Christopher Fleischmann*, Irina Leher, Reinhold Hartwich, Marc Hainke, Stefan Sesselmann

A new approach to quickly edit geometries and estimate stresses and displacements of implants in real-time

Abstract: Although osteosynthesis plates rarely break, such cases still occur. Since the construction of these plates is mainly focussing on the best fit for anatomical structures, they may not provide optimal mechanical stability. It should be considered whether they might be changed in terms of e.g. hole size and number. Finite element analysis can help to estimate whether the plates will withstand loads or break. However, these analyses can be very time-consuming. To counteract this problem, a new software solution (ANSYS Discovery Live) has been established for estimation of stresses in real time. In this study the accuracy of the new software was compared to a conventional finite element analysis software solution to investigate whether this software solution provides appropriate accuracy to be used for medical applications.

Keywords: osteosynthesis plate; shape enhancement; finite element method; finite element analysis

<https://doi.org/10.1515/cdbme-2019-0139>

1 Introduction

Breakage of osteosynthesis plates can lead to complicated follow-up surgeries. The reasons for such rarely occurring events are varying. Important factors are the number of screw holes, the number of inserted screws and the insertion of the screws too closely to the fracture line [1]. Osteosynthesis plates have to comply with a lot of clinical requirements, especially with a good adaption to the human bone [2]. Parameters like the plate size, plate positioning and the number of screw holes must be optimized to prevent the plates from breaking.

***Corresponding author: Christopher Fleischmann:** Institute for Biomedical Engineering, Hetzenrichter Weg 15, Weiden, Germany, e-mail: c.fleischmann@oth-aw.de

Irina Leher, Reinhold Hartwich, Marc Hainke, Stefan Sesselmann, Institute for Biomedical Engineering, Hetzenrichter Weg 15, Weiden, Germany

Finite element analysis (FEA) is a powerful tool to reduce prototyping expenses and accelerate development processes. The FEA has a wide range of applications in the field of biomechanics. Besides calculating stresses and deformations of anatomical structures like bones and joints also new implant designs are routinely assessed by using FEA.

Even though those analyses can help to reduce development processes, the procedure can be time-consuming as well, e.g. because of remodelling, remeshing and recalculating. A new promising approach is ANSYS Discovery Live (ANSYS, Inc., Canonsburg, Pennsylvania, US) which is an innovative real-time FE-solution to assess stresses, displacements or other mechanical parameters under conditions of varying geometries and boundaries without the need of meshing tools. Object geometry can be edited easily by just shifting edges of the 3D-model by mouse dragging. New results are generated without time-consuming remeshing and adaption of boundary conditions in real-time.

Aim of this study was a comparing evaluation of ANSYS Discovery Live with the well-established FE solution ANSYS Workbench (ANSYS, Inc., Canonsburg, Pennsylvania, US). The maximum von Mises stress, the stress contour and the maximum deformation were calculated for comparison of these two software solutions.

2 Material and Methods

In this work a palmar-mounted titanium plate for osteosynthesis of fractures of the distal radius was used for FEA (Figure 1). A CT-scan provided the shape of the plate. Afterwards Catia V5 (Dassault Systèmes, Vélizy-Villacoublay, France) was used to obtain the geometry. The plate was then meshed with tetrahedral elements using HyperWorks (Altair Engineering Inc., Troy, Michigan, US). The model has a total number of about 1×10^6 elements. Mechanical properties for the calculations are defined with a Young's modulus of $E = 140$ GPa and a Poisson's ratio of $\nu = 0.33$.

Boundary conditions, like compression and bending forces, were applied as stated in literature [3, 4]. For the

Figure 1: Reverse engineered palmar-mounted osteosynthesis plate for the distal radius

compression load case, the plate has been rigidly fixed at the four proximal elongated holes. The force has been applied on the distal end with a magnitude of $F = 2000$ N. For the four-point bending test, a maximum load of $F = 250$ N was applied [4]. The distance between the two supporting and the two loading areas was 10 mm, each with the first fixation located at the most proximal hole.

The maximum von Mises stress, the stress contour and the maximum deformation were calculated for comparison of the different software solutions. The displacement and the stress for compression were measured at the node with the maximum value. The stress for the four-point bending test was evaluated at defined measurement points in order to avoid evaluating stress singularities due to the boundary conditions.

3 Results

The maximum displacement calculated in ANSYS Discovery Live is 0.0185 mm for the compression load case and 0.0487 mm for the four-point bending test. ANSYS Workbench shows similar results with values of 0.0189 mm and 0.0512 mm, respectively.

Furthermore, the von Mises stress is in a similar range with 26.09 MPa and 33.22 MPa, respectively for the four-point bending test at the measurement points. For the compression test the maximum von Mises stress amounts 467.04 MPa and 457.42 MPa, respectively.

Stress contours of the simulations are comparable besides slight aberrations (Figure 2).

4 Discussion

Despite deviations of the calculated form of displacements and stresses, ANSYS Discovery Live simulates similar contour levels. Although the results of ANSYS Discovery

Figure 2: Von Mises stress contour levels for a) ANSYS Workbench and b) ANSYS Discovery Live

Live are very similar to the results of ANSYS Workbench, the maximum stress should not be used as an indication for critical breakage values. Nevertheless, at least locations of stress concentrations (e.g. notch stress) can be determined. Thus, geometries can be edited to reinforce high stress areas.

5 Conclusion

This comparing study has certain limitations: There is no possibility to check the number and types of elements nor the mesh quality. This can lead to calculation problems if complex geometries are calculated. As ANSYS Discovery Live is not designed for FEA of anisotropic material, the

software can be used to analyse the implant itself but not under conditions mounted on a bone. Hence, ANSYS Discovery Live is a suitable tool for an accelerated preliminary implant design process. To evaluate the *in vivo* performance of the implant mounted on a bone, it is still necessary to work with FEA solutions that are able to also work with anisotropic material properties, like ANSYS Workbench.

Author Statement

Research funding: The authors state no funding involved.

Conflict of interest: Authors state no conflict of interest.

References

- [1] Lv H, Chang W, Yuwen P, Yang N, Yan X, Zhang Y. Are there too many screw holes in plates for fracture fixation?. BMC Surg. 2017 Apr 21;17(1):46.
- [2] Tkany L, Hofstätter B, Petersik A, Miehl J, Wartzack S, Sesselmann, S. New Design Process for Anatomically Enhanced Osteosynthesis Plates. Wiley Online Library 2019.
- [3] Okazaki Y, Gotoh E, Mori J. Strength-Durability Correlation of Osteosynthesis Device Made by 3D Layer Manufacturing. Materials 2019
- [4] Standard Specification and Test Method for Metallic Bone Plates, ASTM F382 - 17, Book of Standards Volume: 13.01