11º CONGRESSO NACIONAL DE BIOMECÂNICA
A. P. G. Castro, C. Pina, P. R. Fernandes, C. Santos, J. Folgado & A. Amaro (Eds)
Sesimbra, Portugal, 21-22 de Fevereiro, 2025

BIOMECHANICAL EVALUATION OF A PREMOLAR TOOTH SUBJECTED TO APICOECTOMY

I. S. Fareleiro [1], M. A. Neto [1] e A. Messias [1,2], J. M. Santos [3s]

[1] Universidade de Coimbra, Centro de Engenharia Mecânica, Materiais e Processos (CEMMPRE-ARISE), Departamento de Engenharia Mecânica, 3040-248 Coimbra, Portugal

2 Universidade de Coimbra, Faculdade de Medicina, 3040-248 Coimbra, Portugal

3 Universidade de Coimbra, Faculdade de Medicina, Instituto de Endodontia, 3000-075 Coimbra, Portugal

inessdfs@gmail.com; augusta.neto@dem.uc.ptt; ana.messias@uc.pt ; jsantos@fmed.uc.pt

PALAVRAS-CHAVE: Finite Element Analysis, Apicoectomy, Endodontic, Microsurgery

1 INTRODUCTION

Apical surgery or endodontic microsurgery (EMS) is a surgical intervention that involves the removal of a tooth's apex. EMS is most often used to treat persistent periapical infection caused by apical periodontitis and failed root canal therapy as an alternative to non-surgical endodontic retreatment. Until recently, apical surgery was viewed with scepticism due to the lack of knowledge on the factors with impact in its outcome, being mostly practiced as a last resort to preserve a tooth that went to various previous treatments. Modern techniques made EMS a methodically conducted procedure with reliable outcomes, thus associated high success rates [1]. Despite these advancements, the impact of root resection on tooth biomechanics remains insufficiently understood.

Finite element analysis research has been done on the different aspects of EMS, such as Apical Resection (AR) length, periodontal bone loss length, resection angle, root-end preparation material and graft and occlusion type. The most often studied tooth has been a central incisor, all applied loads have been static and material properties have all assumed to be linear elastic, apart from Richer et al. work [2] that assumed the periodontal ligament as hyperelastic.

This research aims to evaluate the influence of apical resection length on the biomechanical response of a single-rooted maxillary teeth, a second premolar, using finite element analysis (FEA).

2 MATERIALS AND METHODS

A CBCT scan of a young (21 years old) female patient whose dentition had no imperfections was selected in the data base of the Faculty of Medicine of the University of Coimbra for this project. The CBCT images were obtained through an i-Cat® FLX Cone Beam 3D (Kavo-kerr, Germany) system, with a scan time of 26.9 seconds, a field of view of 16 x 10 cm and a voxel size of 0.25 mm. The resultant images of the Intraoral Scan were acquired with the Cerec Omnicam® system with scan dimensions of 228 x 16 x 16 mm, using optical triangulation acquisition technology and confocal microscopy.

As to replicate root canal treatment, access cavities were made on the original models of the teeth, by the subtraction of bodies created on Autodesk Inventor® software (Figure 1). Measurements

were made on the pulp chambers of each tooth to accomplish the best possible fit. The shape of the access cavities was based on diagrams and photos of the obturation process.
The Parasolid models were imported into the finite elements software (ADINA AUI version 9.8.2, Bentley Systems, Exton, USA).All nonlinear quasi-static analyses used the Automatic Time-Stepping Method (ATS), the large Displacements/Rotations option and Low-Speed Dynamics (LSD).
The materials were assumed to be isotropic, homogenous and linearly elastic, apart from the periodontal ligament, which was assumed to be hyperplastic. To simulate contact during the closing phase of the chewing cycle, a constant load of 500 N was uniformly distributed on an indenter surface, as seen on Figure 1. Contact conditions were established, assuming material continuity, as all bodies are in direct contact with one another.

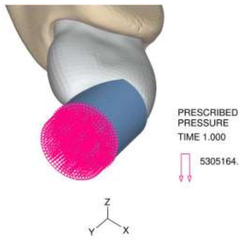

Figure 1 – applies loads in the indenter's surfaces

The mesh densities for all components were defined and after test simulations, the high stress concentration areas were refined. Non-critical areas were made to have lower mesh densities to conserve computational resources. The total number of nodes and finite elements changed among models since the material volume also changed.

3 Results and Discussion

The patterns of stress distribution show a localized increase in the cervical area and present more evenly distribution on the buccal side of the premolar root, probably due to the load incidence on the buccal cusp. In the apically resected models, the previous critical areas reached the resected root surface. The bone models presented their highest stress concentrations on the alveolar crest area, and the apical resections did not seem to affect the stress distributions.
Tooth models subjected to endodontic treatment displayed high stresses, independently from the increase of AR length. Tooth von Mises stress values remained constant, despite the increases in AR lengths.

Acknowledgements

This research is sponsored by national funds through FCT – Fundação para a Ciência e a Tecnologia, under the project UIDB/00285/2020 and LA/P/0112/2020.

Referências

[1] D. Pinto, A. Marques, J.F. Pereira, P.J. Palma, J.M. Santos, (2020). " Long-Term Prognosis of Endodontic Microsurgery—A Systematic Review and Meta-Analysis", Medicina, vol. 56(9), 2022, https://doi.org/10.3390/medicina56090447.

[2] R. Richert, J. C. Farges, J. C. Maurin, J. Molimard, P. Boisse, and M. Ducret, (2022). "Multifactorial Analysis of Endodontic Microsurgery Using Finite Element Models", Journal of Personalized Medicine, vol. 12(6), 2022, https://doi.org/10.3390/jpm12061012.