTION

In 1972, about fifteen years after the finite element method (FEM) initiated a revolution in stress analyses of structures in engineering mechanics, this 'new method to analyze the mechanical behavior of skeletal parts' (Brekelmans *et al*., 1972). was first introduced in the orthopaedic literature. Traditionally, interest existed in orthopaedics and orthopaedic-related sciences with respect to stresses and strains in loaded bones, specifically concerning the relationship between architecture and load-bearing function (e.g. Wolff, 1870; Koch, 1917). The mathematical tools available for stress analyses in classic mechanics, however, were not very suitable for the highly irregular structural properties of bones. Hence, the powerful FEM became the logical choice to fill the gap, due to its unique capability to evaluate stresses in structures of complex shape, loading and material behavior (Huiskes *et al*., 1983).

FUNDAMENTATION

Finite Element Analysis is a methodology commonly used in the mechanical design area to obtain information that allows simulating the behaviour of an engineering model through the submitted efforts.

This method, especially in complex cases, has replaced analytical methods. The problems presented can range from fluid analysis, static, hyperstatic, dynamic, heat transfer analysis, among others. For this, one must choose the appropriate model and method for the nature of the problem.

According to (Modlen *et al*., 2007), the development of a computational analysis is divided into three steps presented below:

Pre-processing: one of the most important steps. It consists on creating the model with the problem initial considerations to be studied with possible simplifications. The ability of the user is important, the skill of the designer during this process is important, because possible simplifications and arguments that can be used have provided a more reliable result.

Analysis: The analysis process is the more important step on the finite element method procedure. It is necessary to explain some definitions important to understand the procedure to include in the calculation of the algorithm.

Initially a finite element method to linear analysis in solid and structural mechanics begin for define the linear spring equation.

$$\{F\} = [k]\{U\} \qquad \text{Eq. 1}$$

Where U is a vector of the system global displacement and R is a vector of forces acting in the direction of these displacements. And k is the stiffness matrix of the complete element assemblage. This matrix is obtained from the stiffness matrices of the individual element using the direct stiffness method. In this procedure the structure stiffness matrix K is calculated by direct addition of the element stiffness matrices. (Bathe *et al.*, 2006)

$$K = \sum_i K_i^e \quad , \qquad \text{Eq.2}$$

where the summation includes all elements.

Post-Processing: The post-processing step consists of receiving the data obtained during the mathematical analysis previously developed and transforming and communication data, such as motion-linked images and values.

According to (Bathe *et al.*, 2006) each stage of the development process is defined by the scheme shown in Figure 1.

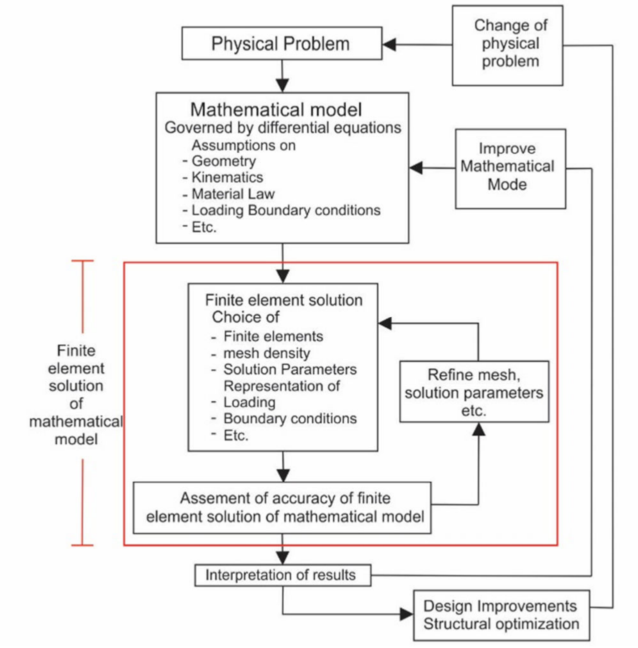


Fig. 1 - Finite element analysis process.
Source: Bathe, 2006

In this work, we intend to analyze the static behavior of transtibial prosthesis by which, according to (Bathe *et al.*, 2006) of the different approaches presented, the linear structural analysis is the most appropriate for the study that we wish to develop.

