ANALYSIS OF THE STRESS DISTRIBUTION GENERATED BY MASTICATORY LOADS ON DENTAL IMPLANTS USING FINITE ELEMENT METHOD.

Selma Hissae Shimura da Nóbrega(1), nobrega@ufrnet.br
Gutemberg Morais Serrano (2), guto_140@globo.com
Arthur Álax de Araújo Albuquerque (2), arthuralax@yahoo.com.br
Álvaro Benevides (3), alvarobenevides@bol.com.br
UFRN – CT, Av. Senador Salgado Filho, 3000 – Lagoa Nova - Natal/RN
(1) Professor, Programa de Pós-graduação em Engenharia Mecânica
(2) Undergraduate students, Civil Engineering
(3) Graduated student, Programa de Pós-graduação em Engenharia Mecânica

Abstract. *The evaluation of the biomechanical behavior of dental implants structures and their interaction with cortical and cancellous bones are of extremely importance for dentists. During years, these professionals have had only clinical histories, accompaniments or clinical trials of high costs to evaluate these implants structures. Nowadays, however, computational analysis based on Finite Element Method has proven its efficiency and its reliability when studying the following aspects: stresses that arise from the masticatory loads on dental implants and their distribution in adjacent structures; the location of implants; proposal of new geometries, implants composition and structure, and prosthetic components, prosthetic planning, and interaction with surrounding tissues. This work deals with linear elastic analysis of implants when submitted to specific masticatory action and evaluates new implants and threads geometries in order to mitigate the effects of bone loss typically observed in the neck of the implant and to provide a better stress distribution in cancellous bone. For this purpose, three implants geometries with different threads geometries were analyzed using 2D finite elements for Plane Strain. The results show that cylindrical implants with rectangular threads have a better performance on stress distribution.*

Keywords*: finite element method, bioengineering, biomechanics, dental implants.*

1. INTRODUCTION

In 2010, the World Health Organization published that almost 50% of Brazilians between the ages of 35 and 44 have lost, at least, 12 teeth and 80% of the elderly have less than 20 teeth. To solve this problem it is necessary not only an oral health care orientation for those with lower educational level and financial conditions, but also provide a cost reduction of the process and the refinement of the implants insertion techniques.

Araújo et al. (2008) have described that implants are used to replace teeth roots and usually made, commercially, of pure titanium, a bioinert and biocompatible material. The bone-implant system is basically composed by the implant, the prosthetic component that includes abutment and its screw, the prosthesis formed by copping and crown, and the supporting bone formed by cortical bone and cancellous bone.

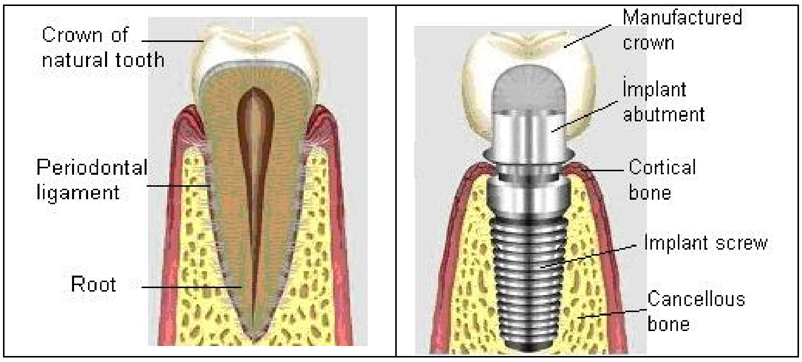


Figure 1 – Tooth structure and bone implant system.
(adapted from http://www.icoc.com.br/outros2.php)

Figure 1 shows, on the left, the tooth structure with a thin tissue involving its root. It is called periodontal ligament whose function is to protect the root of sudden and intense masticatory forces and, moreover, it is responsible for producing a natural damping effect. This tissue does not exist in an implant system because there is a direct contact between the component and the surrounding bone tissue. In this case, when these natural and artificial elements are adapted to each other, it is said that the implant is osseointegrated.

