Műszaki Tudományos Közlemények vol. 22. (2025) 93–97.
DOI: Hungarian: https://doi.org/10.33895/mtk-2025.22.15
English: https://doi.org/10.33894/mtk-2025.22.15

HIGH PRECISION MODELLING AND STRENGTH ANALYSIS OF TOOTH IMPLANTS

Klaudia PAPP,[1] István Attila PIROS,[1] Bálint DEÁK[2]

[1] *John von Neumann University, Department of Innovative of Vehicles and Materials, Kecskemét, Hungary, papp.klaudia@nje.hu*

[2] *University of Szeged, Faculty of Dentistry, Szeged, Hungary, deakbalint7@gmail.com*

Abstract

The design and mechanical analysis of patient-specific dental prostheses play a crucial role in modern dentistry and biomechanical research. With the advancement of digital design and simulation tools, it has become possible to create customized dental prostheses tailored to individual patients, ensuring not only anatomical compatibility but also optimal mechanical stability.

In our study, we present the design and finite element analysis of a bridge-type dental prosthesis, which was created based on the reconstruction of a real mandible. Our goal was to examine the behavior of the dental prosthesis and mandible under load, with a particular focus on stress distribution and deformations.

Keywords*: Tooth implants, NURBS modeling, Finite element analysis, von Mises stress distribution.*

1. Reconstruction of the teeth and mandible

The aim of our research was to create a high-quality, detailed geometric reconstruction of the mandible and teeth based on CT images. The resulting model provided the basis for further biomechanical and strength studies, which will help to design optimal prostheses and to better understand the mechanical behaviour of the tooth-mandible system.

The first step was to reconstruct the mandible. For this purpose, CT images were segmented in three main planes – axial, sagittal and cortical – using 3D Slicer [1]. This process allowed for a better separation of bony and soft tissues, which facilitated a more accurate modeling [2]. The segmented sections were then imported into the PTC Creo 11 design software, where NURBS curves were created during reconstruction and NURBS surfaces were then spanned onto them. Particular attention was paid to the tangential contact between the curves and to keep the number of surfaces as small as possible, as overly complex geometry may complicate subsequent simulation studies [3]. After reconstruction of the mandible, its structure was divided into two layers: spongiosa and cortical bone layers, to better model their different mechanical properties [4].

The next step was to reconstruct the teeth, following similar principles as for the mandible. The teeth were individually reconstructed from the CT images, initially as a solid structure. Our primary goal in this phase was to accurately reconstruct the geometry of the teeth and to correct errors and inaccuracies that may have occurred in the original CT data. These included, for example, undue cavities, minor topological irregularities or noisy surfaces [5]. In this initial phase, the teeth were not yet stratified, as the focus was first on accurately restoring their external shape.

The completed remodeled mandible is shown in **Figure 1.**

The creation of a complete model provided the opportunity to analyse the mechanical stress capacity of the tooth-mandible system and the behaviour of prostheses and implants under different loading conditions by subsequent biomechanical and strength studies. The reconstruction methods used ensured high resolution and smooth topology of the model, which are essential for accurate simulation studies and medical applications.

2. Tooth implants design and modelling

2.1. High quality modelling of tooth implants

In our research, we aimed to create a patient-specific prosthesis that precisely matches the original anatomical structure of the patient's mandible and teeth, following precise modeling of the mandible and teeth. The prosthesis is a bridge-type restoration designed for two ground teeth. The bridge is bonded to the ground teeth. This solution is considered a fixed tooth implant, which acts as a bridge to remedy the tooth deficiency and provides the patient with a durable, functionally load-bearing restoration.

