Investigation on influence of dental implants

A Thesis Submitted for the

Degree of Doctor of Philosophy

By

Pooyan Rahmanivahid

School of Engineering and Design

Mechanical Engineering

Brunel University

September 2014

ACKNOWLEDGEMENTS

This study could not be completed without Prof. Esat supervision and inspiration. I would like to thank Mr Keith Withers, Dr. Mahmoud Chizari, and Miss. Saba Eshraghi for their help and assistance throughout this project.

I would like to thank my other half; Gazelle, this work would not be done without her support.

And especially to God who made all things possible.

ABSTRACT

Osseointegration is defined as the direct physical and practical relation between the living tissue and implant surface. Although, success rate of dental implants is high, implant failure occurs. Overloading implants from occlusal forces are known as one of the main reasons.

In order to have successful implant, a dynamic balance must be provided between mechanical and biological elements (Isidor, Flemming 1996).

Şimşek et al. reported bone quality, oral sanitation, host medical condition and biomechanical parameters as the main reasons for implants failure. Also, implant fixture micromotion and inappropriate stress in the bone implant interface is known as the potential reasons for early bone loss and implant failure (Şimşek, Barış 2006).

Even so, implant position in jawbone, bone density; biomaterial properties of implant surface, treatment technique, loading history and patient clinical status are the influential factors in implant success (Brunski, J.B. 1999).

Although there are many studies on stress distribution of implants in bone-implant interface, majority are limited to current implants in the market. However, current designs have been developed by marketing purposes rather than scientific considerations. Therefore, there is need to introduce and analyse new designs in order to optimize implant structure. Recent investigations have shown reliability of FEA method in simulating human jawbone situation.

This research aims to develop a new dental implant with better life expectancies and introduce an optimized implant based on FEA stress analyses and experimental tests.

Therefore, based on literature recommendations a series of new design factors are defined and analysed. In this study, a primary design is created in AutoCAD and

yields to 3 different implants developed in SolidWorks. Branemark MK IV was selected as the bench model to play role of control group. Then, CT-scan images of human jawbone are imported to MIMICS to create a host bone model. Implant and jawbone models are assembled in 3-Matic and exported to Abaqus for final analyses. A series of loadings are defined to examine implant performance in different conditions.

Branemark and C-3 implants are manufactured from Titanium for experimental analyses. Mechanical tests on sawbone foam blocks and cadavers are targeted to portray realistic performance.

This research demonstrates C-3 model as the optimized dental implant, which presents a new design profile and better performance in low bone densities.

The FEA and experimental results validate the benefit of the new design compare to the conventional ones. Furthermore, results can provide a basis for future designers to develop further optimizations.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES