Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XXXIV, págs. 1161-1167 (artículo completo)
Sebastián Giusti, Martín Pucheta y Mario Storti (Eds.)
Córdoba, 8-11 Noviembre 2016

BIOMECHANICAL ANALYSIS OF BIORESORBABLE MAXILLOFACIAL PLATES

Ezequiel M. Perez[1,2]

[1] *National Scientific and Technical Research Council (CONICET), Av. Rivadavia 1917, (C1033AAJ), Buenos Aires, Argentina, info@conicet.gov.ar http://www.conicet.gov.ar*

[2] *Plastics Research and Development Center, National Institute of Industrial Technology (INTI-Plásticos), Av. Gral. Paz 5445, (B1650KNA), San Martín, Buenos Aires, Argentina, plasticos@inti.gob.ar, http://www.inti.gob.ar/plasticos/*

Keywords: Polylactic acid (PLA), Biomedical applications, FEA simulations.

Abstract. Bioresorbable devices are actually extensively applied for bone healing. These products are exposed to several physical, chemical and mechanical requirements. For this reason, mechanical performance under different loading conditions should be exhaustively analyzed in order to warrant the long term success. In this work, mechanical behavior of polylactic acid (PLA) maxillofacial mini-plate implant was investigated by mechanical tests and numerical simulations. The obtained results showed, that thread profile and screws location, respect to the broken bone plane, represent key factors for stress distributions. On the other hand, experimental tests and simulations exhibited similar displacement values.

1 INTRODUCTION

Bioresorbable devices are actually used in orthopedic, craniofacial, oral, maxillofacial and reconstructive surgeries. These mini products include a wide range of screws, plates, pins, etc and they are produce by biodegradable polymers (polylactic acid (PLA), polyglycolic acid (PGA), among others) their composites and blends (Jamshidian et al., 2010; Shikinami and Okuno, 1999; Shikinami and Okuno, 2001). These products are exposed to several physical, chemical and mechanical requirements during bone healing process. Mechanical performance under different loading conditions should be exhaustively analyzed in order to warrant the long term success (Böstman and Pihlajamäki, 2000; Shikinami and Okuno, 2001). Particularly, standardized testing procedures and Finite Element Analysis (FEA) are well known as powerful tool for product behavior evaluation (Atik et al, 2016; Esen et al., 2008). In this work, the effect of maxillofacial implant geometry on the stress distribution was evaluated.

2 MATERIALS AND METHODS

Mechanical performance of PLA-based maxillofacial mini-plates and screws was analyzed by experimental tests and FEA (Finite Element Analysis).

2.1 Sample preparation

Maxillofacial mini-plate and screws models were based on commercially devices. Mandible bone model was based on a 3D scanning. A commercially 3D printer (Chimak 3D) was used to obtain PLA-based: mini-plates, mandible bone and pins experimentally tested.

2.2 Mechanical tests

The maxillofacial mini-plate was 4 point-bending tested following the ASTM F 2502-11 protocol. The implant system stability was evaluated on a PLA-based broken mandible under flexo-compression test. A universal testing machine (Tinius Olsen H50-KT model) at a crosshead speed of 5mm/min was used for mechanical tests.

2.3 Finite Element model

Three-dimensional geometrical models were analyzed by FEA (CATIA V5R20®). The models considered were: i) maxillofacial mini-plate under 4 point-bending testing condition, ii) mini-plate fixed by pins into a PLA homogeneous mandible, iii) mini-plate fixed by screws into a PLA homogeneous mandible and iv) mini-plate fixed by screws into a mandible bone. The elastic properties of the different materials considered are listed in Table 1(Simsek et al., 2006). Materials used were considered as isotropic, homogeneous and linearly elastic.

	Elastic modulus (GPa)	Poisson's ratio (υ)	Tensile yield stress (MPa)
PLA	2.7	0.36	53
Cortical bone	14.8	0.3	--
Trabecular bone	1.85	0.3	--

Table 1: Material elastic properties used for FEA models.

