Analysis of an abutment screw-screwdriver interaction: a numerical and experimental study

C. Soares[1], A. Cortez[2], M. Sampaio[2], P. Gobbo[3], R. Natal[1] and J. Mendes[1]

[1]Faculdade de Engenharia, Universidade do Porto, Porto, Portugal (cras@fe.up.pt, rnatal@fe.up.pt, jgabriel@fe.up.pt)

[2]Celoplás, Plásticos para a Indústria S.A., Barcelos, Portugal (ana.cortez@celoplas.pt)

[3]Gadget Whisper Lda., Porto, Portugal (pietro@dentalnet.it)

Introduction

The abutment screw in dental implant assembly is an indispensable part for the correct functioning of the entire system.
The **SIMPLIFIED: Easy Tooth Abutment** project requires the design of a new head socket for the abutment screw and its compatible screwdriver to allow the correct tighten of the abutment to the implant. The aim of this study was to verify the design performance of the new interface, when the screw is tightened at the recommended torque (30 N.cm).

CAD Models

The new head socket and corresponding screwdriver were designed based on the project requirements and specifications. The respective 3D models and the system assembly are shown in Figure 1.

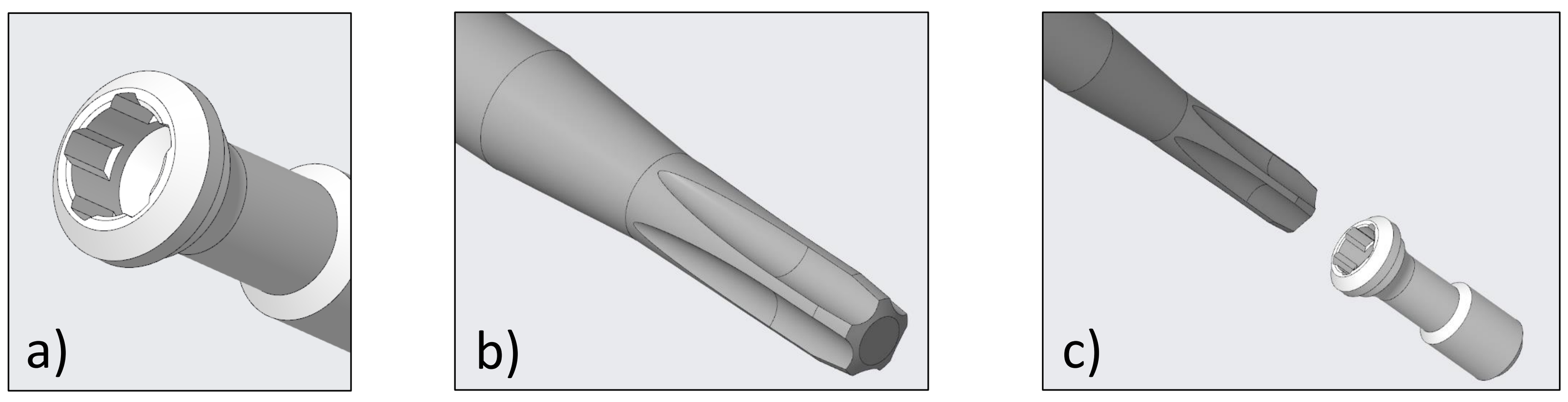

Figure 1. (a) New socket design for abutment screw; (b) Screwdriver; (c) System assembly

Computer simulation

To validate the mechanical performance of the new design, a finite element analysis (FEA) was applied to the assembly model.
For this purpose, a torque load of 30 N.cm (Figure 2a) was applied to the screwdriver body (Figure 2b), while the screw head was fixed (Figure 2c).
The interaction between the two components was simulated with the help of ABAQUS®/CAE 6.14-3 software, assuming a brief transient dynamic event (quasi-static simulation), due to the highly nonlinear behaviour introduced in contact simulation.
The material chosen for the screwdriver was Stainless Steel 420F considering its corrosion resistance and high strength obtained through heat treatment. The screw abutment material was chosen to be Titanium Grade 5.

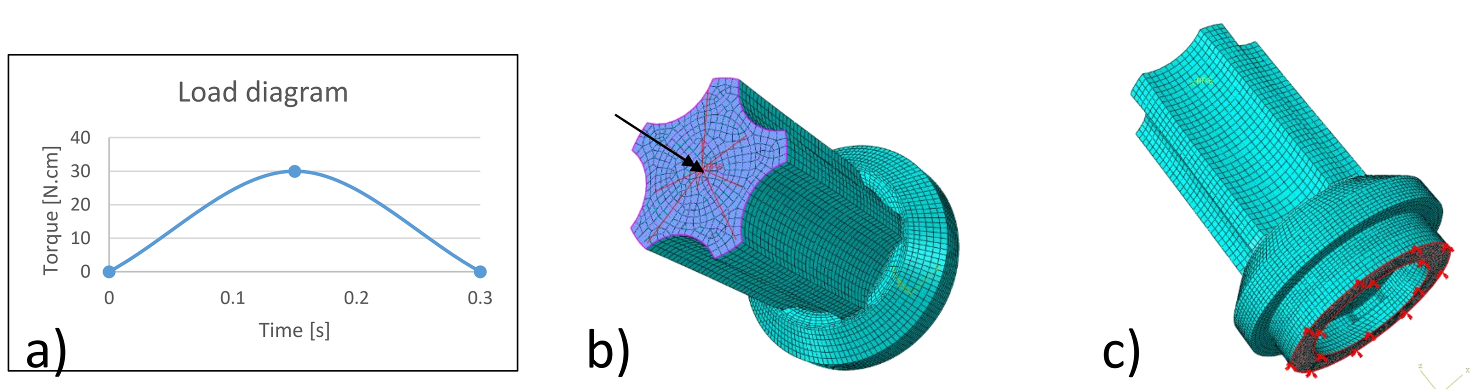

Figure 2. (a) Simulated applied load; (b) Application point of the torque load on the FEA model of the screwdriver; (c) Fixed boundary condition of the simplified screw head; The mesh was generated using a three-dimensional hexahedral element with reduced integration (C3D8R), consisting of 40273 nodes and 34568 total number of elements.

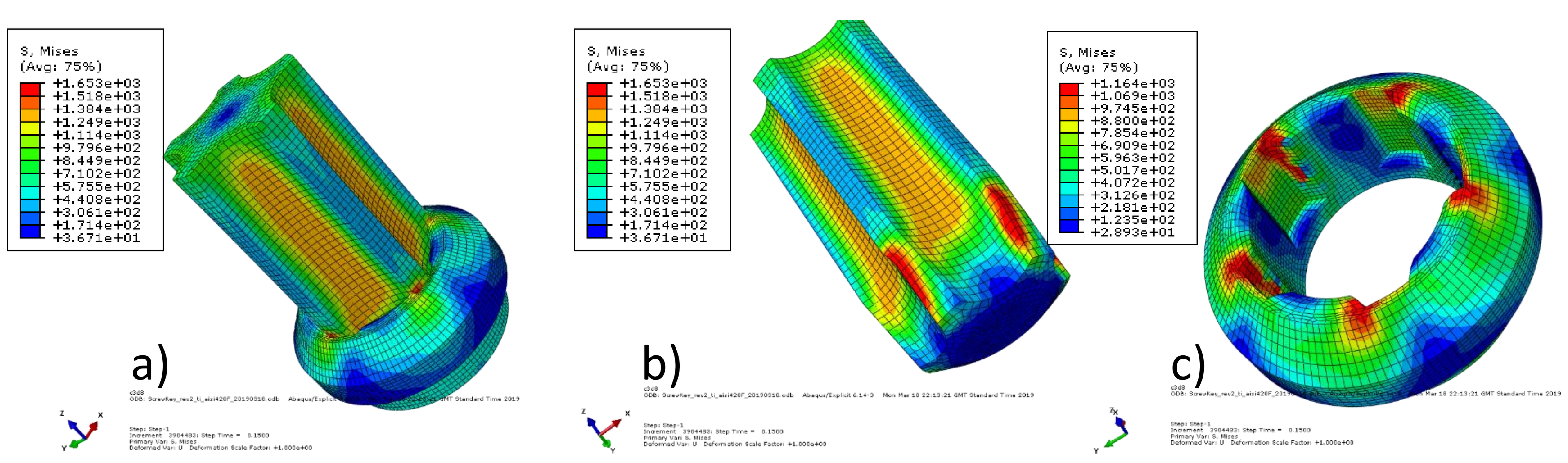

Figure 3. Stress results after the simulation, considering a 0.35 friction coefficient between components and Non-linear geometry in ABAQUS®; (a) Resultant stresses in the assembly model; (b) Screwdriver model detail, as expected the stress concentrations are funded along the ridge's edges from the contact pressure. Although the stress values approaches the ultimate tensile strength (UTS) of the AISI 420F (1660 MPa), only a set of elements is subjected to this value, being the rest of the elements within the material elastic region. Thus, the screwdriver ridges preserves the shape and function; (c) Screw head socket detail, as in the screwdriver analysis, the stress concentrations are encountered in the contact interface, also approaching the UTS of the Ti Gr.5 (1200 MPa). However, interface integrity is maintained due to the lower stress values presented on the outer edge elements.