Considering that, the analysis of finite elements is a numerical analysis and that allows having a prediction of the behavior of a system using computational mathematical models, the criterion of convergence presents a response that allows assert that the mathematical model defined in principle, is valid. Therefore, it can be said that appropriate solutions of finite elements must converge to be satisfactory (Bathe *et al.*, 2006).

METHODOLOGY

In this work, the computational methodology was applied, following the main steps:

Pre-processing
Analysis
Post-processing

Pre-Processing

Geometry

The assembly is composed of 13 pieces, as shown in the Figure 2. It is important to consider the approximate conditions of the finite element analysis method, as some pieces were simplified from the original assembly. The excess of areas or operations in the sketch generate more nodes in the mesh which makes the processing longer.

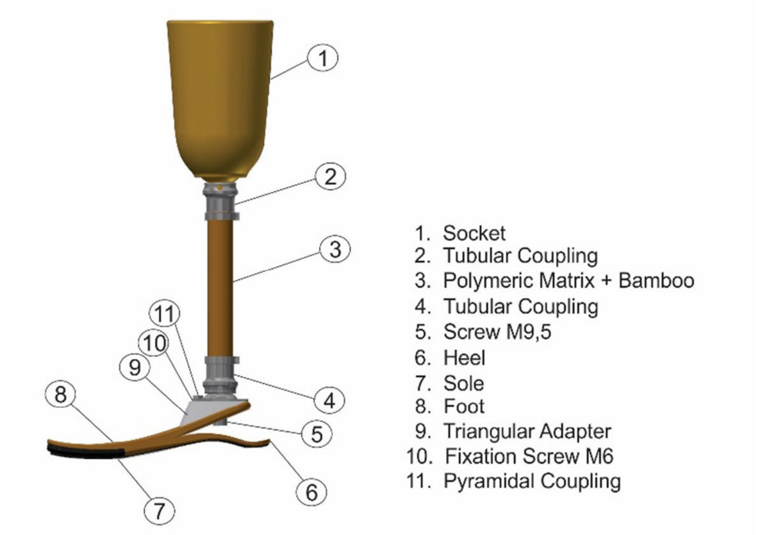


Fig. 2 - Prostheses Assembly.

Law Material

In the design and project process, was considered a total of six materials shown in Table 1, with their respective mechanical properties.

Table 1 - Mechanical properties of materials in study

Item	Quant.	Part name	Material	Density (kg/m^3)	Elastic Modulus (GPa)	Poisson's ratio
1	1	Socket	Carbon Fiber Composite	1,750	70	0.1
2	1	Tubular Coupling	Stainless Steel	7,850	190	0.28
3	1	Bamboo + Polymeric Matrix	Polymeric matrix	1,140	4.940	0.39
3	1		Bamboo reinforcement	790	20	0.3
4	1	Tubular Coupling	Stainless Steel	7,850	190	0.28
5	1	Screw M9,5	Stainless Steel	7,850	190	0.28
6	1	Heel	Bamboo	790	20	0.3
7	1	Sole	Rubber	930	0.003	0.5
8	1	Foot	Bamboo	790	20	0.3
9	1	Triangular adapter	Stainless Steel	7,850	190	0.28
10	2	Fixation Screw M6	Stainless Steel	7,850	190	0.28
11	1	Pyramidal Coupling	Stainless Steel	7,850	190	0.28

Disposition assembly:

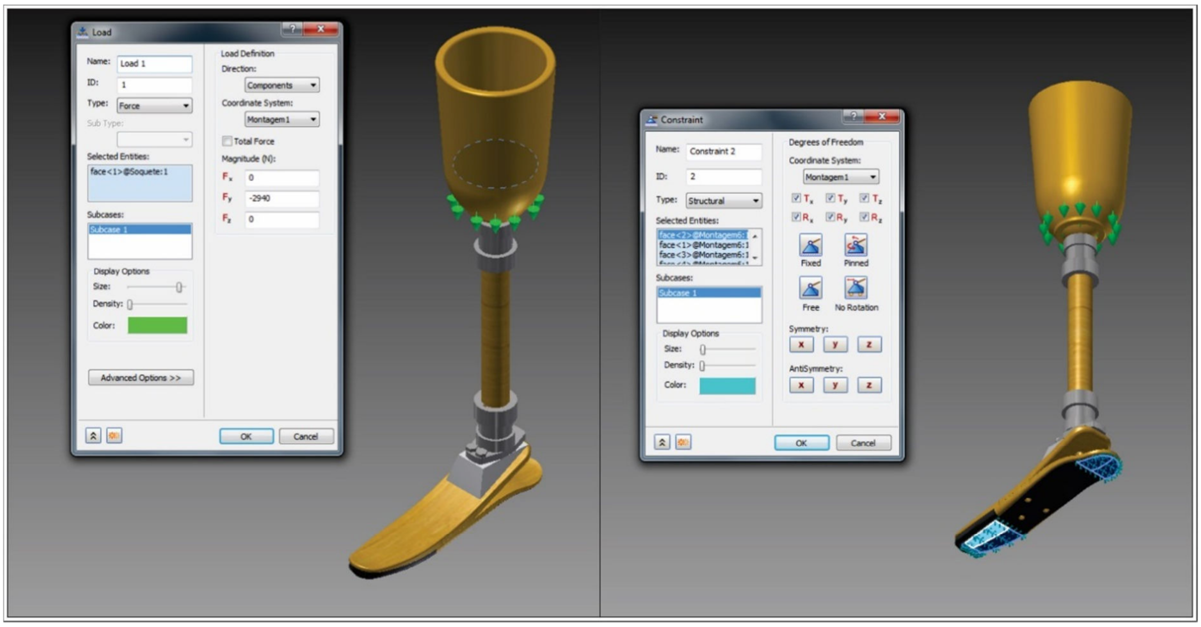
The global assembly was developed in two subassemblies and Table 2 presents the arrangement by assembly. In addition, the triangular adapter, tubular coupling and the pyramidal coupling as they are the same material, were considered as a single piece. This generates an easier step for the stiffness matrix processing.

Table 2 - Assembly disposition

Item	Part Name	Assembly
1	Socket	1
2	Tubular Coupling	1
3	Bamboo Reinforcement	1
3	Polymeric Matrix	1
4	Triangular adapter + Tubular Coupling + pyramidal Adapter	1
5	Screws M6 and M9.5	1
7	Foot	2
8	Sole	2
9	Heel	2

Load and Constraint

A total weight of 100 kg was considered for the analysis as it is a common value for prosthesis analysis, considering a safety factor of 3. The final weight is 2,940 N applied on the lower part of the coupling as showed in Figure 3(a). The area selected to be constraint is shown in Figure 3(b) as this is the main contact area with the floor.



Load

Name: Load 1
ID: 1
Type: Force
Sub Type:
Selected Entities: face<1>@Soquete:1
Subcases: Subcase 1
Load Definition
Direction: Components
Coordinate System: Montagem1
☐ Total Force
Magnitude (N):
F_x 0
F_y -2940
F_z 0

Constraint

Name: Constraint 2
ID: 2
Type: Structural
Selected Entities:
face<2>@Montagem6:1
face<1>@Montagem6:1
face<3>@Montagem6:1
Subcases: Subcase 1
Degrees of Freedom
Coordinate System: Montagem1
☑ T_x ☑ T_y ☑ T_z
☑ R_x ☑ R_y ☑ R_z