Many factors influence the osseointegration process. In fact, it does not occur immediately after implant insertion, but six or eight weeks later and during all life. The osseointegration depends on: blood supply, no trauma occurrence during the implant insertion, the biocompatibility of the implant material, and the stability provided by local geometry. The bone structure integrity around the implant should be preserved in order to properly transfer all efforts during the bone remodeling process. It is related to a modification of bone density and shape as a result of the stresses level developed in structure. Thus, the forces intensities and direction, as well as the implant geometry, are critical parameters.

The bone structure is another important factor to be considered when studying the biomechanical behavior of bone-implant system. There are two kinds of osseous tissues: the cortical bone, external and characterized by better mechanical properties imposing movement restriction on implants; and the cancellous bone, internal and typified by low level of mechanical strength.

It is known that the stresses developed in the neck of the implant during the masticatory process are critical. Nevertheless, it is important to remember that the implant is totally fixed in cancellous bone. Thus, in order to have a satisfactory biomechanical performance, it is crucial to reduce the stress level on the neck of the component and to provide an adequate stress distribution along the implant longitudinal axe.

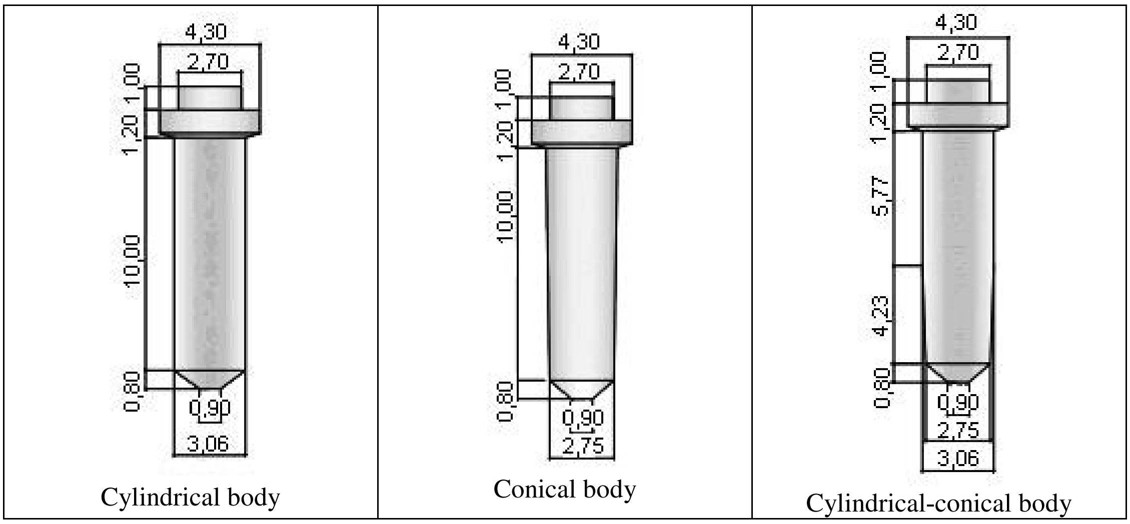
In practice, Implantology or Prosthesis professionals have only clinical histories of their patients and, sometimes, clinical trials, limited by the high costs of the equipments and the difficulties of maintaining the specimens alive, in order to analyze the functionality and performance of implant structures.

In this context, computational analysis based on the Finite Element Method (FEM) has proven its efficiency, versatility and reliability in applications of Dentistry (van Staden et al., 2006). Investigations of the stress level that arise from the masticatory loads applied on dental implants and its distribution on adjacent structures, the study of implants placement, the proposal of new geometries, new material composition and new threads geometries including prosthetic components, prosthetic planning, and the evaluation of the surrounding tissues interaction have been undertaken in recent years in order to provide a better understanding of the biomechanical behavior of implants. (Huang et al. 2010; Erasian and Inam, 2010; and Faegh and Müftü, 2010).

In this work, implant structures subjected to masticatory forces are analyzed by FEM aiming at the evaluation of implant and threads geometries influence on reducing stress level in bone-implant region.

2. METHODOLOGY

The Finite Element analysis was carried out using ADINA (Automatic Dynamic Incremental Nonlinear Analysis) code, version 8.7, considering three basic geometries of implant body: cylindrical, conical and cylindrical-conical (implants with prosthetic connection interface or hexagonal or HE), shown in Fig. 2. In sequential studies, specific geometries of threads like trapezoidal, triangular and rectangular were considered, as illustrated in Fig. 3.

