23rd International Conference
ENGINEERING MECHANICS 2017
Svratka, Czech Republic, 15 – 18 May 2017

ANALYSIS OF BOUNDARY CONDITIONS IN FINITE ELEMENT SUB-MODELS OF INTERACTION OF HUMAN MANDIBLE WITH DENTAL IMPLANT

P. Marcián*, Z. Florian, L. Horáčková***, L. Borák******

Abstract: *This paper deals with the computational modelling of dental implant interacting with bone tissue. For the purposes of the study, one human mandible and a segment of a different human mandible were examined. In total, five variants of geometry models with dental implant were investigated: One full mandible and four segments. For the latter, two different boundary conditions were used (first based on the sub-modeling approach and second assuming fixed support). If the coarse model for the particular bone segment model is not available, it is possible to use a generic boundary condition for a typical mandible subjected to a typical loading.*

Keywords: Biomechanics, FEM, Dental Implant, Sub-modeling.

1. Introduction

Constantly developing computer technology entails increasing demands for quality of computational models in many fields of engineering practice as well as of research. Biomechanics is not an exception. On the contrary, despite the fact that biomechanical systems have some specific traits including complex (and often uncertain) material properties of living tissues or boundary conditions that are difficult to grasp, it is a common request from scientific community members (such as journal reviewers etc.) to provide a model that would include high-level features that are often needless or unnecessary for the solution of the problem. Such requests often expose one's misunderstanding of what the modeling is about than his expertise or experience.

Typical challenge in biomechanics is a computational modeling of complex systems consisting of small artificial elements such as dental implants or micro screws (Ridwan-Pramana et al., 2016). In such cases, it appeared to be useful and appropriate to use the sub-modeling method which allowed computational solution of a given problem at a sufficient level. One of the most significant features of the bone in which the implant or screw is embedded is the trabecular architecture of its cancellous part. Nowadays, it is possible to create a detailed model of such architecture using micro-FE methods. However, creating a computational model including the trabecular architecture in the entire bone is impractical. The most common procedure is to create only a segment of bone with appropriate boundary conditions. The boundary conditions might be derived from a coarse model which consists of a larger part of (or whole) bone with coarser FE mesh and with other simplifications such as screws/implants modeled without a thread. After the coarse model solution is carried out, the boundary condition might be applied to the sub-model with much finer FE mesh and more detailed representation of the implant/screw. However, the coarse model might not be always available and the question is whether a typical boundary condition for a sub-model might be found.

* Ing. Petr Marcián, PhD.: Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, 616 69 Brno, Czech Republic; CZ, marcian@fme.vutbr.cz

** Assoc. Prof. Ing. Zdeněk Florian, PhD.: Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, 616 69 Brno, Czech Republic; CZ, florian@fme.vutbr.cz

*** Assoc. Prof. RNDr. Ladislava Horáčková, PhD.: Department of Anatomy, Faculty of Medicine, Masaryk University, Kamenice 126/3, 625 00 Brno, Czech Republic; CZ, lhorac@med.muni.cz

**** Ing. Libor Borák, PhD.: Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, 616 69 Brno, Czech Republic; CZ, liborborak@seznam.cz

The aim of this study is to analyze different levels of boundary conditions for a sub-model of a human mandible with a dental implant.

2. Methods

2.1. Geometry model

For the purpose of this study, one human mandible and a segment of a different human mandible were scanned on CT (voxel size 0.5 x 0.5 x 1.0 mm). CT images were further processed in STL Model Creator (Matlab, 2012) to obtain 3D models of the mandible (Fig. 1), (Marcián et al., 2011). In this software, surfaces of cortical and cancellous bones were created using the automatic segmentation (Huotilainen et al., 2014). Finally, volume models of both bone tissues were created in ANSYS 17.2. Branemark dental implant (Brånemark® System Mk III Groovy, NP Ø 3.3 mm, 11.5 mm) with abutment was chosen as typical implant for this study (Fig. 1).

