Medical Engineering and Physics 37 (2015) 862–869

Contents lists available at ScienceDirect

Medical Engineering and Physics

journal homepage: www.elsevier.com/locate/medengphy

Failure location prediction by finite element analysis for an additive manufactured mandible implant

Jinxing Huo[a], Per Dérand[b], Lars-Erik Rännar[c], Jan-Michaél Hirsch[d], E. Kristofer Gamstedt[a,*]

[a] Division of Applied Mechanics, Department of Engineering Sciences, Uppsala University, Box 534, SE-751 21 Uppsala, Sweden
[b] Department of Oral and Maxillofacial Surgery, Skåne University Hospital, SE-221 85 Lund, Sweden
[c] Sports Tech Research Centre, Department of Quality Technology and Management, Mechanical Engineering and Mathematics, Mid Sweden University, SE-831 25 Östersund, Sweden
[d] Department of Surgical Sciences, Oral & Maxillofacial Surgery, Uppsala University, 751 85 Uppsala, Sweden

ARTICLE INFO

Article history:
Received 27 November 2014
Revised 13 May 2015
Accepted 9 June 2015

Keywords:
Mandible implant
Failure
Computed tomography
Homogenisation
Finite element simulations

ABSTRACT

In order to reconstruct a patient with a bone defect in the mandible, a porous scaffold attached to a plate, both in a titanium alloy, was designed and manufactured using additive manufacturing. Regrettably, the implant fractured in vivo several months after surgery. The aim of this study was to investigate the failure of the implant and show a way of predicting the mechanical properties of the implant before surgery. All computed tomography data of the patient were preprocessed to remove metallic artefacts with metal deletion technique before mandible geometry reconstruction. The three-dimensional geometry of the patient's mandible was also reconstructed, and the implant was fixed to the bone model with screws in Mimics medical imaging software. A finite element model was established from the assembly of the mandible and the implant to study stresses developed during mastication. The stress distribution in the load-bearing plate was computed, and the location of main stress concentration in the plate was determined. Comparison between the fracture region and the location of the stress concentration shows that finite element analysis could serve as a tool for optimizing the design of mandible implants.

1. Introduction

Congenital malformations of mandibles and defects after traumatic injuries or tumour resection frequently result in functional and aesthetic problems. Several surgical remedies are available to treat this via mandible reconstruction. The method of choice today for reconstructing extensive mandibular defects uses a free vascularised fibula flap [1,2]. Although this method commonly produces good functional and cosmetic end results, it includes considerable morbidity for the patient and is a time consuming and costly procedure which can only be performed by surgeons with specific competence. The use of titanium scaffolds for patients who do not require soft tissue transfer has the potential to substantially eliminate the drawbacks of a micro-vascular reconstruction [1,3]. However, to the best of our knowledge, the design of such implants nowadays is much dependent only on the essential experience of surgeons [4]. One way to prevent such failures is to employ preoperative finite element (FE) analysis to predict structural strength and assist the design of implants [5–7].

As reported, a large number of investigations have been devoted to the FE analysis of implants. Many studies have assumed isotropic bone material properties in order to simplify simulations [8–10]. Some other studies have adopted an improved orthotropic bone material model [7,11]. Yet, these studies were not based on clinical cases. Also, validation of the FE models was through comparison with results of in-vitro experiments [12]. The present study investigates the failure of a titanium-alloy mandibular implant from a clinical case, with the objective of exploring the capability of FE analysis in implant biomechanics and providing ideas of how to design patient-specific implants in the future.

The mandibular implant used for tumour treatment of a patient failed several months after surgery. The preoperative computed tomography (CT) data, contaminated by metallic artefacts from previously installed osseointegrated dental titanium implants, were first preprocessed using the metallic deletion technique to eliminate the metallic artefacts. Then, the three-dimensional (3D) geometry of the patient's mandible was reconstructed on the basis of the preprocessed CT data, and the implant was fixed by screws in Mimics medical image processing software (Materialise, Leuven, Belgium) to the bone in its post-surgery in-vivo position. The whole assembly of the mandible and the implant was then imported into Abaqus (Dassault Systemes) and meshed with tetrahedral elements. Under

* Corresponding author. Tel.: +46 18 4713026; fax: +46 18 4712000.
E-mail address: kristofer.gamstedt@angstrom.uu.se (E.K. Gamstedt).

http://dx.doi.org/10.1016/j.medengphy.2015.06.001
1350-4533/

appropriately formulated boundary conditions, the mastication of the mandible was modelled and the stress distribution in the load-bearing plate of the implant was determined. The fracture region and the location of the stress concentration were compared.

2. Methods and materials

2.1. Mandible geometry reconstruction

In the preoperative CT data of the mandible of a 68-year-old male patient, suffering from mandibular tumour and getting the tumour bone replaced with a titanium alloy implant [13], 40 out of 187 slices were severely interrupted by metallic artefacts giving rise to, e.g., star and streak patterns. Therefore a cleaning process was implemented prior to mandible geometry reconstruction in order to obtain more accurate geometries. The metal deletion technique software was employed. It uses a linear interpolation algorithm, forward projection and an edge-preserving blur filter to eliminate noise and streaking artefacts [14].

After preprocessing, the CT data were imported into Mimics for geometry reconstruction. A global threshold-based segmentation method was chosen to extract the bone regions from the soft tissues and the background. Besides, the titanium dental implants previously transplanted were isolated from bony regions using an analogous method.