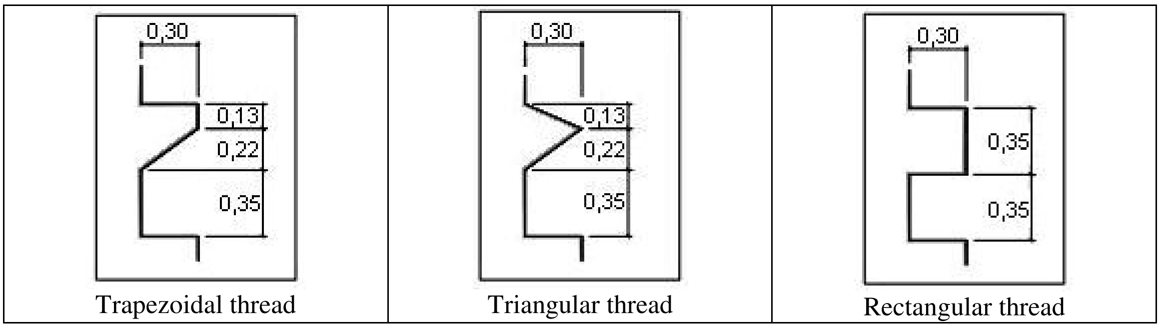


Cylindrical body

Conical body

Cylindrical-conical body

Figure 2 - Implants Geometries



Trapezoidal thread

Triangular thread

Rectangular thread

Figure 3 - Threads Geometries

The Plane Strain analysis was based on four groups of 2D elements which mechanical properties are presented in Tab. 1 and based on Wang et al. (2010). All degrees of freedom were restricted in the limits of cortical bones and in cancellous bone. Masticatory forces considered as a 175 N concentrated force vertically applied and applied at 45 degrees as shown in Fig. 4. This figure also illustrates the basic finite element mesh composed by four node elements, applied force and boundary conditions used for the cylindrical, conical and cylindrical-conical implant body analysis.

Table 1 - Mechanical properties of the element groups

ELEMENT GROUP	MATERIAL	YOUNG´S MODULUS (GPa)	POISSON´S RATIO
1	Implant	115	0,35
2	Prosthesis	90	0,30
3	Cortical bone	13,7	0,30
4	Cancellous bone	1,37	0,30

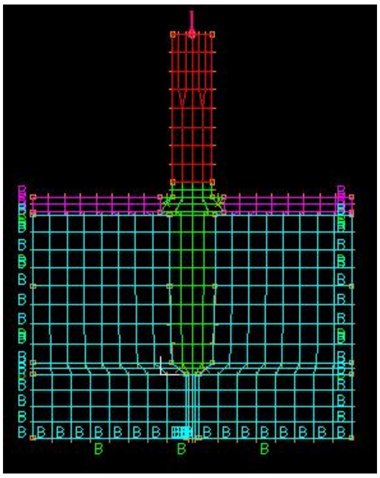
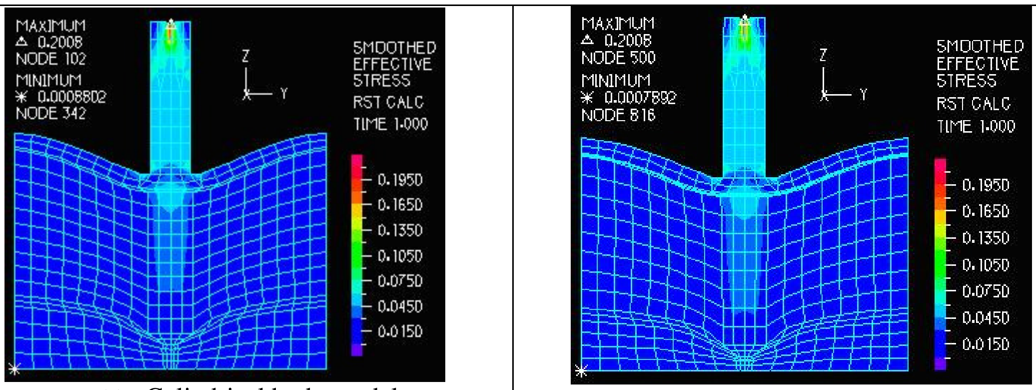


Figure 4 – Finite element mesh.

