EFFECT OF CUTTING FLUTE ON DENTAL IMPLANT INSERTION PROCESS WITH EXPLICITFINITE ELEMENT ANALYSIS

Baixuan Yang (1), Ainara Irastorza Landa (2), Peter Heuberger (2), Heidi-Lynn Ploeg (1)
1. Mechanical and Materials Engineering, Queen's University, Canada; 2. Nobel Biocare Services AG, Switzerland

Introduction

Finite element analysis (FEA) has been widely applied to investigate dental implant biomechanics and predicate bone loading which is impossible to measure *in vivo*. Accuracy of computational models must be established by model verification and validation (V&V). However, in this field, only 48 studies satisfied V&V requirements out of 601 articles published from 1997 to 2016, and nine of them were indirectly validated using past literature [1]. In this study, implant insertions were modelled by explicit nonlinear FEAs and mechanical tests were conducted to investigate the effect of cutting flute (CF) on insertion mechanics. FEA results were validated by strain distributions measured by digital image correlation (DIC) during mechanical testing.

Methods

Dental implants (with and without CF, NobelActive® NP 3.5 x 13 mm, Nobel Biocare AB, Göteborg, Sweden) were inserted into prepared bone surrogate (40 × 40 × 8 mm, density: 0.32 g/cm^3, Sawbones, WA, USA) with 2.4/2.8 mm pilot hole under controlled angular and axial displacement rates (4.5 rpm and 0.3 mm/s, TA Electroforce 3230 ,TA instruments, MN, USA). Surface deformation was measured using stereo DIC (two cameras, VIC-3D, Correlated Solutions, Inc. SC, USA). The two insertion tests were modelled and analysed using Abaqus 2017 (Simulia, RI, USA). Linear elastic and multilinear plastic material model approximately represented the bone surrogate. Implants and fixtures were modelled as rigid bodies. Bone surrogate Young's modulus (123 MPa), yield strength (8.06 MPa) and fracture strain (30%) were determined from uniaxial compression testing. Damage evolution started at the onset of the plasticity plateau due to local buckling and micro damage. Failure (densification) was assumed at 30% strain. The bone surrogate was meshed with 724,582 linear hexahedral elements (global edge length 0.5 mm) with incompatible modes. The osteotomy region was meshed with 0.1 mm local edge length (Fig. 1). Implants were meshed with 26,754 and 8,629 bilinear rigid quadrilateral elements (global edge length 0.1 mm) with CF and without, respectively. A reference point of the fixture was fixed in all degrees of freedom. The rigid implant was only allowed to rotate and move downwards in the y-direction with constant angular velocity (4.5 rpm) and axial velocity (0.3 mm/s). General contact algorithm was used with element-based surfaces between bone and implant and applied to new exposed element-surface after element deletion. Penalty formulation was adopted for friction tangential behaviour between bone and implant with a constant coefficient of 0.61 [2]. Rough formulation (no slip) was used between bone and the fixture.

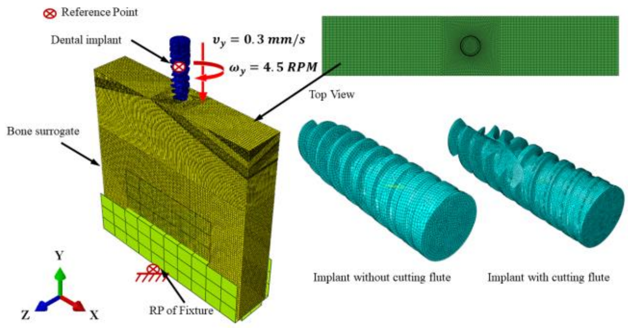

Figure 1: Finite element model of insertion: mesh, loading and boundary conditions.

Results and Discussion

FEAs were validated by surface von Mises strain measured with DIC (Fig 2). CF reduced bone surrogate damage by 15% with 23.5 mm^3 versus 27.5 mm^3 damage volumes (starting plastic deformation, SDEG > 0.001) with and without CF (Fig 3).

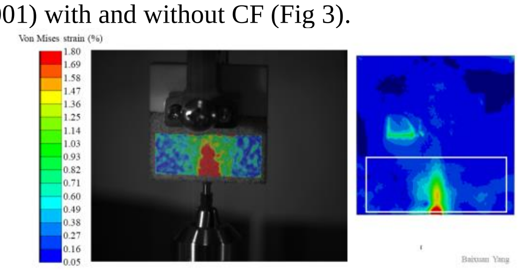

Figure 2: Surface von Mises strain with implant inserted. Left: DIC; Right: FEA.

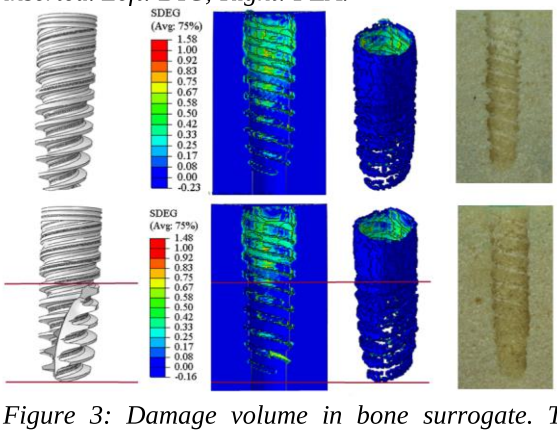

Figure 3: Damage volume in bone surrogate. Top: implant without CF; Bottom: implant with CF.

References

1. Yuanhan Chang et al, Int J Implant Dent, **4**:7-21, 2018.
2. Hone Guan et al, Finite Elem Anal Des, **47**:886-897, 2011.

Acknowledgements

We acknowledge funding from Nobel Biocare Services AG, the support of the Natural Sciences and Engineering Research Council of Canada (NSERC), Ploeg's Research Initiation Grant, and the Human Mobility Research Centre, Queen's University, Kingston, ON, Canada.