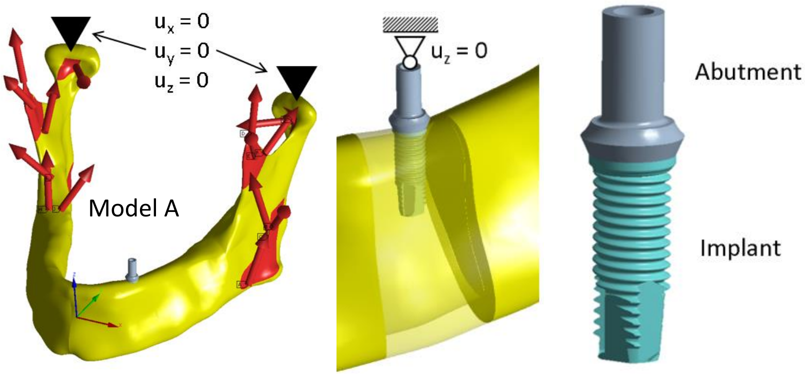


Fig. 1: Human mandible and detail of dental implant. Model A – loading and boundary conditions.

In total, five variants of geometry model of mandible with dental implant were investigated: Model A – The entire mandible (Fig. 1); Model B – A larger segment of mandible from Model A (length of 60 mm, Fig. 2a); Model C – A shorter segment of mandible from Model A (length of 15 mm, Fig. 2b); Model D – The segment from different mandible (length of 15 mm, Fig. 2c); Model E – Model D with modified cortical bone thickness (Fig. 2d).

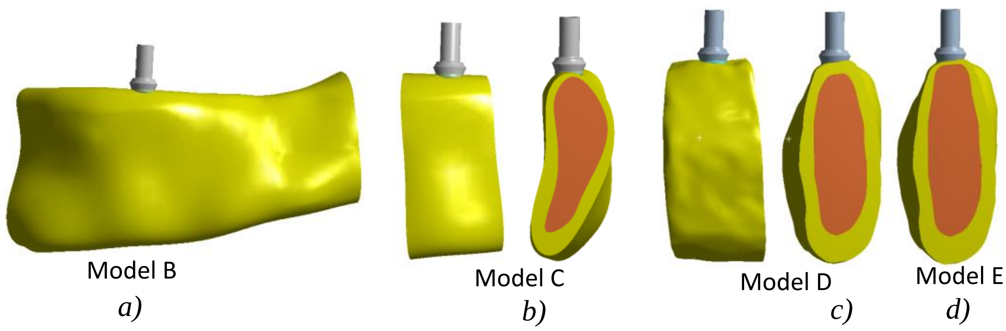


Fig. 2: Mandibular segments. a) Model B; b) Model C; c) Model D; d) Model E.

2.2. Loads and boundary conditions

Model A was loaded by forces applied in locations of muscle inserts and in directions of that muscles. The force magnitudes were taken from literature (Narra et al., 2014). The implant was constrained in axial direction of the implant as shown in Fig. 1. These loads and constraint represent an occlusion of the first premolar.

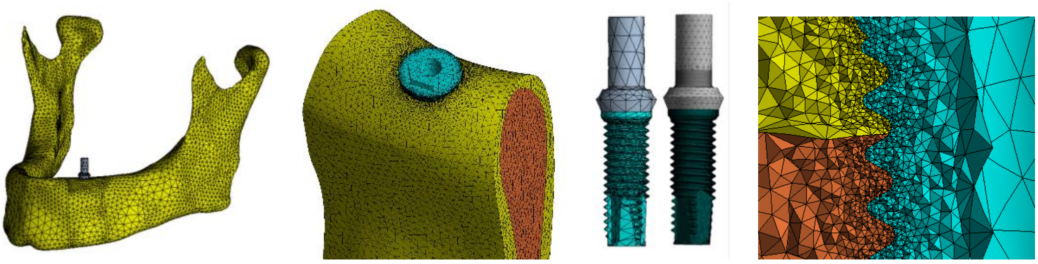
Models B through E were treated in two ways:

1. As sub-models using displacements from Model A (regardless that Models D and E have slightly different geometry). The implant was constrained in axial direction of the implant. Sub-models with this treatment were denoted with number 1 (B1, C1, D1, E1).
2. As fixed segments, i.e. the segments were fixed by both faces. This approach is very popular and often used in many studies dealing with bone-implant interactions (Marcián et al., 2014; Mellal et

al., 2004). The implant was loaded by an axial force obtained from Model A (i.e. the reaction force in the implant constraint). Models with this treatment were denoted with number 2 (B2, C2, D2, E2).

