人工牙根植體受重覆性疲勞負荷之破壞斷面分析

Fracture Surface Analysis of Dental Implant after Repeated Loading Fatigue Failure

中文摘要

雖然人工植牙的成功率很高，但植體失敗的機會仍時有所聞，而其中最常見的原因，則是植體的疲勞性斷裂以及螺釘鬆脫。儘管各家製造廠商在其植體系統中紛紛設計各種增強組件，但仍無法有效避免其發生的機會。當發生植體失敗時，簡單則只需要更換螺絲組件等，若無法如此解決，則常意味著要二次手術。人工牙根鎖合螺釘發生疲勞斷裂，其原因可能是患者的特殊咀嚼習慣，若在第二次手術或重新更換鎖合螺釘時，手術者未能瞭解此特殊的情形，則重新裝入的鎖合螺釘依然會發生疲勞斷裂。但當人工牙根的鎖合螺釘發生疲勞破壞後，並沒有一個方法或指標，可以提供任何訊息給牙醫師，使其知道此鎖合螺釘已經承受了多少次與多大的重複性負荷以及病人的人工植體是否處於一個正常的受力狀態中。因此本研究計劃利用動態材料試驗機，對植體的疲勞強度作一測試。首先進行緊配扭力檢測及靜態抗壓檢測，以求得適當的實驗參數，接著在負荷控制模式下進行動態疲勞測試，除提供斷裂試件以電子顯微鏡觀察植體破壞面的金相外，更利用有限元素分析描繪出受測人工牙根螺釘的疲勞限、S-N curve。結果顯示，所受的負荷力量愈小或負荷次數愈多時，其連接體螺釘斷面的平滑面比例愈大，且彼此成線性關係。而其中又以斷面平滑區長度比例與負荷力量的相關性最高（R2=0.8506, p<0.01）。有限元素的數值模擬，不但提供了內應力集中位置，更有效的計算出內應力大小。本研究希望藉此發展一套指標作爲評估植體破壞的進程與原因的工具，並將分析所得的資料與臨床上的觀察作一比較，期提供植體設計與臨床應用一有用的資料。

英文摘要

Dental implants have become increasingly important in oral rehabilitation. However, implant treatments still fail frequently. The implant per se has to withstand stresses induced by intraoral forces. Abnormal loading, as well as fatigue under physiological loads, can lead to fractures of certain implant components. These problems are always complicated by cyclic fatigue.Fracture or loosening is most likely to occur in the screw. As for fracture problems, however, it is difficult to predict which component will suffer fatigue and the resulting effects on the entire system. Important aspects of implant design are related to biomechanics of implant systems and the different materials used for implants. However, a reliable method for dentist or manufacturer to evaluate the mechanisms of implant fracture was still unavailable.In this study proposal, a serious dynamic mechanical were carried out to assess the maximum dynamic loading and fatigue life of the implants. To provide the optimal testing parameters, preloading tests and static tests were performed first. After the pretests, dynamic tests were performed under the mode of loading control with a sinusoid force. To provide an indicator for assessing the fracture mechanisms of the fracture screw, the fracture surface of the failed screws were observed and recorded by SEM observation. The fatigue life as well as S-N curve of the sample implant were also obtained and further analyzed by numeric analysis with finite element method.The results showed the ratio of the length and area of the smooth portion on the fracture surface increased with decreased loading magnitude and increased loading cycles. It demonstrated linear relationships and the reliability is the highest between length ratio of the smooth area and loading magnitude (R2=0.8506, $p<0.01$). Besides, finite element analysis also demonstrated not only the stress concentrated area but the magnitude. The results obtained from this project will serve as important references for the future advance studies.