The porous trabecular bone in the centre of the mandible was left as cavity, since its relatively low stiffness has negligible effect on the stress distribution on the implant. Similarly, in [15] the cancellous part of clavicle finite element model is not taken into account since the interest is devoted to the implant stresses. MacLeod et al. [16] ignored the cancellous bone in their model, and concluded that its contribution to the strains around screw holes is small [16]. According to Misch et al. [17], the cortical bone has the dominating influence in distribution occlusal loads. Moreover, the influence of the trabecular bone on the stress distribution in the implant was studied using a simple mandible model. It was found that the maximum stress difference between the model with and without trabecular bone is around 1% under conditions of small deformation. The trabecular bone was marked manually slice by slice from the segmented images and excluded from the mandible. Subsequently, 3D mandible objects were generated by a region growing method and smoothed for the purpose of meshing.

2.2. Homogenisation of the scaffold

The titanium alloy implant consisted of two distinct but integrated parts: the load-bearing plate and the scaffold as shown in Fig. 1(a). As the struts in the scaffold were thin beams, FE discretization of the scaffold causes computational difficulties. Therefore, the scaffold was converted into a homogenised scaffold through computational homogenisation with a representative volume element (RVE) as shown in Fig. 1(b). Since our interest was to determine the stress distribution in the load-bearing plate, simplifying the geometry of the scaffold was supposed to have little influence on the stress distribution over the plate whose dimensions are larger than those of the mesh RVE.

The RVE comprises four titanium alloy struts with length 10.4 mm and diameter 0.70 mm which follow the space diagonals of a cube and intersect in the centre, shown in Fig. 1(b). The struts were made of a titanium alloy with Young's modulus 110 GPa and Poisson's ratio 0.35 [7]. The RVE was meshed with 17456 quadratic tetrahedron elements and 28849 nodes after mesh sensitivity analysis. For implementation of periodic boundary conditions, the mesh patterns on opposite boundary surfaces must match. To fulfil this requirement, one eighth of the RVE was established and meshed with quadratic tetrahedrons in an in-house CAD (computer aided design) software. The rest of the RVE was then created in MATLAB (MathsWorks) through symmetries. The RVE was homogenised in MATLAB. The effective three-dimensional orthotropic elastic properties (Young's moduli, Poisson's ratios and shear moduli) were calculated in the directions of the three material coordinate axes (Fig. 1(b)). Detailed explanations of periodic boundary conditions for homogenization can be found in the work of van Dijk [18].

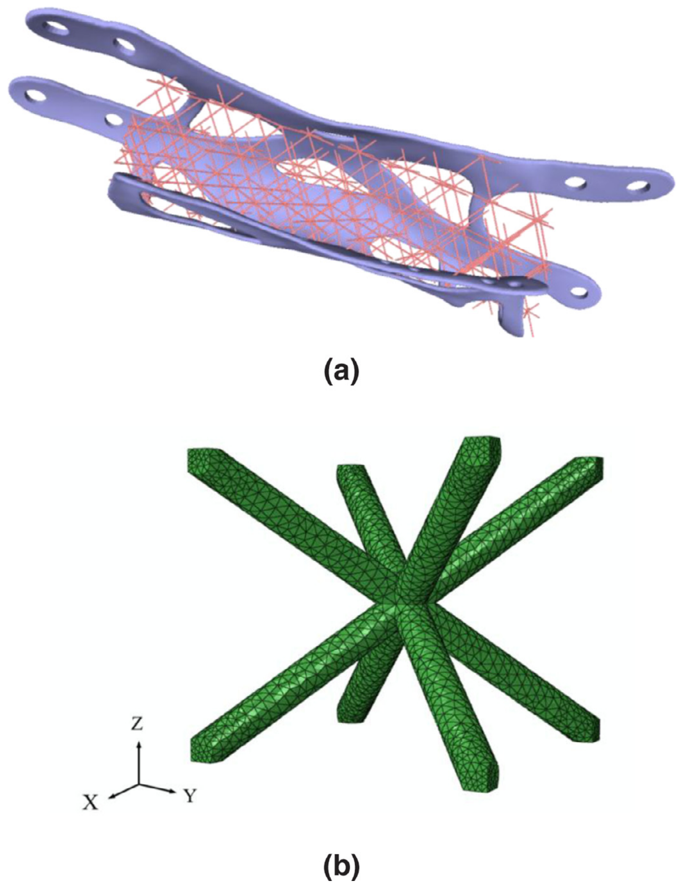

Fig. 1. The titanium alloy mandible implant and the RVE of the scaffold. (a) The geometry of the titanium alloy mandible implant with length around 50 mm and the supportive interior scaffold (in red) of the plate with length about 75 mm, and (b) FE model of the RVE of the scaffold (6.0 mm × 6.0 mm × 6.0 mm).

The stereolithography files of screws, the load-bearing plate and the homogenised scaffold of the scaffold were imported into Mimics. The threads in both screws and screw holes were neglected. The screws, the plate and the homogenised scaffold were translated and rotated relative to the mandible to imitate their real positions in the surgery as represented in Fig. 2(a). The tumour part of the mandible was resected in concordance with the surgery, cf. Fig. 2(b), and screw holes were created.

3. Finite element simulations

3.1. Mesh and contact considerations

The assembly of mandible and implant was imported into Abaqus to simulate the mandible mastication under appropriate boundary conditions. The FE model of the assembly (shown in Fig. 3) was meshed with 512,146 linear tetrahedral elements and 112,725 nodes.

The contact interactions between the load-bearing plate, the homogenised scaffold and the mandible during simulations were defined. For these contacts, the interaction in the normal direction was set so that there is no mutual penetration between contacting parts. The interaction in the tangential direction was set as frictionless for