The bridge designed for existing teeth is shown in **Figure 2.**

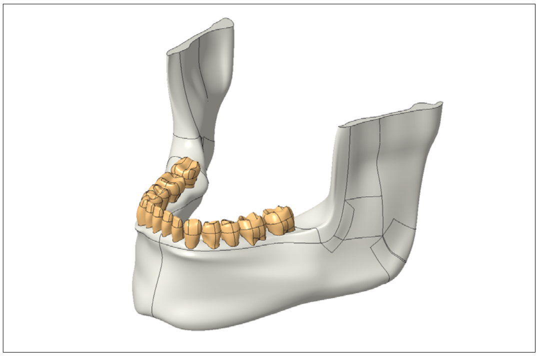


Fig. 1. *Mandible modelled from CT images.*

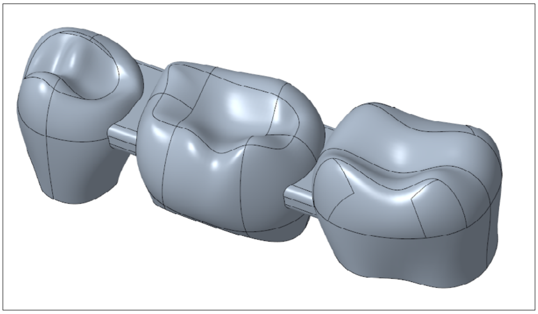
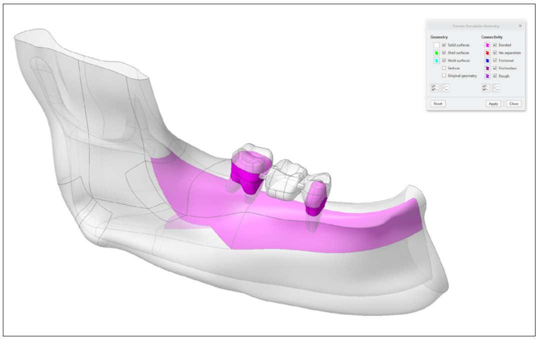


Fig. 2. *The bridge type of implant*

The geometry of the bridge was created by geometrically offsetting the surfaces of the original teeth, thus ensuring a proper fit of the prosthesis on the surface of the extracted teeth **[6]**.

The fit of the restoration on the ground teeth is shown in **Figure 3.**



Preview Simulation Geometry

Geometry
☐ ☑ Solid surfaces
☑ Shell surfaces
☑ Weld surfaces
☐ Vertices
☐ Original geometry

Connectivity
☑ Bonded
☑ No separation
☑ Frictional
☑ Frictionless
☑ Rough

Fig. 3. *The fit and connection of the implant to the ground teeth.*

In the picture, the area marked in pink is the spongiosa layer of the mandible, which forms part of the supporting structure of the teeth, and the two ground teeth on which the bridge rests. These parts are connected to each other in a bonded way to prevent displacement **[7]**.

The geometry of the implant closely follows the patient's individual anatomy to ensure proper fit and stability. The structural design of the bridge is such that the bite forces are properly distributed over the cantilever teeth, minimising the risk of overloading.

The bridge-type implant thus created provides an opportunity for further mechanical analyses that can help optimise the final design and facilitate preparation for clinical application.

2.2. Strength analysis of tooth implant

Following the design of a patient-specific bridge-type implant, a strength analysis was performed in PTC Creo 11 software using the integrated Ansys simulation module. Our goal was to investigate the mechanical behaviour and resistance of the implant in a static loading environment.

The following boundary conditions and loads were applied in the finite element analysis:

- Boundary conditions: the arthroplasty and crown protrusion of the mandible were fixed in order to investigate the loading behaviour of the implant under realistic conditions, and, in addition, since only the half of the mandible was investigated in order to run the simulations faster, a symmetry condition was added to the mandible where this mandible was cut, so that displacement was only allowed along the axis of the cut surface.
- Load: a static force of 100 N in the vertical direction was applied to the bite surface of the middle tooth of the bridge, corresponding to an average bite force **[8]**.

The applied boundary conditions and loads are shown in **Figure 4.**

During the modeling of the implant, care was taken to define the bite surfaces as separate, well-defined surfaces **[9]**. As shown in **Figure 5** this allowed for accurate load assignment and precise determination of the actual mechanical stresses on the implant.

Before starting the strength analysis, the whole model was meshed with finite element meshing to ensure the accuracy of the numerical calculations **[10]**. This meshing is shown in **Figure 6.**

For meshing, a tetrahedral finite element mesh was used, which is particularly suitable for accurately mapping complex organic geometries such as the shape of the mandible and the dentition **[11]**.

The tetrahedral mesh is an excellent fit for the curved, complex shape of the mandible and tooth implant, as tetrahedral elements fill the available volume more efficiently than conventional hexahedral elements. Another significant advantage is that it can be generated automatically, requiring less manual intervention, which makes it faster and easier to apply to organic shaped models. In addition, to achieve accurate stress distribution results, it is possible to vary the element size to create a denser mesh in critical areas, improving the accuracy and reliability of the analysis. The tetrahedral elements are able to handle more complex boundary conditions and loading situations, which was a particularly important consideration in the modeling of implant.