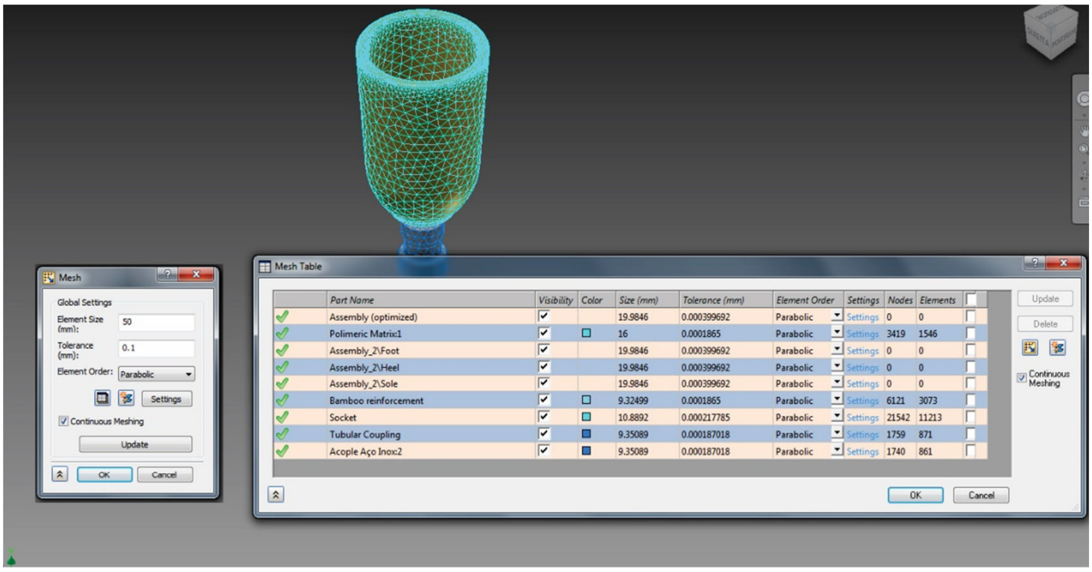
(a) Loads locations

(b) Constraint locations

Fig. 3 - Loads and constraint location

Meshing

One of the most important aspects to obtain a correct simulation is mesh configuration. This configuration is divided in two steps, the first one is global configuration and the parameters to choose depend on the element size and the tolerance between neighboring parts. High values in element size generate more nodes to be distributed. Tolerance is important when there are two neighboring parts, in certain situations the nodes are shared up to the point of the closing stiffness matrix. A sufficiently wide range of parameters is necessary for a smooth calculation, Figure 4 presents the parameters configuration considered in this analysis.



Mesh

Global Settings
Element Size (mm): 50
Tolerance (mm): 0.1
Element Order: Parabolic
☑ Continuous Meshing

Mesh Table

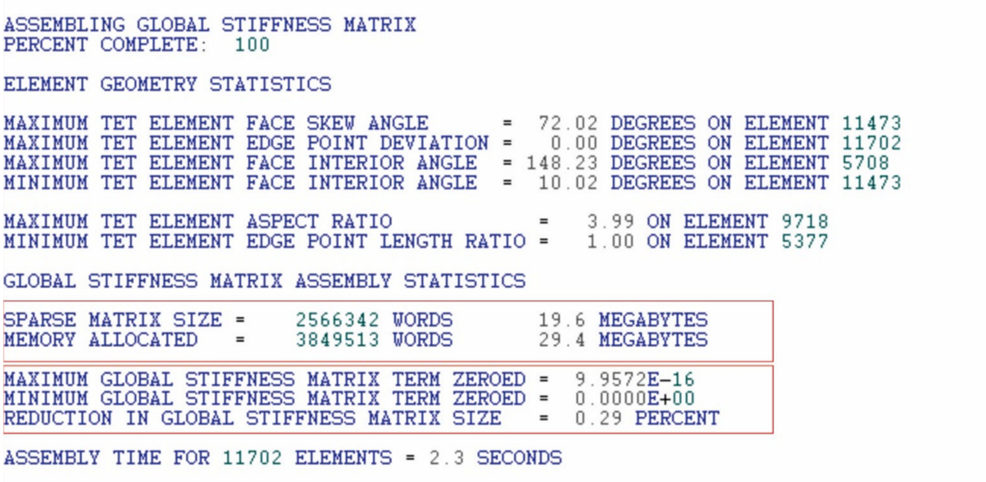
	Part Name	Visibility	Color	Size (mm)	Tolerance (mm)	Element Order	Settings	Nodes	Elements	☐
✓	Assembly (optimized)	☑		19.9846	0.000399692	Parabolic	Settings	0	0	☐
✓	Polimeric Matrix:1	☑	■	16	0.0001865	Parabolic	Settings	3419	1546	☐
✓	Assembly_2\Foot	☑		19.9846	0.000399692	Parabolic	Settings	0	0	☐
✓	Assembly_2\Heel	☑		19.9846	0.000399692	Parabolic	Settings	0	0	☐
✓	Assembly_2\Sole	☑		19.9846	0.000399692	Parabolic	Settings	0	0	☐
✓	Bamboo reinforcement	☑	■	9.32499	0.0001865	Parabolic	Settings	6121	3073	☐
✓	Socket	☑	■	10.8892	0.000217785	Parabolic	Settings	21542	11213	☐
✓	Tubular Coupling	☑	■	9.35089	0.000187018	Parabolic	Settings	1759	871	☐
✓	Acople Aço Inox:2	☑	■	9.35089	0.000187018	Parabolic	Settings	1740	861	☐

☑ Continuous Meshing

Fig. 4 - Mesh parameters and Global configuration

Analysis and Post Processing

During the analysis two simulations were used. The first partial simulation was the first assembly; the corresponding values to the stiffness matrix were processed and more than 23,000 nodes were presented, as well as 71,154 degrees of freedom. Those degrees of freedom are used at everyone node. Each node provides 3 degrees of freedom, Figure 5 shows the post processing analysis and figure 6 shows assembly 1 in detail.



```
ASSEMBLING GLOBAL STIFFNESS MATRIX
PERCENT COMPLETE:  100

ELEMENT GEOMETRY STATISTICS

MAXIMUM TET ELEMENT FACE SKEW ANGLE       =  72.02 DEGREES ON ELEMENT 11473
MAXIMUM TET ELEMENT EDGE POINT DEVIATION =   0.00 DEGREES ON ELEMENT 11702
MAXIMUM TET ELEMENT FACE INTERIOR ANGLE  = 148.23 DEGREES ON ELEMENT 5708
MINIMUM TET ELEMENT FACE INTERIOR ANGLE  =  10.02 DEGREES ON ELEMENT 11473

MAXIMUM TET ELEMENT ASPECT RATIO            =   3.99 ON ELEMENT 9718
MINIMUM TET ELEMENT EDGE POINT LENGTH RATIO =   1.00 ON ELEMENT 5377

GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS

SPARSE MATRIX SIZE =     2566342 WORDS         19.6 MEGABYTES
MEMORY ALLOCATED   =     3849513 WORDS         29.4 MEGABYTES

MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  9.9572E-16
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  0.0000E+00
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE   =  0.29 PERCENT

ASSEMBLY TIME FOR 11702 ELEMENTS = 2.3 SECONDS
```

Fig. 5 - Post-processament data analyses

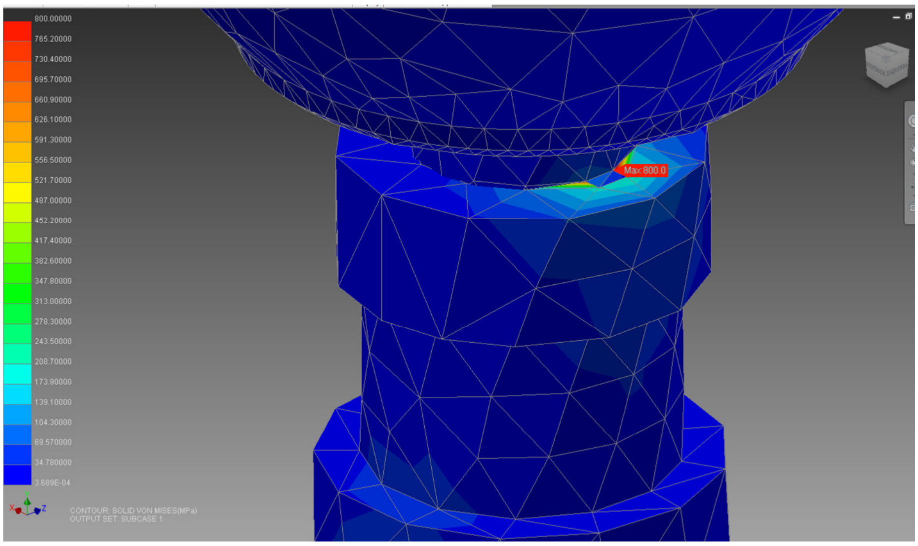
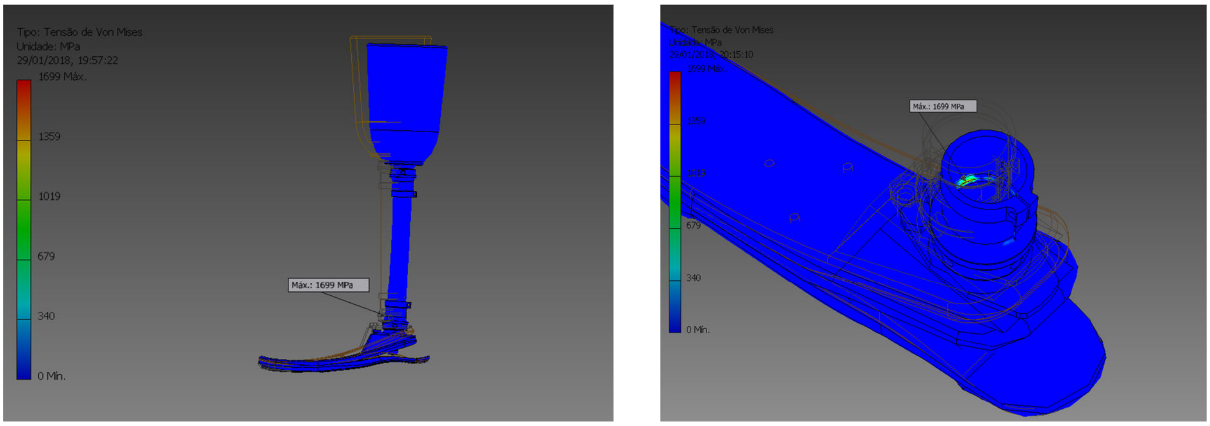