3. RESULTS

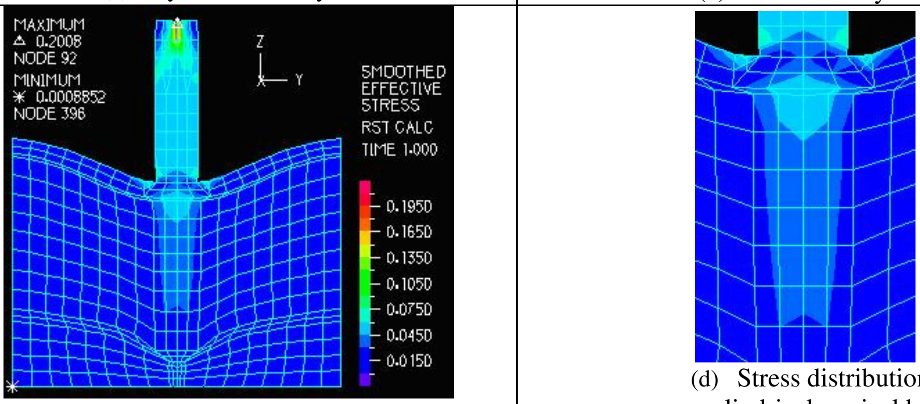
Studies were carried out using the three basic implants geometries with subsequent inclusion of trapezoidal, triangular and rectangular threads. Figures 5a, 5b and 5c show the stress level developed when a vertical force is applied on implants without threads.

The effective stress distribution in implant systems with different geometries is visually similar. Higher stress values are observed (in lighter color) at the neck of the implant and at the contact between implant and cortical bone (with higher stiffness). Besides, it is also possible to verify an internal distribution along the implant body and concentrated in its longitudinal axis (Fig 5d). Implants failures are commonly caused by fractures in the neck of the implant and by the bone reabsorption in surroundings, which eventually cause gaps between the bone and the component. Food residues in these gaps lead to generation of bacteria colonies that undermine the implant system. In some situations, the bone tissue becomes so damaged that it is necessary to place a bone graft for posterior insertion of a new implant. Table 2 shows the effective stress level resulting from the analysis.



(a) Cylindrical body model

(b) Conical body model



(c) Cylindrical-conical body model

(d) Stress distribution for cylindrical-conical body

Figure 5 - Effective stress level for different implants geometries (a, b, c) and detail on the neck of the implant and its surroundings (d)

Table 2 - Maximum effective stresses for cylindrical, conical and cylindrical-conical implant bodies.

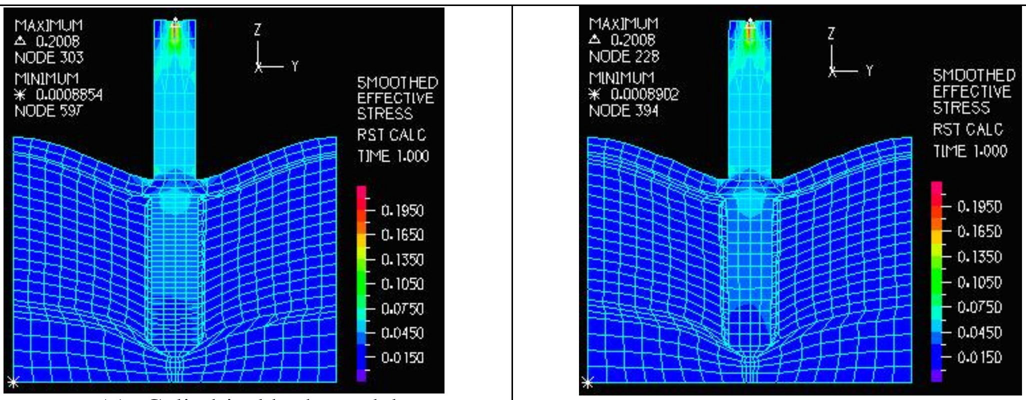
	Region	Cylindrical body	Conical body	Cylindrical-conical body
Vertical force	Neck of the implant	7.85763E-02 GPa	7.83337E-02 GPa	7.90040E-02 GPa
	Cortical bone	3.50255E-02 GPa	3.29410E-02 GPa	3.59801E-02 GPa
Oblique force (45°)	Neck of the implant	9.91861E-01 GPa	9.94647E-01 GPa	9.93293E-01 GPa
	Cortical bone	2.01275E-01 GPa	2.02169E-01 GPa	2.02704E-01 GPa

The stresses developed at the neck of the conical body implant and at the cortical bone-implant contact are the lowest when compared with the other geometries analyzed with the vertical load. When oblique force is used, obviously the stress values are higher than those verified when vertical force is considered. So, in the latter situation, implants with cylindrical body show the lowest stress level for both neck and cortical bone regions. The stresses shown in Tab. 2 will be used as reference in evaluations where threads will be considered.

