Original

**Osseointegration on Temporomandibular Joint Implants with Different Novel Surface Modifications**

Hung-Ta Lin and Han-Yi Cheng

Abstract: The aim of this study was to investigate stresses resulting from different thicknesses of grit blast (GB) and micro arc oxidized (MAO) treated layers at the interface between temporomandibular joint (TMJ) implants and bones using three-dimensional (3D) finite element models. Several studies have investigated finite element models for TMJs, but few have examined a model for TMJ implants with treated layers. The maximum stresses in the bone occurred at the position of the first screw. Data analysis indicated a greater decrease in this stress in the case of using TMJ implants with MAO treated layers, and the stresses decreased with increasing layer thicknesses. Results confirmed that the treated layers improve biomechanical properties of the TMJ implants and release abnormal stress concentration in them. The results of this study offer the potential clinical benefit of inducing superior biomechanical behavior in TMJ implants.

Key words: Biomechanics, Grit blast, TMJ implant, Micro arc oxidized, Von Mises stress

**Introduction**

Surface treatment technology of implant is widely used in orthopedics because it lends outstanding biocompatibility properties to bone. However, few studies have investigated the interface between implant and bone, because of technological difficulties faced during the experiments. The number of clinical cases of temporomandibular implant surgery is presently on the rise. Temporomandibular disorders (TMDs) are common in adults; one-third of adults are reported to have one or more symptoms, which include joint pain, headaches, clicking, or muscle tenderness. TMDs are defined by a cluster of conditions characterized by pain in the temporomandibular joint (TMJ) during jaw motion. TMJ implant has been used in cases of serious degenerative disease, acute joint trauma, and tumors. The TMJ is a geometrically complex and extremely mobile joint; its motion is described by large displacements, rotations, and deformations. Many therapies are commonly used for treatment; however, surgery is suggested only in severe cases after noninvasive therapies have failed to provide adequate relief.

Finite element method (FEM) is a useful tool that can be applied to quantify the stress distribution in the TMJ and surrounding tissues, and it has been used previously to study the biomechanical behavior of orthopedic devices, including hips, knees, and spinal implants, under various loading conditions. Until recently, only a few studies have been conducted for comparing the bone qualities, and diameters and treated layers of TMJ implants using computer tomography (CT) images to understand the stress fields.

Mechanical behavior of the implant-bone interface is an important factor in successful clinical application of the implant, but investigations of this behavior through experimental and theoretical analysis have thus far been scarce. To examine the biomechanical behavior of TMJ implants with treated layers, the magnitude and location of the maximum stresses under physiological force must be studied. The biocompatibility of specimens was with high wettability and a thick grit blast (GB) and Micro Arc Oxidized (MAO) layers in the implant. Therefore, the aim of this study was to demonstrate the maximum stresses in the TMJ implant with GB and MAO treated layers and mandible using 3D finite element models (FEM) reconstructed from in vivo CT data.

**Materials and Methods**

**Clinical computer tomography (CT) images**

3D finite element model of a human mandible was developed for decades, including the cancellous and cortical bones, was obtained from CT images (Brillance CT, Philips Medical Systems). These models were reconstructed in an adult patient and for geometry acquisition, a set of images were obtained from CT slices of the TMJ and segmented for data extraction to describe the surfaces of the mandible (Fig. 1). An edge detection algorithm was run with the AVIZO 7.2 (Internet Securities, Inc) program to distinguish the cortical bone from the cancellous bone and to detect the various boundary components of the mandible.

**Material properties and muscle force**

The surface mandible is characterized by the properties of the cortical bone, whereas the internal nodes are characterized by the properties of the cancellous bone. The biomechanical properties of the cortical bone, cancellous bone, Ti, GB and MAO have been described in previous literature. The thicknesses of the GB and MAO treated layers were set at 0-500 nm. The muscles of mastication have significant influence on the TMJ and transmit functional chewing forces to TMJ implants. This study considered the three muscles that are involved in mouth closure since their movements result in maximum physiological loading to the TMJ. The direction and magnitude of the muscle forces were taken from previous studies; it was necessary to determine the cross-sectional area (CSA) and calculate the maximum muscle force via the following mathematical function. The muscle forces (left side/right side) were as follows: masseter (176.86 N/161.32 N), temporalis (104.71 N/125.80 N), and medial pterygoid (87.69 N).
F max = K x CSA                      (1)

Eq. (1) estimates the maximum possible muscle forces based on the CSA, where K = 37 N cm⁻²¹⁶¹[1].

Finite element analysis

The 3D image reconstruction models were built using the ANSYS Workbench 12.1 (ANSYS, Inc) finite element program. The TMJ implant and mandible structures were regarded as continuous integers. Fig. 2 shows TMJ implants of the standard type (Inc., Golden, CO); length: 10.03 mm, thickness: 2.53 mm used in combination with a bilateral TMJ model as shown in Fig. 2. For obtaining accurate results using the FEM, two important processes are converging and reinforcing of the mesh, which allow the model to approximate the actual object more accurately. We can assume the mesh to be reliable after the convergence process. The average numbers of nodes and elements were approximately 132,432-160,532 and 86,460-113,689, respectively (Fig. 3).