2.3. FE Mesh

The models were meshed by quadratic tetrahedral elements SOLID187 (Fig. 3). In case of Model A, the general element size was 3 mm and element size in the implant region was 1 mm. In total, Model A consisted of 184 409 nodes. In case of Models B through E, the general element size was 1 mm and element size in the implant region was 0.05 mm. The segment models consisted of 4.5 – 5 million nodes. The dental implant was assumed to be fully osseointegrated; therefore, the bonded contact algorithm was employed and contact elements CONTA160 and TARGE174 were used to model the interaction between the implant and the bone (Marcian et al., 2014).



a) *b)* *c)* *d)*

Fig. 3: Finite element (FE) mesh. a) Model A; b) Typical mesh of segment (Model B); c) Dental implant; d) bone-implant interface.

2.4. Material Properties

Cortical as well as cancellous bones were assumed to be linear, isotropic, elastic and homogeneous material. This simplification was considered to be sufficient for the purpose of this study. Specifically, both bone tissues were modeled by following Young's modulus and Poison's ratio values: Ec = 13 700 MPa, μc = 0.3; Es = 700 MPa, μs = 0.3 (Natali et al., 2003)). The dental implants are usually manufactured from a titanium alloy. For the purpose of this study, the implant was modeled using Young's modulus and Poisson's ratio of Ei = 110 000 MPa and μi = 0.3 respectively (Mellal et al., 2004).

3. Results

Bones were evaluated for von Mises strains and dental implant was evaluated for von Mises stresses. Both stress as well as strain distributions in the region of interest in all models are presented in Fig. 4. For clarity, only maximum stresses (typically in the second thread from the top) and maximum strains (in first thread from the bottom) were compared in the graph in Fig. 4.

4. Discussion

The results of Model A indicate (especially in comparison with those of Models B1 and C1) that Model A is inappropriate for detailed analysis of bone-implant interaction. The results from this model are not satisfactory as the FE mesh in the region of interest is not fine enough. Such the mesh was employed on purpose to show the differences between this coarse model and the sub-models. If a sensitivity study was carried out and if Model A was run with a sufficient mesh, the calculation would be at least as time-consuming as Models B1 and C1, but more likely much more. It is evident from the graph in Fig. 4 that Model C is sufficient for the investigation of bone-implant interaction, i.e. the segment length of 15 mm is good enough to provide accurate results in a reasonable time.

It is evident, that Models B2 and C2 provide stresses that are significantly lower than those of B1 and C1 because the implant is not subjected to a significant bending that occurs in reality (and in Models B1 and C1). The results of Models D1 and E1 show that it is possible to use boundary condition from coarse model of different mandible (under the condition that a typical geometry and loading are assumed). Maximum von Mises stresses in Model D1 are approx. 7 and 12 MPa lower than in Models B1 and C1.

The maximum difference of 18 MPa can be observed in Model E1; in this case, the difference is caused primarily by different cortical bone thickness as the cortical bone thickness is a significant factor affecting the implant performance (its success or failure). The maximum strain results in all models have similar trends as the stresses in the implants (Fig. 4).