3 RESULTS AND DISCUSSION

Mechanical behavior of mini-plate and screws was analyzed. Experimental tests and FEA were performed. All EQV distributions and displacement values informed were obtained with a load applied of 1N.

3.1 Mini-plate 4-point bending test

The FEM used was composed of 450000 4-nodes tetrahedral elements, including 390000 elements for the mini-plate. Mechanical performance of PLA mini-plate was tested by 4-point bending test and FEA. Figure 1 shows the simulated model, experimental configuration, Maximal Equivalent Von Mises stress (EQV) distribution and obtained curves. Mechanical test was interrupted below maximal load due to sample sliding. EQV maximal values were placed at the hole surface. On the other hand, 4-point bending test and FEA displayed similar slopes into the load-displacement linear region.

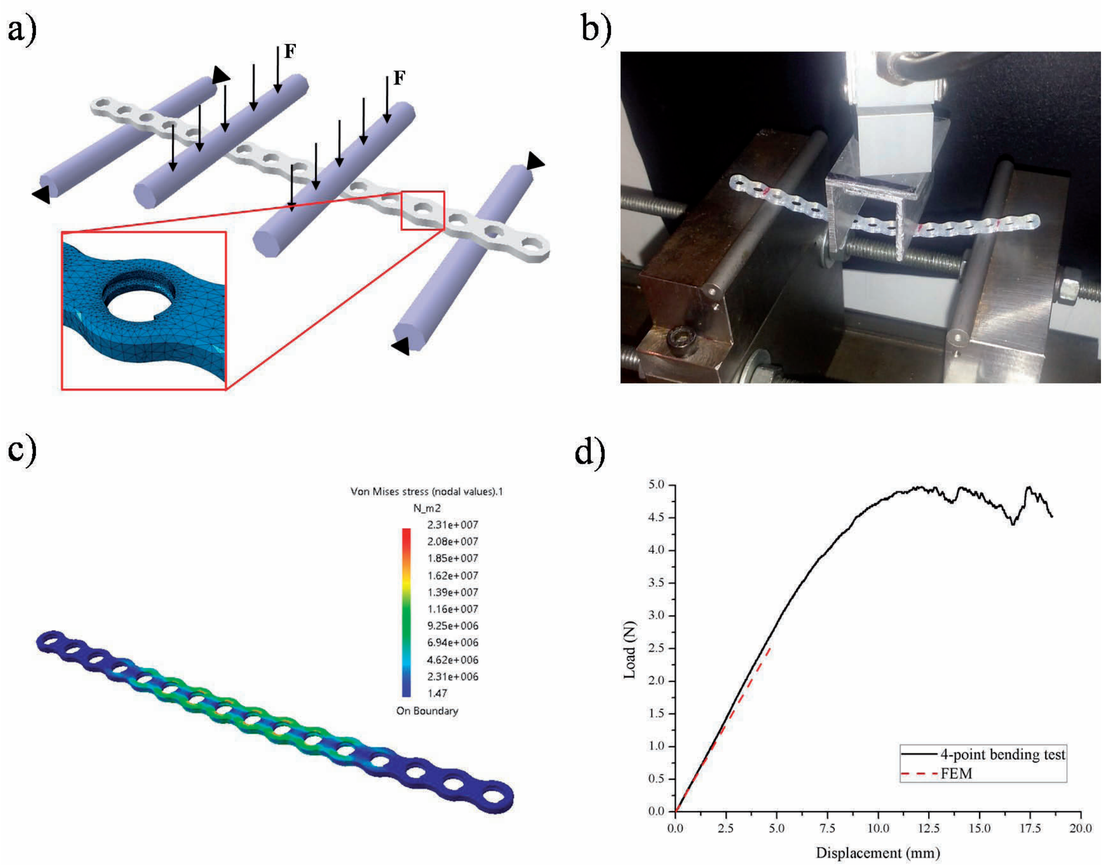

Figure 1: Mini-plate 4-point bending test

3.2 Mini-plate fixed by pins into PLA homogeneous mandible

The FEM used was composed of 240000 4-nodes tetrahedral elements, including 23400 elements for the mini-plate and 6800 elements for the pin. A refined mesh was considered on the contact regions between components. Figure 2 shows the simplified implant system considered. The mini-plate was fixed by 4 pins into the PLA-based broken mandible (Figure 2.a) for flexo-compression testing. The maximal EQV values were coincident with mini-plate failure location. In addition, mechanical test and FEA displayed similar slopes into the load-displacement linear region.