Results

The highest stresses occurred at the first screw for all the TMJ models. Figs. 4 and 5 show the von Mises stress distribution for the TMJ implant in the GB-Ti-500 group. The highest stress in the TMJ implant varied from 101.64 to 110.15 MPa with the MAO treated layer, and 105.58 to 111.54 MPa with the GB treated layer. The highest stress was below the failure stress, which could mitigate the risk of over-loading. On the other hand, the stress distributions were more uniform among the group of thicker treated layers. Remarkably, the maximum von Mises stress was reduced approximately 10.48% in the MAO-Ti-500 group during the jaw movement, relative to the untreated group. The stress distributions showed significant stress focusing in...
the untreated group. Surface treatments were demonstrated to have the potential to decrease the abnormal stress concentration for the TMJ implant.

The highest stress for the bone was 10.67% lower in the MAO-Ti-500 group than in the untreated group as shown in Figs. 6 and 7. Treated layers were demonstrated to decrease the stresses in the bone tissue. The maximum stresses in the bone were observed at the first screw. The maximum von Mises stress for the MAO treated group showed a tendency to be smaller than that for the GB treated.

Figs. 8 and 9 show the stress distributions for screws in the GB-Ti-500 group. In the control group, the stress of the screw that was >140 MPa, while the maximum stress in the MAO-Ti-500 group was 128.71 MPa. At all different type models, the maximum stresses varied from 128.71 to 143.54 MPa. The stresses of screws with surface treatment were less than the untreated implant. The maximum observed von Mises stress occurred at the interface between the implant and the bone at the first screw.

The stress patterns in both models were similar to each other. Analysis of the present study indicated that stresses GB and MAO surface treatment were transferred more uniformly.

As described above, data analysis indicated that reduced stress was sustained by TMJ implants, bones and screws for all treated groups. The stresses transferred more uniformly in MAO treated group than in GB treated group, and the maximum stress of MAO-Ti-500 group was the smallest in all groups. Part of the contact region transfers the load from the mandibular condyle to the TMJ implant during its function; in this study, the maximum stress was observed at the contact points near the first screw. Applied stress is considered the most important factor in the failure of TMJ implants.

Discussion

Finite element analysis is used in mechanical engineering analysis and design to evaluate potential decay. They have been successfully applied in for biological applications to model implants for various human joints. In the dental field, models have been used to determine stresses in different biological structures, such as TMJ implants, facial
skeletons, and periodontal ligaments, and for dentition. Most of these FEM studies have analyzed the biomechanical behavior of individual structures or materials. However, biomechanical models of the human masticatory finite element model are not perfect; on the whole, they are based on a number of assumptions and simplifications. In order to obtain the correct geometry of the model, we selected a precise slice thickness (0.625 mm) for reconstruction using CT images. Magnetic resonance images (MRI) can also be used to obtain information about TMJ structures without exposing the patient to radiation, but it is more applicable for evaluating the soft structures of the TMJ, especially the articular discs. The contours of the hard tissue are not rigorously represented from MRI data; however, this study focused on the hard tissues of the masticatory system, such as the cortical and cancellous bones.

Several researchers have attempted to directly assess or measure the mechanical loads on the TMJ during jaw movement by the strain gauge technique; their results have indicated that substantial forces are induced, so the TMJ acts as a load-bearing organ in nature. However, these experimental approaches have shortcomings, such as the need for surgical invasion of the TMJ structures. However, analytical techniques such as the finite element method have long been used to investigate stresses and forces in the TMJ using a mathematical model. These techniques have been successfully applied in the field of biomechanics because of several advantages that they offer. The model for the present analysis was precisely developed according to the geometry obtained using the CT images, and the appearance of the model can well approximate that of the actual object. It measures stresses in the internal model as well as easily simulating various situations for different magnitude forces or bone densities.

Most of the von Mises stresses in the TMJ implants were located in the region around the first screw; these overloading stresses can easily initiate implant failure. Various studies have demonstrated that these regions of the mandibular condyle are important functional regions that sustain masticatory loading. On the other hand, 7–10 screws were not useful in TMJ implant surgery on the healthy bone; nevertheless, superior anchorage force was important for patients with low bone density.

It is difficult to accurately compare this result with the results obtained using models in previous studies, because of various differences in the material properties, muscle forces, mesh, and constrained conditions employed in these models. Another important factor that governs the analysis of results is the mechanical and physical properties of the models relative to the actual object. In this study, we considered four main materials: bone tissue, Ti, GB, and MAO.

A previous study demonstrated that bone stresses of TMJ implants were about 8.0-15.0 MPa for every degree of bone quality, which supports this results. The yielding stress of bone was approximately 130 MPa25). In this study, the maximum stress along path distance was 11.54 MPa, which did not exceed the yielding stresses of bone. However, the present study indicated that osteoporosis induced reduced bone density. The model for the present analysis was precisely developed according to the geometry obtained using the CT images, and the appearance of the model can well approximate that of the actual object. It measures stresses in the internal model as well as easily simulating various situations for different magnitude forces or bone densities.

The authors have declared that no competing interest exists

References

9. Beek M, Koolstra JH, van Ruijven LJ and van Eijden TMGJ.