By constructing the meshing, we ensured that the mandible, the implant and the fixed teeth were represented at the appropriate resolution in the finite element analysis, thus obtaining more accurate and detailed results on their loading behaviour.

The first step in the finite element simulations was to investigate the total deformation, which shows the extent to which static loading causes parts of the model to move from their original position. The results are shown in **Figure 7,** which shows that the largest deformation was at the lower part of the mandible, at the mandible tip, where the maximum displacement was 0.1706 mm.

This result suggests that the mandible undergoes minimal elastic deformation under the applied 100 N bite load, which is a natural phenomenon. The greater displacement in the mandible is explained by the fact that this area is furthest from one of the fixed boundary conditions (mandibular arthroplasty and crown protrusion) and therefore has a greater degree of freedom to move **[12]**.

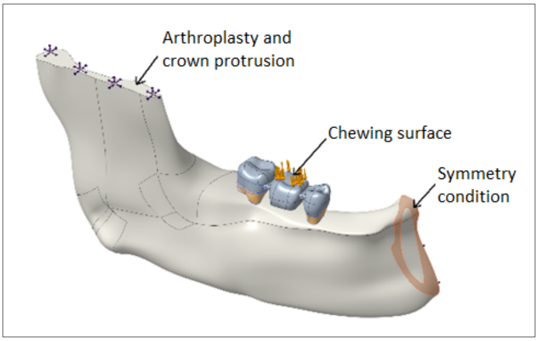
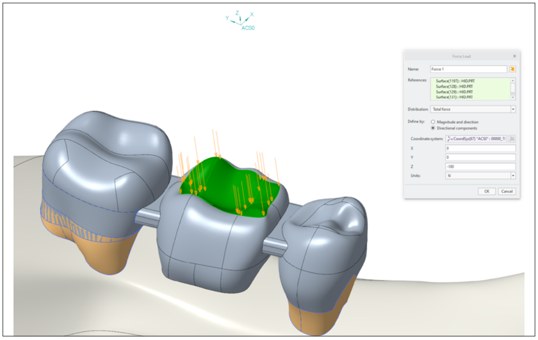


Fig. 4. *Loads and boundary conditions.*



Force Load

Name: Force 1

References:
Surface(1197) : HID.PRT
Surface(128) : HID.PRT
Surface(129) : HID.PRT
Surface(131) : HID.PRT

Distribution: Total force

Define by:
○ Magnitude and direction
● Directional components

Coordinate system: CoordSys(67) "ACS0" : 00000_T

X 0

Y 0

Z -100

Units: N

OK Cancel

Fig. 5. *Application of the load to the bite surface.*

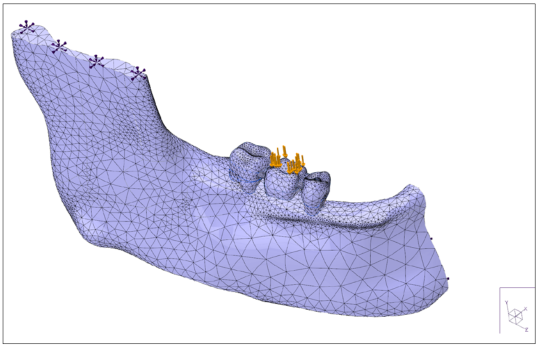


Fig. 6. *Meshing on the mandible.*

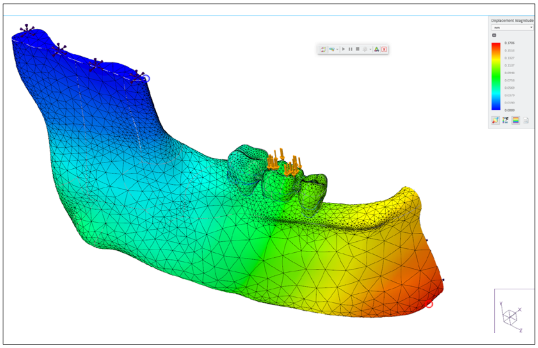


Fig. 7. *Total deformation.*

As a next step in the finite element simulation, a von Mises stress test was performed to analyse the stress distribution in the implant and mandible material under static loading. The result of the stress test is shown in **Figure 8.**

In the case of the mandible, the highest stress was in the ramus region, where the maximum value exceeded 10 MPa. This result is in agreement with the fact that the ramus plays a key role in the load transmission due to the masticatory force, and therefore the greatest mechanical stress is concentrated in this area **[13]**.

