Journal of Biomechanics 42 (2009) 1224–1229

Contents lists available at ScienceDirect

Journal of Biomechanics

journal homepage: www.elsevier.com/locate/jbiomech
www.JBiomech.com

Validating a voxel-based finite element model of a human mandible using digital speckle pattern interferometry

F. Gröning [a,*], J. Liu [b], M.J. Fagan [c], P. O'Higgins [d]

[a] Department of Archaeology and Hull York Medical School, University of York, York YO10 5DD, UK
[b] Department of Computer Science, University of Hull, Hull HU6 7RX, UK
[c] Department of Engineering, University of Hull, Hull HU6 7RX, UK
[d] Hull York Medical School, University of York, York YO10 5DD, UK

ARTICLE INFO

Article history:
Accepted 11 March 2009

Keywords:
Finite element analysis
Digital speckle pattern interferometry (DSPI)
Computed tomography
Mandible
Bone strain

ABSTRACT

Finite element analysis is a powerful tool for predicting the mechanical behaviour of complex biological structures like bones, but to be confident in the results of an analysis, the model should be validated against experimental data. In such validation experiments, the strains in the loaded bones are usually measured with strain gauges glued to the bone surface, but the use of strain gauges on bone can be difficult and provides only very limited data regarding surface strain distributions. This study applies the full-field strain measurement technique of digital speckle pattern interferometry to measure strains in a loaded human mandible and compares the results with the predictions of voxel-based finite element models of the same specimen. It is found that this novel strain measurement technique yields consistent, reliable measurements. Further, strains predicted by the finite element analysis correspond well with the experimental data. These results not only confirm the usefulness of this technique for future validation studies in the field of bone mechanics, but also show that the modelling approach used in this study is able to predict the experimental results very accurately.

1. Introduction

Finite element analysis (FEA) is a very powerful modelling tool, now widely used and accepted in many areas of science and engineering. It is particularly useful for predicting the mechanical behaviour of complex biological structures such as bones, but to be confident in the results of any FE model, some form of experimental validation is ideally required. In validation experiments of bone models, the strain in the bone is traditionally measured with strain gauges that are glued to the bone surface. In the literature, numerous examples of studies can be found in which FE models of different bones have been successfully validated against strain gauge data (e.g. Daegling and Hylander, 2000; Ichim et al., 2007; Knoell, 1977; Kupczik et al., 2007; Strait et al., 2005; Vollmer et al., 2000).

However, the use of strain gauges can be difficult. For example, the bone surface needs to be flat, and if it is not then obtaining good adherence to the surface can be problematic. Additionally, strain gauges only provide single point measurements on the surface. Thus, the use of strain gauges is time-consuming and requires careful application to obtain what, in the end, is a picture of strain distributions with very limited resolution.

The optical full-field and non-contact strain measurement technique of electronic or digital speckle pattern interferometry (ESPI or DSPI) overcomes these problems (Jones and Wykes, 1989; Yang and Ettemeyer, 2003). To date, few studies have applied DSPI to bone, either for measuring strains (Kessler et al., 2006; Su et al., 2005; Tyrer et al., 1995; Yang and Yokota, 2007; Yang et al., 2007) or elastic properties of loaded bones (Barak et al., 2009; Shahar et al., 2007; Zaslansky et al., 2005; Zhang et al., 2001). The results of these studies confirm the high reliability and practicality of this technique compared to strain gauges. However, the use of DSPI as a tool for validating FE models of bones has not yet been explored.

In this study, DSPI is used to measure surface strains in a dry human mandible under simple loading in the laboratory. The results are compared to FE models of the same specimen. Thus, we aim to assess the potential and limitations of this novel technique for the validation of FE models in the field of bone mechanics. In addition, the DSPI measurements are used to test the accuracy of our FE modelling approach. We discuss the implications for the assignment of material properties, selection of element type and model resolution.

2. Material and methods

2.1. Experimental loading conditions

Loads were applied to a dry adult human mandible with a Lloyd's EZ50 tensile testing machine (Ametek-Lloyd Instruments Inc., UK). The mandible was placed

* Corresponding author. Tel.: +44 1904 328872.
E-mail address: fg513@york.ac.uk (F. Gröning).

0021-9290/$ - see front matter
doi:10.1016/j.jbiomech.2009.03.025

upside down in the machine so that it rested on the two condyles and the anterior teeth (Fig. 1). Compressive loads were then applied to the mandibular angles on both sides of the mandible. During each of the eight load series, the loads were increased in 50 N steps from zero to 250 N.

2.2. Strain measurements

Deformations were measured using a Q-100 DSPI measuring system (DANTEC Dynamics GmbH, Ulm, Germany). Two measuring areas on the right side of the mandible were selected, each ca. $25 \times 33\,mm^2$ in size, which is the maximum field of view of the Q-100 DSPI system. Figs. 1 and 2 show the location of the measuring fields on the mandible. Prior to loading of the specimen, the bone surface in the respective areas was covered with a thin layer of white spray (DIFFU-THERM developer BAB-BCB, Technische Chemie KG, Herten, Germany) in order to create a non-reflecting surface and the three-legged adaptor rings for the Q-100 sensor were glued onto the bone surface using X60 two components adhesive (HBM Inc., Darmstadt, Germany). After the mandible was placed in the tensile testing machine, the sensor was screwed to one of the adaptor rings.

First, the 3D surface topography was measured in each measurement area prior to loading. This topographic measurement allows the accurate calculation of the strains even on objects with complex surface contours (Yang and Yokota, 2007). Next, the loads were applied and the resulting speckle patterns were used to estimate x-, y- and z-displacements for each load step using the software IstraQ100 2.7 (DANTEC Dynamics GmbH, Ulm, Germany). From these, the surface strains were computed and exported as 2D and 3D colour-coded maps, and text data files. Since the test specimen was loaded under compression, the minimum principal strain (ε_3) values were taken as most meaningful and therefore used for this study.