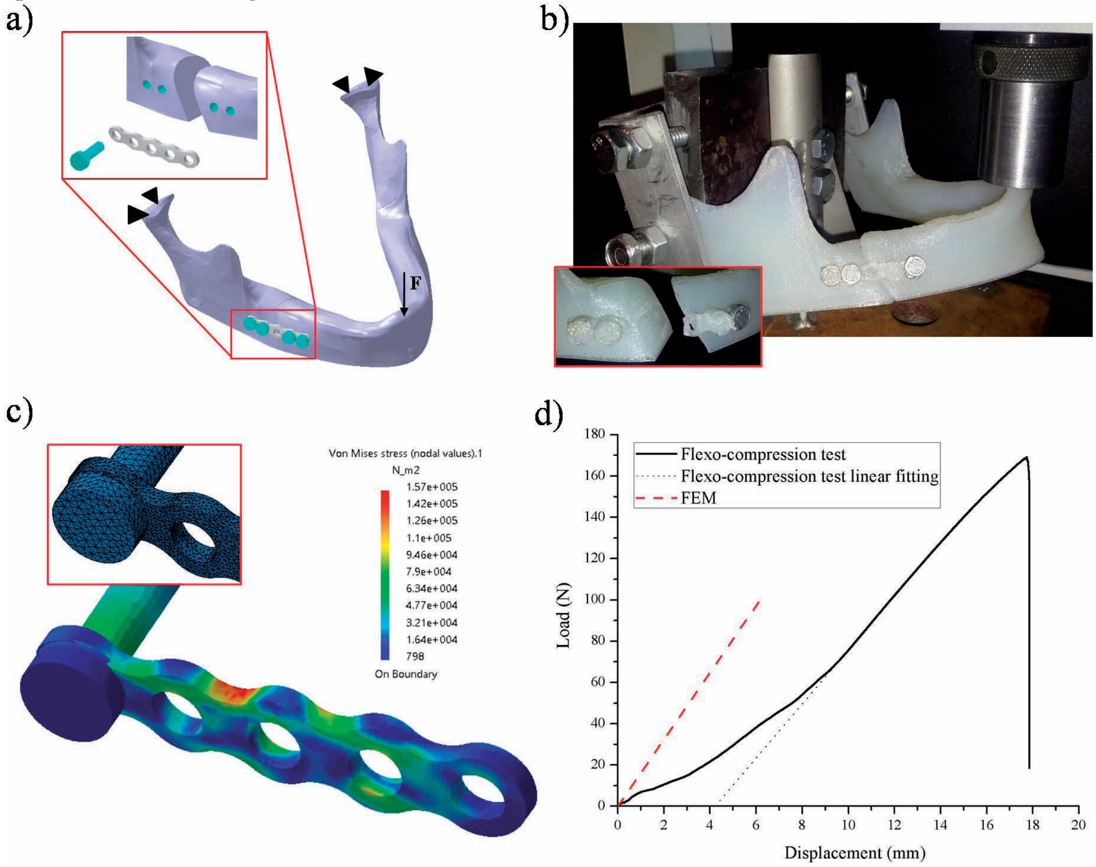

Figure 2: Simplified mini-plate system into PLA homogeneous mandible

3.3 Mini-plate fixed by screws into PLA homogeneous mandible

The FEM used was composed of 220000 4-nodes tetrahedral elements, including 24000 elements for the mini-plate and 8900 elements for the screw. A refined mesh was considered on the contact regions between components. The simulated mini-plate was fixed by 4 screws into the PLA-based broken mandible (Figure 3.a). For the different screws, the maximal EQV values were placed at the mini-plate contact surface. But, these values were no constant with a variation of 50%, approximately. The highest value corresponds to the screw 2, while lowest to the screw 4. For the mini-plate, the maximal EQV value was placed at the threaded hole. This value was 900%, approximately, increased respect to the simplified implant without threaded holes. Increased maximal displacement value (30%, approximately) was obtained, compared to simplified mini-plate implant system (Section 3.2).

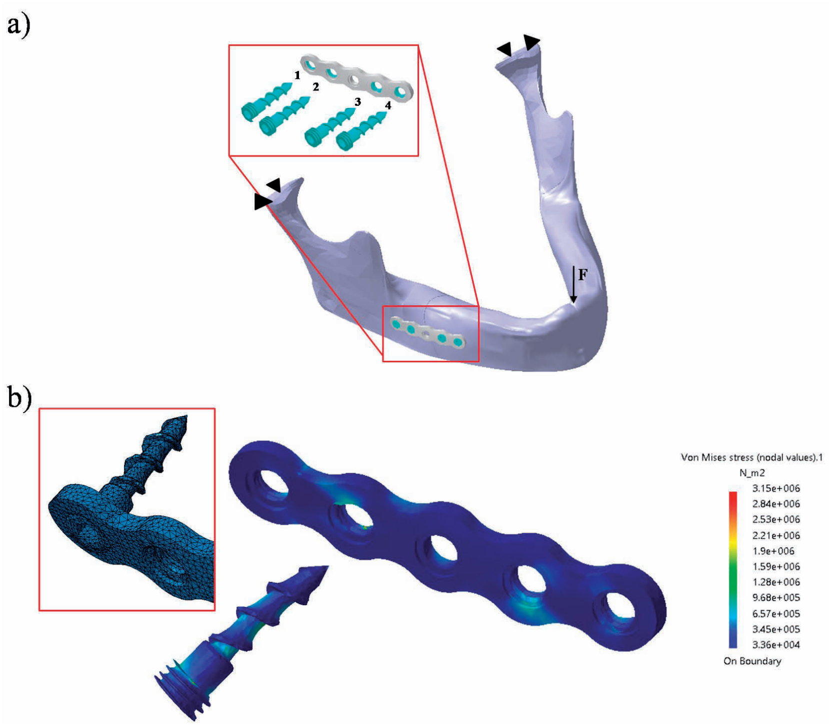

Figure 3: Mini-plate system into PLA homogeneous mandible

3.4 Mini-plate fixed by screws into mandible bone structure

The FEM used was composed of 330000 4-nodes tetrahedral elements, including 24000 elements for the mini-plate and 8900 elements for the screw. A refined mesh was considered on the contact regions between components. The simulated mini-plate was fixed by 4 screws into the broken mandible bone (Figure 4.a). For the different screws, the maximal EQV value was 60%, approximately, reduced compared with data previously described (Section 3.3). In a similar way, the maximal EQV values were placed at the mini-plate contact surface with a reduction of 50%, approximately, between the screws 2 and 4. For the mini-plate, the maximal EQV value was placed at the threaded hole. This value was 80%, approximately, reduced respect to data previously described (Section 3.3). On the other hand, reduced maximal displacement value (80%, approximately) was obtained, compared with the implant system fixed by screws into the homogeneous mandible.