Experimental setup

Based on the numerical models studied, some prototypes were developed for a final phase of mechanical testing, to compare and validate the simulation results.

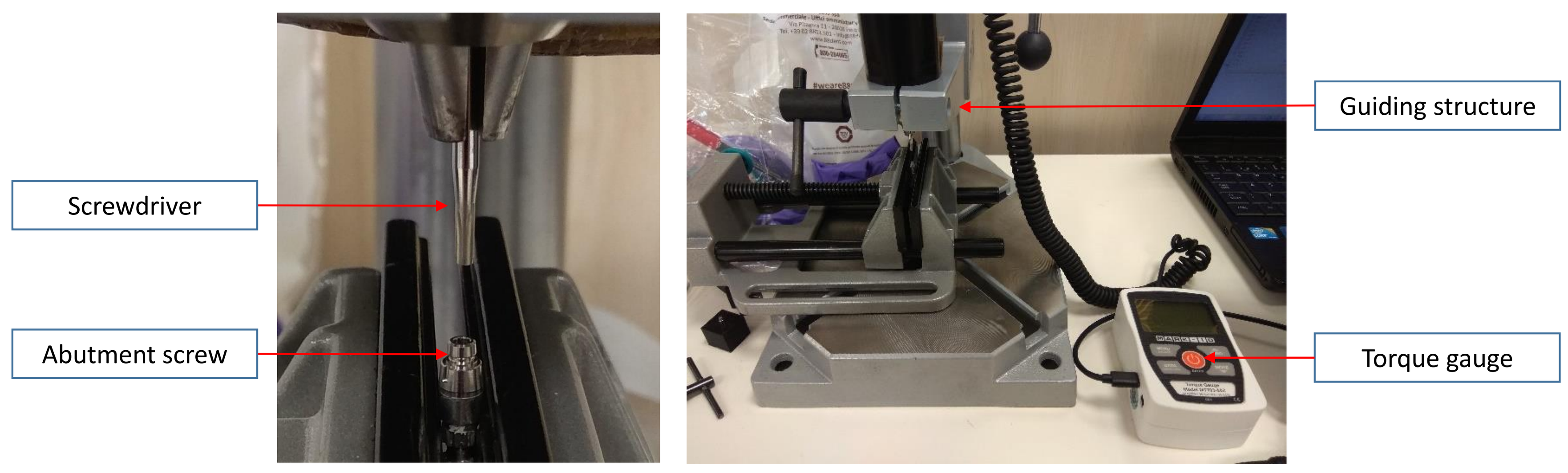

Figure 4. Test setup with digital handheld Torque gauge MTT03-50Z

The experimental procedure was divided in two steps:

1. A single cycle of torque and subsequent detorque with a maximum load of 30 N.cm;
2. Six cycles of torque and detorque with a maximum load of 30 N.cm;

A digital torque meter was used to control the maximum torque value applied to the screw, as well as the maximum detorque required. During the cycle loading the torque recording was continuum.
To assess the possibility of some deformation along the edge of the head socket interface, the samples were submitted to a visual analyses under the microscope.

Results and discussion

The acquired torque/detorque values are presented in Figure 5. The 30 N.cm mark was systematically reached throughout the entire procedure without screwdriver slipping in the screw head socket.

