PRIMARY FIXATION OF DENTAL IMPLANTS IN BONE SURROGATE - FINITE ELEMENT ANALYSIS CONSIDERING INSERTION DAMAGE

Baixuan Yang (1), Ainara Irastorza-Landa (2), Peter Heuberger (2), Heidi-Lynn Ploeg (1)

1. Queen's University, Canada; 2. Nobel Biocare Service AG, Switzerland

Introduction

Primary fixation, mechanical engagement between implant and bone under functional loading, can be evaluated by implant motion (IM). To improve the understanding of primary fixation, computational models such finite element analysis (FEA) have been developed; however, one common limitation in current FEAs is an assumption of intact bone properties without considering stress or damage generated from insertion [1]. Therefore, the objective of this study was to develop and validate an accurate FEA to investigate the primary fixation of threaded dental implants in a rigid polyurethane (PU) foam considering effects of the insertion process.

Method

Two implant designs: parallel-walled (Brånemark MkIII TiUnite NP 3.5×13 (P1) and Brånemark MkIII TiUnite WP 5.0×13 (P2), Nobel Biocare), and tapered implants (NobelActive® NP 3.5×13 (T1) and NobelActive RP 4.0×13 (T2), Nobel Biocare) were inserted into PU foam blocks (40×40×8 mm^3, Ø 2.4/2.8 mm or Ø 3.2/3.6 mm, 20 PCF, Sawbones) in dry condition and room temperature (MACH-1 V500CST, Biomomentum). Fixation tests were conducted 2 days after insertion with 2 loading protocols: axial and 30-degree off-axis loading (Electroforce 5500, TA instruments) with 5 repeats for each implant design (Fig.1a). The IM was measured as vertical displacement (v) using a deflectometer (3540-001M-ST, Epsilon Technology Corp); and, as implant rotation angle (θ) analysed for off-axis loading using 2D tracking from video (Canon EOS Rebel SL2 DSLR, EF-S 18-55mm Lens, Canon. Inc., Digital Image Correlation Engine version 2.0 [2]). A series of non-linear explicit FEAs were conducted including an insertion step and followed by a loading step (Abaqus 2017, Simulia). The insertion step was developed and validated against surface strains of the PU foam measured in the insertion tests [2]. After the insertion step, a v measured from the tests was applied on the implant reference point (RP) for the axial loading, while an off-axial force and a θ were applied on the implant RP for the off-axis loading (Fig.1b).The PU foam was modelled with a linear elastic (Young's modulus of 123 MPa) and multilinear plastic material (8.06 MPa yield strength, and 30% fracture strain, from compression test data) with hexahedral elements with incompatible modes. The implant was modelled as a rigid body with bilinear rigid quadrilateral elements (Fig. 1b). An element-based surface contact algorithm was used with penalty formulation and friction coefficient, 0.61, between the implant and PU foam.

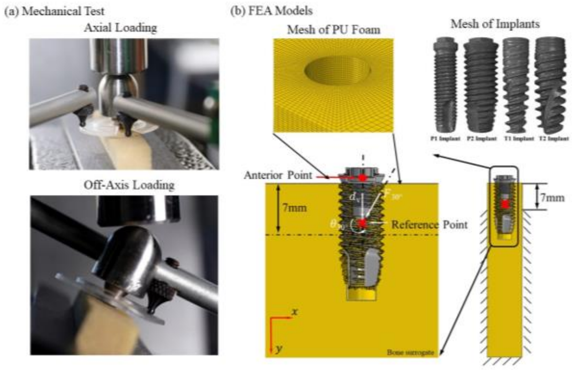

Figure 1: (a) Fixation Test Setup. (b) FEA Models.

Results

The force-displacement curves predicted by the FEA were comparable to the experimental results (Fig.2). Differences in the PU foam-implant stiffness between FEA and experimental results ranged from 3.64% - 17.2% for axial loading, and for the off-axis loading 20.4%-20.7% for P1, P2 and T1 implants, and 45.0% for the T2 implant.

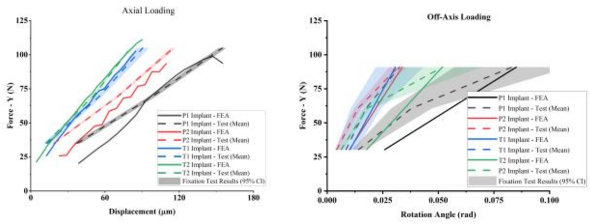

Figure 2: The bone-implant stiffness of axial loading (left) and off-axis loading (right). Dash line: test; Solid line: FEA

Discussion

The FEA model, which considers the foam deformation and damage near the implant due to insertion, achieved an overall good prediction of the stiffness in comparison to mechanical test results, particularly under vertical loading. Results were specific to the implant designs and PU foam properties used in the study.

References

1. Ovesy M et al, J Mech Behav Biomed Mater, 98:301-310, 2019.
2. Yang B et al, ASME VVUQ, 2023.

Acknowledgements

We acknowledge funding from Nobel Biocare Services AG and the support of the Natural Sciences and Engineering Research Council of Canada, Canadian Foundation of Innovation, and Ontario Research Fund.