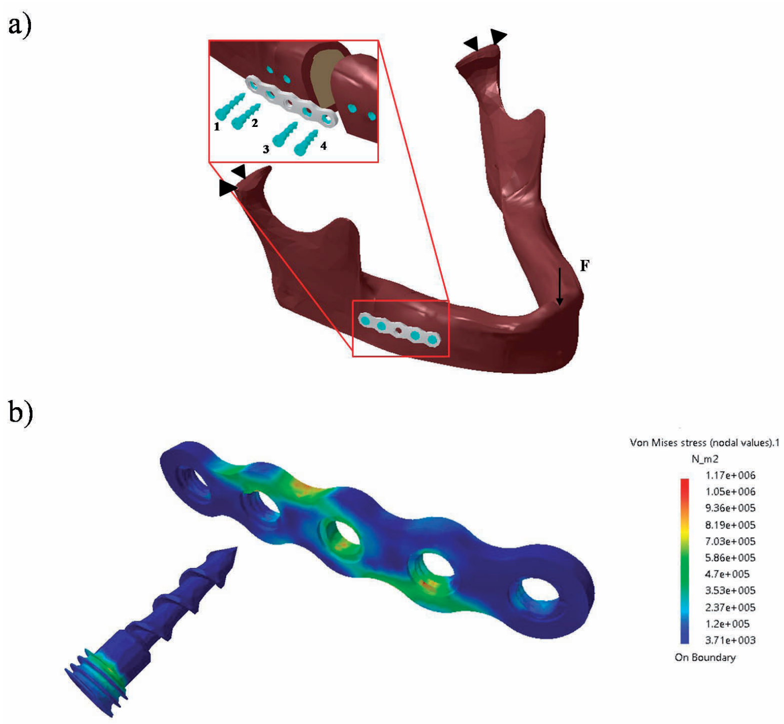

Figure 4: Mini-plate system into mandible bone structure

In addition, for all analyzed situations the maximal EQV stress values did not reach the PLA yield stress for any component of the analyzed systems allowing the linear elastic behavior adopted.

4 CONCLUSIONS

Mechanical performance of a maxillofacial mini-plate implant system was analyzed. Experimental and numerical simulations of a simplified implant systems displayed similar liner elastic behaviors suggesting the FEM models accuracy.

The thread profile of mini-plate and screw represents a key factor for stress distributions, close related to the mechanical performance success. On the other hand, the screws location, respect to the broken bone plane, affects the EQV distribution. All obtained EQV stress values were lower to the PLA yield stress allowing the linearly elastic behavior adopted.

EQV and displacement distributions variations can be observed if mandible bone structure is considered.

ACKNOWLEDGMENTS

The author of this work wants to really acknowledgment the 3D printing made by NanoFab of FAN (Argentinean Foundation of Nanotechnology).

REFERENCES

Atik, F., Ataç, M.S., Özkan, A., Kılınç, Y., Arslan, M., Biomechanical analysis of titanium fixation plates and screws in mandibular angle fractures. *Nigerian Journal of Clinical Practice,* 19:386-390, 2016.

Esen, A., Ataoglu, H., Lokman G., Comparison of stability of titanium and absorbable plate and screw fixation for mandibular angle fractures. *Oral Surgery, Oral Medicine, Oral Pathology, Oral Radiology,* 106:806-811, 2008.

Jamshidian, M., Tehrany, E.A., Imran, M., Jacquot, M., Desorby S., Poly-Lactic Acid: Production, Applications, Nanocomposites, and Release Studies. *Comprehsnsive Reviews in Food Science and Food Safety,* 9:552-570, 2010.

Shikinami, Y., Okuno, M., Bioresorbable devices made of forged composites of hydroxyapatite (HA) particles and poly-L-lactide (PLLA): Part I. Basic characteristics. *Biomaterials*, 20:859-877, 1999.

Shikinami, Y., Okuno, M., Bioresorbable devices made of forged composites of hydroxyapatite (HA) particles and poly-L-lactide (PLLA). Part II: practical properties of miniscrews and miniplates. *Biomaterials*, 22:3197-3211, 2001.

Simsek, B., Erkmen, E., Yilmaz, D., Eser, A., Effects of different inter-implant distances on the stress distribution around endosseous implants in posterior mandible: A 3D finite element analysis. *Medical Engineering & Physics*, 28:199–213, 2006.