

## MECHANICAL AND COMPUTER SIMULATION TESTS ON SOME EXPERIMENTAL MODELS OF ORAL IMPLANTS

M. SABAU<sup>1</sup>, CAMELIA SZUHANEK<sup>2</sup> N. FAUR<sup>3</sup>, A. CERNESCU<sup>3</sup>, M. GHIȚĂ, C., IGNA<sup>1</sup>

<sup>1</sup> Faculty Of Veterinary Medicine Timișoara

<sup>2</sup> University Of Medicine And Pharmacy „Victor Babeș” Timișoara

<sup>3</sup> Polytechnic University of Timișoara

### Summary

This paper purpose is to evaluate, through finite elements analysis, the reactions of the structures placed at bone-implant interface and through mechanical tests of the strength of orthodontic implants for temporary anchorage (1, 3, 4). The implants were tested mechanically concerning their axial tension, compression, bending and torsion (1, 4) on Multitest and Vortex-X stands .

### Materials and methods

The mechanical tests were made at CIDUCOS laboratories of Polytechnic University of Timisoara which has the quality system implemented, in conformity with the European demands from SR EN ISO/CEI 17025:2001 standard.

Mechanical tests: traction, compression and bending were made on the same equipment, with specific devices. The stand is a Multitest 5-I and has the following characteristics: maximum strength 5000 N, effective run 600 mm, maximum fixture distance 610 mm, maximum piece dimension 730 mm, speed range 1-500mm/min. The torsion tests were made on Vortex-X stand that has the characteristics: maximum moment 10Nm, maximum displacement 2500 rot, maximum fixture distance 350 mm, speed range 0,1-20 rpm.

For computer simulation we have used a three-dimensional geometrical model of the orthodontic implant and of the bone structure surrounding it performed with the help of Solid Works software. The 3D models were then imported in Ansys analyze software resulting a computational model. On this model we have applied orthodontic forces similarly to the ones applied within the clinical treatment with extra dental anchorage (3).

The bone model was made of spongius surrounded by cortical bone and has been modeled to have an elastic linear behaviour, in conformity with the previous studies (Gallas, EJO 2005).The characteristics of the bone structure are listed in table 1.

Table 1

**Characteristics of bone structure**

| Material       | Young elasticity module (MPa) | Poisson coefficient |
|----------------|-------------------------------|---------------------|
| Cortical bone  | 13 760                        | 0.30                |
| Spongious bone | 7 930                         | 0.30                |

The forces applied have disto-mesial direction and 20N intensity. The software permitted the equivalent tension Von Misses to be calculated in concordance with specifically shape modifying energy, and the results have been graphically represented through a colored spectrum corresponding to a value of equivalent tension.

**Results and discussions**

The experimental implants are made of titanium alloy  $TiAl_6V_4$ . The critical cross section is the one corresponding to 2,5 mm diameter (the bottom of the thread, with a 2 mm course of thread and 0,5 mm high.

Mechanical strength on traction of  $TiAl_6V_4$  is:

$$R_m = (700...1200) N / mm^2$$

The values of the intermittent stress, which must not exceed the breaking strength at fatigue, bending and torsion, have a significant importance in these implants. These strengths are smaller then mechanical strength on traction  $R_m$  and have the values

$$\sigma_0 = (525...1080) N / mm^2$$

$$\tau_0 = (280...600) N / mm^2$$

If the fatigue strength has been exceeded, the implant will break without deforming because of the internal atomically bonds destroy. The final reports, caused by the implant's form, are due to structure defects (segregations, superficial fissures). The fractures following overstress are rigid type. They are present at the place where, because of the shape of implant, a concentration of forces appears, and acts multiaxially.

The medium value of the force at which the break took place at pulsatory bending challenge, after five determinations was:

$$F_{i_1} = 183,98 N$$

Corresponding to these forces, the pulsatory bending breaking stress is:

$$\sigma_{01ef} \approx 610 N / mm^2$$

These values are placed between the accepted limits:

$$\sigma_0 = (525...1080) N / mm^2$$

The maximum moment at which the break took place on pulsatory twisting was:

$$M_{t_1} = 4,579 Nm$$

Corresponding to these values the pulsatory torsion strength is:

$$\tau_{01} \approx 307 N / mm^2$$

These values are placed between the accepted limits:

$$\tau_0 = (280...600) N / mm^2$$

The breaking took place in the proximity of the thread part, where the bending moment has maximum value.

In case of computer simulation, the positive or negative values indicate tension and compression. The color scale goes from blue (0 value) to red (maximum value). It may observe the unequal repartition of Von Misses tension, with concentration at bone-implant interface and at the implant's neck.

### Conclusions

1. On both mechanical and computer simulation tests it may observe a concentration of the forces at the implant neck. The breakings took place near the end of the thread part, where the bending moment has a maximum value.
2. Fractures due to overstress are rigid. They are present at the place where, because of the shape a force concentration that appears and acts multiaxial.
3. The values of breaking force at pulsatory bending strength, pulsatory torsion strength are between normal values.

### References

1. **Bazkaya, D., Muftu, S.**, Mechanics of the tapered interference fit in dental implants, Journal of Biomechanics No 36, 1649-1658, 2003
2. **Bratu, D., Ciosescu, D., Romănu, M., Colojoară, C., Ardelean-Lerețer, M., Uram-Țuculescu, S.**, Titaniul în stomatologie, Stomatologia, vol XL, 1, 10-24, București, 1992.
3. **Marckerle, J.**, Finite Element Modelling and Simulations in Dentistry: A Bibliography 1990-2003, Computer Methods in Biomechanics & Biomedical Engineering, Taylor & Francis, Vol. 7, Nr 5, 2004.