2.3. Model creation

In order to create FE models of the test mandible, the specimen was CT-scanned prior to mechanical testing. High-resolution CT data were obtained using an X-Tek HMX 160 μCT system (X-Tek Systems Ltd., Tring, UK). Since the mandible was slightly above the size limit for this μCT scanner, the two halves of the specimen were scanned separately. The primary reconstructions were performed using NGI CT Control Software (X-Tek Systems Ltd., Tring, UK) and the resulting data volumes were exported as 16-bit TIFF image stacks with a voxel size of 0.122 mm for the right half and 0.135 mm for the left half.

In addition, medical CT scans were taken of the specimen with a GE Medical Systems BrightSpeed scanner (General Electric Co., USA). The image stacks were reconstructed with a pixel size of 0.488 mm and a slice interval of 0.625 mm and exported as DICOM image stacks.

Image segmentation was performed with Amira 4.1.1 (Mercury Computer Systems Inc., USA). Bone and teeth were separated from the surrounding air by a user-defined density threshold. The voxels of the segmented medical CT scan were converted into an isotropic data set with a resolution of 0.488 mm in all three axes. The μCT data already consisted of isotropic voxels, but since the two sides of the mandible were scanned individually with the μCT scanner, the two segmented halves were reconnected again in Amira by landmark-guided superimposition of the overlapping areas.

The resulting 3D volume data were exported as BMP image stacks and converted into 8-noded linear brick finite element meshes by direct voxel conversion, i.e. with each voxel of the data volumes being converted into a finite element, resulting in element numbers and sizes of ca. 450,000 and 0.488 mm for the low-resolution model and 19.6 million and 0.135 mm for the high-resolution μCT model.

2.4. FEA

The FEA was performed using the non-commercial FEA software VOX-FE (Fagan et al., 2007). Isotropic material properties of 17 GPa for Young's modulus and 0.3 for Poisson's ratio were assigned to the model, for both the bone and the teeth, which are values that lie within the range of published values for human mandibles (Arendts and Sigolotto, 1989, 1990; Ashman and van Buskirk, 1987; Dechow et al., 1993; Schwartz-Dabney and Dechow, 2003). In order to simulate the experimental loading conditions, the FE model was constrained in the vertical axis

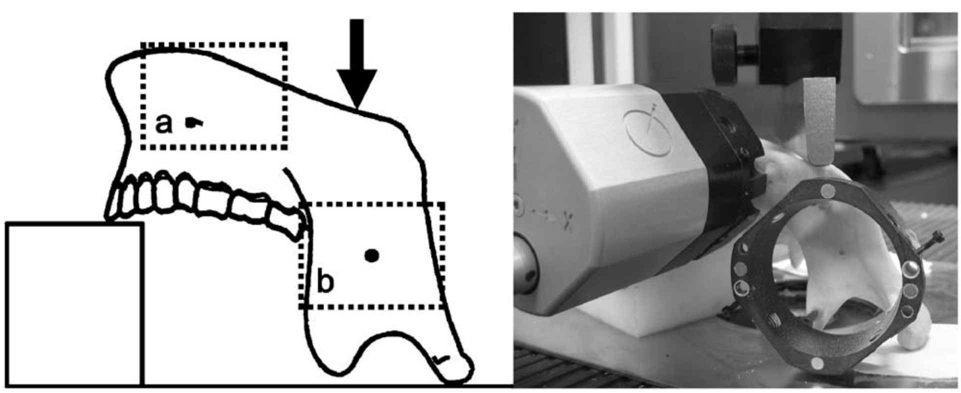

Fig. 1. Drawing and photo of the experimental setup. The arrow in the drawing indicates the position and orientation of the applied force, which acts symmetrically on both sides of the mandible. The dashed rectangles highlight the two measurement areas (a) and (b). The photo shows the mandible in the mechanical testing machine, with the two adaptor rings and the Q-100 DSPI sensor attached to the upper ring on the mandibular corpus.

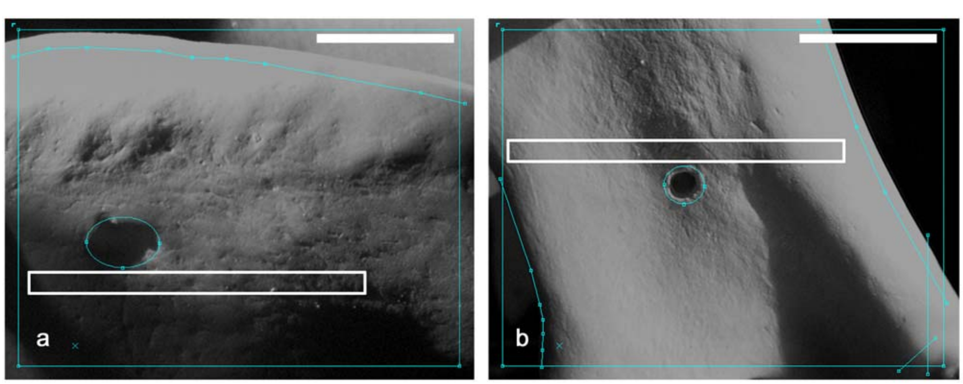

Fig. 2. Photos of the two illuminated areas of the bone surface: (a) area on the mandibular corpus around the mental foramen and (b) central area of the mandibular ramus. The thin blue lines indicate the boundaries of the measurement areas, whereas the white rectangles show the linear areas from which strain profiles were extracted. Scale bars = 1 cm (for interpretation of the references to color in this figure legend, the reader is referred to the web version of this article).