The introduction of trapezoidal threads modifies the stress field in the implant body, reducing the concentration on top and producing a uniform distribution of stress along its axis. Figure 6 illustrates the effective stresses for vertically applied load and Tab. 3 displays the results.

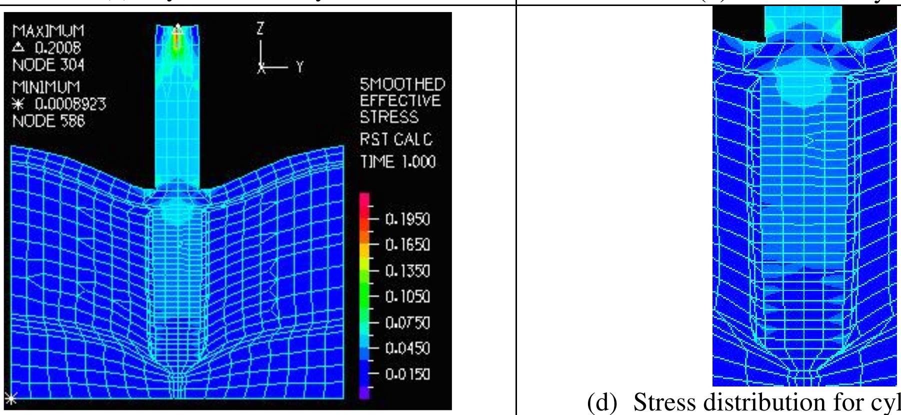
Table 3 – Extreme effective stresses for cylindrical, conical and cylindrical-conical body with trapezoidal threads.

	Region	Cylindrical body	Conical body	Cylindrical-conical body
Vertical force	Neck of the implant	7.75703E-02 GPa	7.77579E-02 GPa	7.78597E-02 GPa
	Cortical bone	3.37217E-02 GPa	3.45575E-02 GPa	3.45628E-02 GPa
Oblique force (45°)	Neck of the implant	9.91344E-01 GPa	9.90663E-01 GPa	9.91553E-01 GPa
	Cortical bone	1.96871E-01 GPa	2.00221E-01 GPa	1.98162E-01 GPa



(a) Cylindrical body model

(b) Conical body model



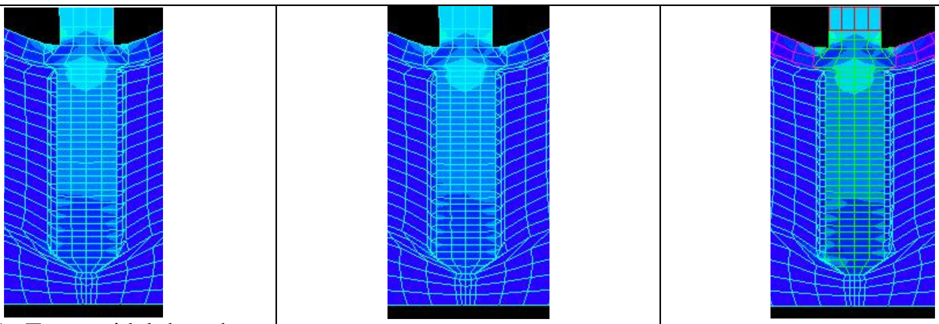
(c) Cylindrical-conical body model

(d) Stress distribution for cylindrical-conical body model

Figure 6 - Effective stress for implants with trapezoidal thread (a, b and c) and detail of stress distribution (d)

Table 3 can be compared with Tab. 2. It is possible to verify that threads are beneficial since they reduce the magnitude of the effective stress level at the same time that improve its distribution which is now extended to the implant lateral borders. Only for the conical body model it is observed an increase of effective stress level in cortical bone contact area.