In the case of implant, the highest stresses were found on the bridge structure, which is located between the two grinded teeth. Here, the principal stress reached 33.9 MPa, indicating that this part of the implant is a highly stressed zone. This value is also significant because the main loading point of bridges is the middle part, where bending and shear stresses are generated due to the bite force **[13]**.

The maximum stresses on the bridge are shown in **Figure 9** and **10.**

The results show that both the mandible and the implant structure exhibit suitable mechanical resistance, as the applied stresses are within the expected range **[14]**. Further fine-optimization of the implant geometry can help to reduce the maximum stress values and to distribute the forces more evenly.

3. Conclusions

The aim of our research was the design and mechanical testing of a patient-specific implant based on the geometric reconstruction of a real mandible. The planned implant is a bridge-type restoration that was fitted on two ground teeth. The complete model was created and simulation studies were performed in PTC Creo 11 software using the Ansys finite element module.

As a first step in the simulation process, the entire structure was fitted with a tetrahedral finite element mesh, which is an excellent tool for tracing the complex geometry more accurately. Subsequently, mechanical analyses were carried out, treating the mandibular arthroplasty and crown protrusion as boundary conditions and the mandible cut surface as a symmetry condition, while the loading of the implant was modelled by a static force of 100 N in the vertical direction applied to the bite surface of the central tooth of the bridge.

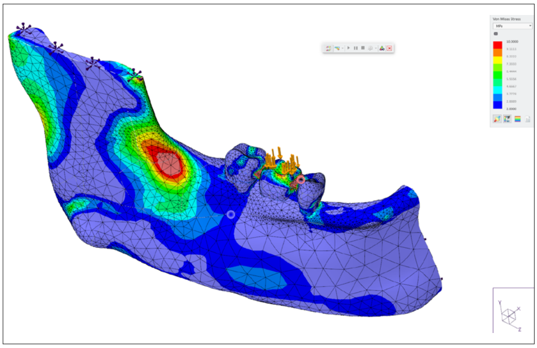


Fig. 8. *Von Mises stress analysis.*

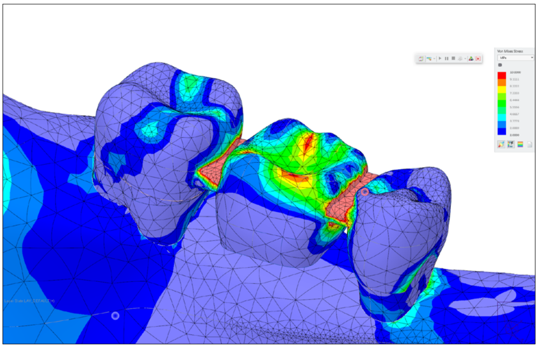


Fig. 9. *Maximum stress on the bridge, top view.*

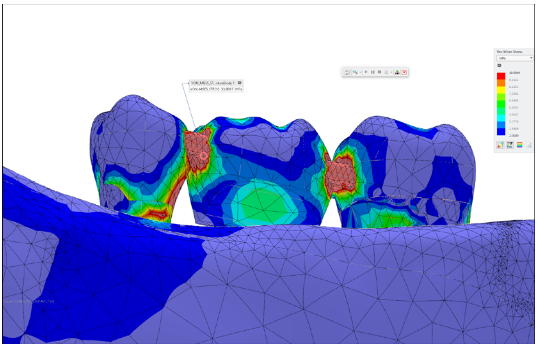


Fig. 10. *Maximum stress on the bridge, bottom view.*

The results showed that the maximum value of the total deformation was 0.1706 mm, which occurred at the lower part of the mandible. When the stress distribution was examined, the highest von Mises stress was developed in the ramus part of the mandible, exceeding 10 MPa. In the case of the implant, the most stressed zone was the central part of the bridge, where the stress increased to 33.9 MPa.