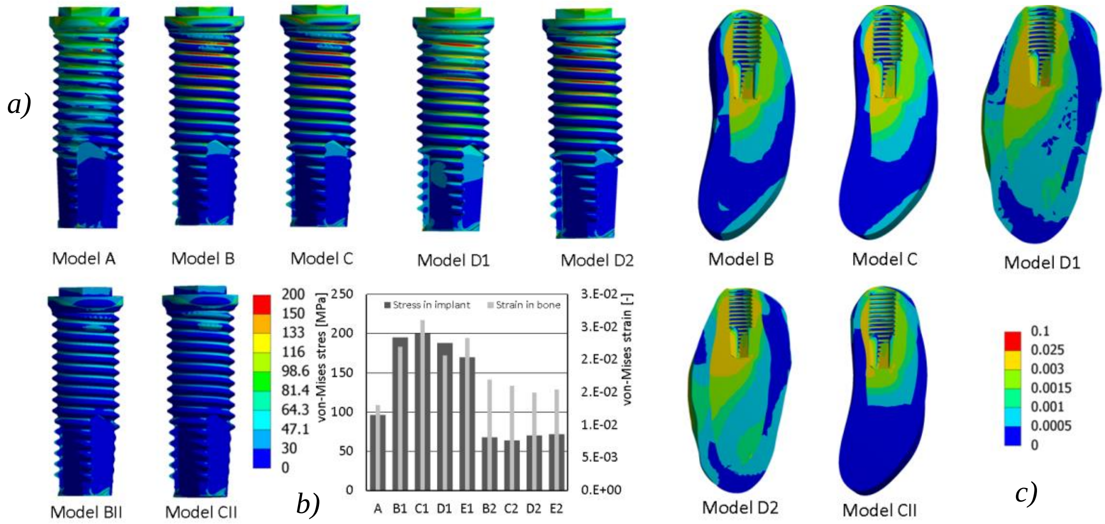


Fig. 4: Results: a) von Mises stress distribution in dental implant; b) maximum von Mises stress in dental implant and von Mises strain in bone; c) von Mises strain in bone-implant.

5. Conclusion

If the coarse model for the particular bone segment model is not available, it is possible to use a generic boundary condition for a typical mandible subjected to a typical loading. In such a case, using a generic boundary condition would be a better option than using a fixed constraint as the results tend to be biased with such an oversimplification. This observation can be helpful in models with a detailed trabecular architecture of cancellous bone. In such a case, it is highly impractical to model the entire mandible as the solution would be extremely time-consuming with no benefit.

Acknowledgement

The research was supported by the Czech Science Foundation by grant No. 16-08944S.

References

Ridwan-Pramana, A., Marcián, P., Borák, L., Narra, N., Forouzanfar, T. and Wolff, J. (2016) Structural and mechanical implications of PMMA implant shape and interface geometry in cranioplasty – a finite element study. Journal of Cranio-Maxillofacial Surgery, 44, 1, pp. 34-44.

Marcián, P., Konečný, O., Borák, L., Valášek, J., Řehák, K., Krpalek, D. and Florian, Z. (2011) On the Level of Computational Models in Biomechanics Depending on Gained Data from Ct/Mri and Micro- Ct, in: MENDEL 2011 - 17th International Conference on Soft Computing. 1. Brno: Brno University of Technology, pp. 255-267.

Huotilainen, E., Jaanimets, R., Valášek, J., Marcián, P., Salmi, M., Tuomi, J., Mäkitie, A. and Wolff, J. (2014) Inaccuracies in Additive Manufactured Medical Skull Models Caused by the DICOM to STL Conversion Process. Journal of Cranio-Maxillofacial Surgery, 42, 5, pp. e259 (e265 s.).

Narra, N., Valášek, J., Hannula, M., Marcián, P., Sándor, G., Hyttinen, J. and Wolff, J. (2014) Finite element analysis of customized reconstruction plates for mandibular continuity defect therapy. Journal of Biomechanics, 47, 1, pp. 264-268.

Marcián, P, Borák, L., Valášek, J., Kaiser, J., Florian, Z. and Wolff, J. (2014) Finite element analysis of dental implant loading on atrophic and non-atrophic cancellous and cortical mandibular bone – a feasibility study. Journal of Biomechanics, 47, 16, pp. 3830-3836.

Natali, A.N., Hart, R.T., Pavan, P.G. and Knets, I. (2003) Mechanics of bone tissue, in: A.N. Natali (Ed.), Dental Biomechanics, Taylor & Francis, London, 2003, pp. 1-19.

Mellal, A., Wiskott, H.W., Botsis, J., Scherrer, S.S. and Belser, U.C. (2004) Stimulating effect of implant loading on surrounding bone. Comparison of three numerical models and validation by in vivo data. Clinical Oral Implants Research 15, 2, pp. 239-248.