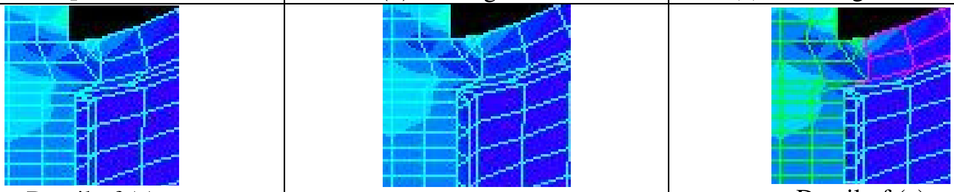
In order to evaluate commercial threads geometries, the cylindrical body that presented lower stress intensities for vertical and oblique loading was analyzed. It was also considered the triangular and rectangular threads, as done by Erasian and Inan (2010). Figure 7 shows the effective stresses for such situations and Tab. 4 the numerical results.



(a) Trapezoidal threads

(b) Triangular threads

(c) Rectangular threads



Detail of (a)

Detail of (b)

Detail of (c)

Figure 7 - Effective stress level for cylindrical body with threads (a, b and c)

Table 4 - Maximum effective stresses for cylindrical body with trapezoidal, triangular and rectangular threads.

	Region	Trapezoidal	Triangular	Rectangular
Vertical force	Neck of the implant	7.77579E-02 GPa	7.76849E-02 GPa	7.73564E-02 GPa
	Cortical bone	3.45575E-02 GPa	3.36359E-02 GPa	3.33794E-02 GPa
	Implant base	---	---	1.31582E-01 GPa
Oblique force (45º)	Neck of the implant	9.90663E-01 GPa	9.90940E-01 GPa	9.91235E-01 GPa
	Cortical bone	2.00221E-01 GPa	1.94990E-01 GPa	1.96246E-01 GPa

Cylindrical implants with rectangular threads present lower intensities of effective stress level on the neck and on cortical bone-implant connection when compared with other geometries. Another fact that can be detected is a better stress distribution along the implant body reaching its base with the maximum value 1.31582E-01 GPa. This is the only situation in which stresses are observed at the base of the implant in contact with cancellous bone. It is possible to conclude that rectangular threads in cylindrical implants are apparently the most suitable geometry to be used because they allow a stress level reduction in regions traditionally considered as critical and promote its distribution until the implant base.

Table 5 indicates the results from Tab. 2 and Tab. 4 for the cylindrical implant in order to make easier the comparison among other results. There is a significant reduction of the stress level in the cortical bone contact area and an adequate stress distribution until the implant base. These facts are of great interest for professionals in Implantology. High stresses on the neck of the component can be satisfactorily reduced by implant geometry changes.

Table 5 – Effective stresses for cylindrical body

	Region	Without threads	Rectangular threads	Stress reduction
Vertical force	Neck of the implant	7.85763E-02 GPa	7.73564E-02 GPa	1,55%
	Cortical bone	3.50255E-02 GPa	3.33794E-02 GPa	4,70%
Oblique force (45º)	Neck of the implant	9.91861E-01 GPa	9.91235E-01 GPa	0,06%
	Cortical bone	2.01275E-01 GPa	1.96246E-01 GPa	2,50%

Commercially, there are implants with conical body shape and trapezoidal threads with variable size. It means that the horizontal dimension of threads grows while implant depth increases in order to provide better and greater attachment on cancellous bone. This situation was analyzed and the results are shown in Tab. 6 with reference values of Tab. 2 and Tab. 3.

Table 6 - Maximum Effective Stress on conical body implants

	Region	Without threads	Uniform trapezoidal threads	Variable trapezoidal threads
Vertical force	Neck of the implant	7.83337E-02 GPa	7.77579E-02 GPa	7.70698E-02 GPa
	Cortical bone	3.29410E-02 GPa	3.45575E-02 GPa	3.41644E-02 GPa
	Implant base	---	---	1.43175E-01 GPa
Oblique force (45º)	Neck of the implant	9.94647E-01 GPa	9.90663E-01 GPa	9.87807E-01 GPa
	Cortical bone	2.02169E-01 GPa	2.00221E-01 GPa	1.99662E-01 GPa

FE analyses show that the proposed implant with trapezoidal threads and variable horizontal dimension reduces the effective stress level on the neck and promote its distribution until the implant base, what means that this geometry may be suitable for practical use. Regarding the results when considering masticatory oblique forces, there are reductions on both strains: on the neck of the implant and on its surroundings.

4. CONCLUSIONS

This research aims to evaluate geometries traditionally used for implant components and their correlation with the stress level developed on bone tissue. The desired situation is the one in which stresses are well distributed throughout the implant body and not only concentrated in critical areas as observed in practical situations of Implantology: the neck of the implant and its contact with the cortical bone.

Preliminary studies considering three different implants geometries without threads (the cylindrical, the cylindrical-conical and conical body) were used as a reference only, since threads are of extreme importance to fix and to ensure a better efforts distribution on implants surroundings. After these analyses, the same geometries with trapezoidal threads were studied. The results showed that the implants with cylindrical body had presented effective stresses of minor magnitude.

Based on this information, investigations were done in order to correlate the threads geometries with the distribution of effective stresses along the implant body. So, implants with cylindrical body and trapezoidal, triangular and rectangular threads were evaluated. The most convenient stress distribution was observed in the presence of rectangular threads, responsible for reducing stresses on the neck and on the contact region of cortical bone, and for propagating stresses until the component base.

A subsequent study was conducted in order to evaluate an implant commercially used: a conical body with trapezoidal threads where horizontal dimension gradually grows with the implant depth. The numerical results confirm that this component produces a stress intensity reduction in critical regions, and an adequate distribution until its base. Recent works published in 2010 demonstrate similar conclusion (Faegh and Müftü, 2010; Huang et al., 2010).

These analyses are considered preliminaries and were conducted in compliance with the commercially available geometries. Further studies with higher level of complexity of the mathematical models will be conducted to ensure that proposals for new geometries of implants and threads can be made, as well as studies of employability of new materials. In this context, analysis considering a 3D model and dynamic masticatory forces are very important. In recent works Okumura et al. (2010) and Dgerliyurt et al. (2010) have applied these 3D analyses.

5. ACKNOWLEDGEMENTS

To Prof. DSc. Delane Maria Rego for the contribution in discussing the ideas of this research.

6. REFERENCES

Araújo, L.A.; Figueiredo, A.M.G.; Las Casas, E.B.; Buono, V.T. “Implantes odontológicos com *abutment* de liga superelástica de NITI – Uma análise numérica comparativa com implantes convensionais”. SIMMEC 2008 – Simpósio em Mecânica Computacional 2008, Anais em CD-ROM.

Degerliyurt, K.; Simsek, B.; Erkmen, E.; Eser, A. “Effects of different fixture geometries on the stress distribution in mandibular peri-implant structures: a 3-dimensional finite element analysis”. Oral Surg Oral Med Oral Pathol Oral Radiol Endod, Vol. 110, pp. e1-e11, 2010.

Erasian, O.; Inan, Ö. “The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis”. Clin. Oral Invest., Vol. 14, pp. 411-416, 2010.

Faegh, S.; Müftü, S. “Load transfer along the bone-dental implant interface”. Journal of Biomechanics, Vol. 43, pp. 1761-1770, 2010.

Huang, H.-L.; Hsu, J.-T.; Fuh, L.-J.; Lin, D.-J; Chen, M.Y.C. “ Biomechanical simulation of various surface roughnesses and geometric designs on an immediately loaded dental implant”. Computers in Biology and Medicine, Vol. 40, pp. 525-532, 2010.

Okumura, N.; Stegaroiu, R.; Kitamura, E. Kurokawa, K.; Nomura, S. “ Influence of maxillary cortical bone thickness, implant design and implant diameter on stress around implants: A three-dimensional finite element analysis”. Journal of Prosthodontic Research, Vol. 54, pp. 133-142, 2010.

Van Staden, R.C.; Guan, H.; Loo, Y.C. “ Application of finite element method in dental implant research”. – Comput. Methods Biomech. Biomed. Engin., 9(4): pp. 257-270, 2006.

Wang, J. C.; Huang, S. F; Lin, C. L. “Biomechanical responses of endodontically treated tooth implant-supported prosthesis”. Article in press in Journal of Endodontics.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.