九州大学学術情報リポジトリ
Kyushu University Institutional Repository

Subject-specific Finite Element Analysis of Orthodontic Anchorage Screws

アルボガ, ムハッマド ハッサン

https://hdl.handle.net/2324/1500641

出版情報：Kyushu University, 2014, 博士（歯学）, 課程博士
バージョン：
権利関係：Public access to the fulltext file is restricted for unavoidable reason (3)

KYUSHU UNIVERSITY

（様式３）

氏　　名　：アルボガ　ムハッマド・ハッサン

論 文 名　：Subject-specific Finite Element Analysis of Orthodontic Anchorage Screws

（矯正用アンカースクリューの個体別有限要素解析）

区　　分　：甲

論 文 内 容 の 要 旨

The present study aimed to evaluate the stability of orthodontic anchorage screws using subject-specific finite element analysis. To achieve this purpose, we first analyzed five different finite element models of orthodontic anchorage screw, to show the importance of subject-specific approach compared to generic approach of finite element modeling. The models were constructed using CT data of one patient and microCT data of a screw. The five models differed in bone properties assignment method, contact surface simulation method, or geometry of bone. The distribution contours of the stress and strain were produced for each model, and compared to each other.

In the second and third parts of the study, subject-specific finite element analysis (FEA) was applied for twenty-eight OASs placed in 16 patients. Every model was constructed using two CT datasets. The pre-insertion CT image was used to generate the 3D models of bone, second premolar, first molar and periodontal ligament of these two teeth. The second post-insertion CT image was used to transfer the position of screw to the final model using 3D alignment technique. A microCT of a screw is used develop the 3D model of the screw, and the 3D model of screw is combined with bone, teeth and periodontal ligaments in one model. The 28 subjects included six OASs failed during the first five months after placement, and 22 successful OASs. Mechanical parameters related to stress and strain were calculated in the surrounding alveolar bone using FEA, and logistic regression analysis was performed to verify the relationship between these parameters and the stability of OASs. Risk factors such as bone elastic properties were calculated and regression analysis of strain as an indicator for stability was conducted on each risk factor. Factors that were found to be significantly related to strain (the criterion we verified) were included in multiple regression analysis to describe the pattern of relationship between these factors.

Subject-specific finite element models presented different distribution of stress and strain, when compared to generic models (first part of the study). Strain-related parameters were significantly related to stability of screws more than stress-related parameters. The widely used criterion, von Mises stress, had very little reliability to predict failure. Maximum values of maximum principal strain in bone area 0.5-1 mm distant from OAS surface were the best parameter to predict failure with high reliability (R^2 = 0.8151). Young's modulus, proximity to tooth roots and the vertical angle of the OAS were indicated as the factors mainly affecting to the maximum principal strain in bone area 0.5-1 mm distant from OAS surface.

In conclusion, subject-specific finite element models are highly recommended and preferred over the generic models to simulate the orthodontic screws, because generic models might give misleading and incorrect distribution of stress and strain around screws. FEA is a useful tool to predict the success and failure of OAS. The maximum principal strain in the surrounding bone around OAS is recommended as a reliable indicator of stability. Young's modulus, proximity to the tooth root and vertical angle of OAS could be the critical parameters and affecting factors for prediction of success and failure of OASs.