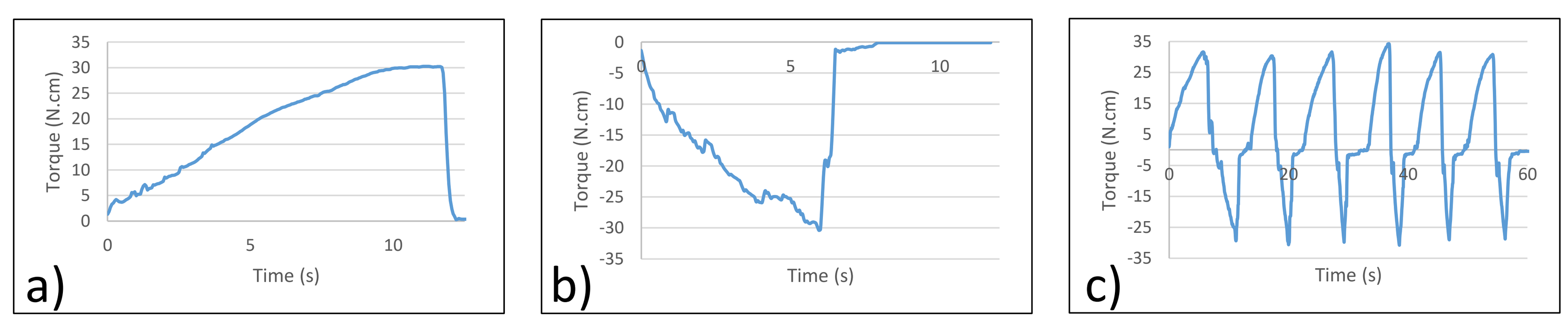

Figure 5. Torque measured during the experimental procedures
(a) Torque applied to the screw; (b) Required torque to loosen the screw; (c) Torque applied during the cycle test

The visual analysis of the top screw head shows no apparent plastic deformation of the interface (Figure 6a), however in the section along the outer edge of the interface it is possible to observe some localized deformation (Figure 6b), which can be accepted for the screw function.
The screwdriver interface shows an absence of deformation after being used for all the tests (Figure 6c).

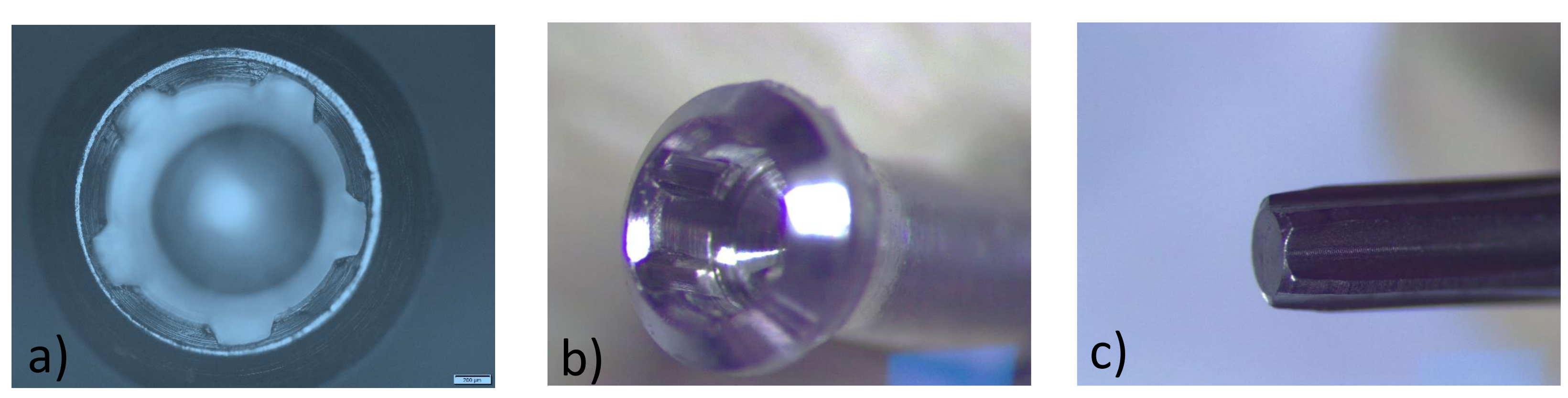

Figure 6. Visual analysis under the microscope 5x Zoom; (a) Top view of the screw interface; (b) Lateral view of the contact interface edge (c) Screwdriver interface view

Conclusions

The experimental results corroborates the ones obtained through the FEA simulation showing a very small localized deformations along the edges of the abutment screw and screwdrivers.
According to the torque results, it is possible to confirm the proper functioning of the new system, and the ability to achieve the 30 N.cm torque without screwdriver slipping in the screw head socket.

Acknowledgments

This work was co-funded by FEDER through the COMPETE2020 project "**SIMPLIFIED: Easy Tooth Abutment**", contract number POCI-01-0247-FEDER-017982.

We would also like to thank INL, Braga, for the torque gauge used in this work.