References

[1] Wang C.-S., Lin M.-C., Wang C.-C., Chen C.-F., Hsieh J.-C.: *Feature Reconstruction for 3D Medical Images Processing*. In: Gao J.X., Xu D., Sun X., Wu Y. (eds.) *Proceedings of the 2013 6th International Conference on Biomedical Engineering and Informatics* (BMEI 2013), Vol. 1 and 2, 69–74. IEEE, New York (2013).
https://doi.org/10.1109/BMEI.2013.6746909

[2] Nguyen, V.S., Tran, M.H., Le, S.T.: *Visualization of Medical Images Data Based on Geometric Modeling.* In: Dang, T.K., Kng, J., Ta-kizawa, M., Bui, S.H. (eds.) *Future Data and Security Engineering.* Lecture Notes in Computer Science, pp. 560–576. Springer, Cham (2019).
https://doi.org/10.1007/978-3-030-35653-8 36

[3] Aubin, C.-., Dansereau, J., Parent, F., Labelle, H., Guise, J.A.: *Morphometric Evaluations of Personalised 3D Reconstructions and Geometric Models of the Human Spine.* Medical and Biological Engineering and Computing, 35/6. (1997) 611–618.
https://doi.org/10.1007/BF02510968

[4] AlZubi, S., Jararweh, Y., Al-Zoubi, H., Elbes, M., Kanan, T., Gupta, B.: *Multiorientation Geometric Medical Volumes Segmentation Using 3D Multiresolution Analysis.* Multimedia Tools and Applications, 78/17. 24223–24248 (2019)
https://doi.org/10.1007/s11042-018-7003-4

[5] He, J., Chen, S., Zhang, H., Tao, X., Lin, W., Zhang, S., Zeng, D., Ma, J.: *Downsampled Imaging Geometric Modeling for Accurate CT Reconstruction via Deep Learning*. IEEE Transactions on Medical Imaging, 40/11. (2021) 2976–2985
https://doi.org/10.1109/TMI.2021.3074783

[6] Liu, C.-C., Hsu, C.-H., Hsiao, I.-T., Lin, K.M.: *Effect of Geometric Models on Convergence Rate in Iterative PET Image Reconstructions.* Journal of Instrumentation, 4(05), 05010 (2009)
https://doi.org/10.1088/1748-0221/4/05/P05010

[7] Garrido, A.H., Cerra, P.P., Zapico, A.O., Lopez, M.D.G.: *Geometric Data Collection in Medical Imaging*. DYNA, 86/2. (2011) 222–231.
https://doi.org/10.6036/3868

[8] Pemmada, R., Telang, V.S., Tandon, P., Thomas, V.: *Patient-Specific Mechanical Analysis of PCL Periodontal Membrane: Modeling and Simulation.* Journal of the Mechanical Behavior of Biomedical Materials, 151. (2024) 106397
https://doi.org/10.1016/j.jmbbm.2024.106397

[9] Franchini, S., Gentile, A., Vassallo, G., Vitabile, S.: *Implementation and Evaluation of Medical Imaging Techniques Based on Conformal Geometric Algebra*. International Journal of Applied Mathematics and Computer Science, 30/3. (2020) 415–433.

[10] Abdel-Aziz, H.S., Zanaty, E.A., Ali, H.A., Saad, M.K.: *Generating Bezier Curves for Medical Image Reconstruction.* Results in Physics, 23. (2021) 103996
https://doi.org/10.1016/j.rinp.2021.103996

[11] Majeed, A., Mt Piah, A.R., Rafique, M., Ab-dullah, J.Y., Rajion, Z.A.: *NURBS Curves with the Application of Multiple Bones Fracture Reconstruction.* Applied Mathematics and Computation, 315. (2017) 70–84.
https://doi.org/10.1016/j.amc. 2017.05.061

[12] Lv, X., Fu, D.: *Research on Rapid Imitation of Human Tibia and Five-Axis CNC Machining Based on Computer-aided.* Journal of Physics: Conference Series, 1648(3), 032130 (2020)
https://doi.org/10.1088/17426596/1648/3/032130

[13] Yao, Q., Zhuang, Y., Aji, Y., Zhang, Q., Luo, Y., Li, S.: *Biomechanical Impact of Different Isthmus Positions in Mandibular First Molar Root Canals: a Finite Element Analysis.* Clinical Oral Investigations, 28/6. (2024) 311
https://doi.org/10.1007/s00784-024-05715-1

[14] Demir, O., Uslan, I., Buyuk, M., Salamci, M.U.: *Development and Validation of a Digital Twin of the Human Lower Jaw under Impact Loading by Using Non-Linear Finite Element Analyses.* Journal of the Mechanical Behavior of Biomedical Materials, 148, (2023) 106207.
https://doi.org/10.1016/j.jmbbm.2023.106207