Fig. 6 - Representation in the partial analyzed of assembly 1

RESULTS

The tensions shown in Figure 7(a) present the higher tension concentration as is detailed in figure 7(b). The higher tension has magnitude of 1,698.65 MPa and the total displacement is 7.344 mm. The main model considerations are showed in Table 3.



(a) Representation of the maior tension (b) tension presented in the tubular coupling

Fig. 7 - Representation of the Maior tension in the prostheses.

Table 3 - Results for the FEA method

Name	Minimum	Maximum
Volume	1,119,110 mm^2	
Mass	1.45712 kg	
Von Mises Tension	0 MPa	1,698.65 Mpa
Displacement	0 mm	7.344 mm
Tension XX	-686,193 MPa	927.832 MPa
Tension YY	-2,116,54 MPa	1,868.52 MPa
Displacement in X	-6.30174 mm	0.296409 mm
Displacement in Y	-3.81826 mm	0.522518 mm
Deformation XX	-0.248956 mm	0.252634 mm
Deformation YY	-0.541858 mm	0.277003 mm

CONCLUSION

Through the numerical analysis employing finite element methods, the distribution of static charges applied to the different elements of a transtibial prosthesis could be calculated. It was verified that the project developed with the use of a polymeric bio composite consisting of resin and bamboo fiber proved to be resistant to attend the mechanical needs satisfactorily to support the efforts of people with up to 100 kg.

The main Von Mises stress obtained was 1,698.65 MPa. The critical element of the system is the lower tubular coupling due to the torsion it suffers as the force vector is vertical and its vector position for the region is in contact with the ground. The total displacement obtained is a value according to the manual numerical calculations developed considering that the total dislocation corresponds to the sum of the individual dislocations of each element.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the funding by Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - CAPES.

REFERENCES

[1] Bathe, K.-J. (2006). Finite element procedures. Second edition.

[2] Ghoreisi, K.; Safari, M.R. Prevalence of heat and perspiration discomfort inside prostheses: Literature review. Journal of rehabilitation research and development, v. 51, n. 6, pp. 855, 2014.

[3] Modlen, G. (2010). Introduction to Finite Element Analysis, (January 2008), 87.

[4] Nurhanisah, M.H. *et al.*, Design of Prosthetic Leg Socket from Kenaf Fibre Based Composites. Green Biocomposites, [s.l.], pp.127-141, 2017. Springer International Publishing.

[5] PereiraA, M. A. D. R. (2012). Projeto Bambu: Introdução de especies, manejo, caracterização e aplicações. Tese para obtenção de livre docência, 2000.

[6] Nastran In Cad and Autodesk Inventor 2015 - Student Version.