

FAILURE ANALYSIS OF A CUSTOM-MADE ACETABULAR CAGE WITH FINITE ELEMENT METHOD

Martin O. Dóczy
Péter T. Zwierczyk

Department of Machine and Product Design
Budapest University of Technology and
Economics
Műegyetem rkp. 3., Budapest 1111, Hungary
doczi.martin@gt3.bme.hu
z.peter@gt3.bme.hu

Róbert Szódy

Péterfy Hospital and Manninger Jenő
National Institute of Traumatology
Fiumei street 17., Budapest 1081, Hungary
robert.szody@gmail.com

KEYWORDS

Acetabular bone defect, Acetabular cage, Patient-specific, Finite element analysis

ABSTRACT

Research significance: In this paper, a custom made acetabular cage was studied in order to calculate the mechanical stresses of the implant. The goal is to have a validated finite element model, which can provide qualitatively accurate results.

Methodology: The geometry models of the acetabular cage, the pelvis and the fixation are based on the patient's computer tomography data. The geometry models were made during a reverse engineering and surface modeling process.

The hemipelvis was modeled, according to the literature research. The boundary conditions, loading model, material properties were from the literature research as well.

Results: Analyzing the von Mises stresses of the acetabular cage and its screws, the most loaded areas could be detected. Viewing the first and the third principal stresses, the tensioned and the compression areas could be separated. With the patient's control CT data, the results could be validated, analyzing the deformation of the cage, and the fracture of an implant.

Discussion: The FE simulations of the acetabular cage provide results, which are consistent with observed implant failure.

INTRODUCTION

The total hip replacement is an effective solution to treat osteoarthritis, reducing hip pain and the patients can have a better daily living. During the procedure, a socket is inserted in the acetabular part of the pelvis, and a metal stem into the hollow part of the femur. A metal or ceramic ball is placed on the stem, and usually a polymer liner inside the socket.

However, this solution usually requires a revision surgery after 10-20 years, where the reparation of damaged prosthesis' elements are done.

In some cases, one has to deal with large acetabular bone defects (Figure 1).

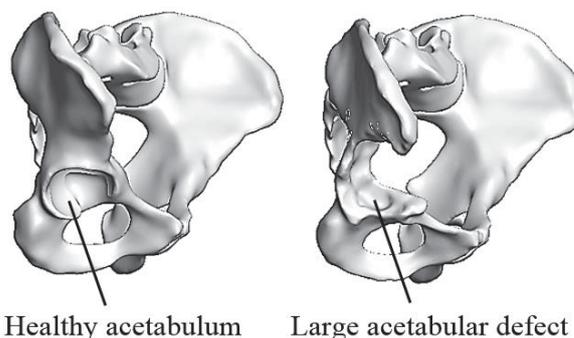


Figure 1: Different clinical states of the pelvis

The treatment of a large acetabular bone defect is a challenging clinical task. There is no consensus about a general treatment, and it is a hard problem because these are always very individual cases. (Ahmad and Schwazkopf 2015; Paprosky et al. 1994)

In this study, the treatment was a cold worked, custom made sheet metal acetabular cage by Róbert Szódy and his colleagues. (Szódy et al. 2017)

The patient could walk again, but after one year from the surgery, small plastic deformation of the cage and a screw's fracture could be discovered on the patient's CT examination.

The task is to make a finite element model which can provide qualitatively accurate results for the stress concentration areas. This can help later to detect the zones where geometry improvement is required.

DATA AND METHOD

Data

There were available three Computer Tomography (CT) data of the patient. The first represented the status before the surgery (pre-operative CT), the second was after the surgery (post-operative CT), and the third was the control after one year (control CT).

Each CT data had a different role for the simulation. The geometry model of the pelvis based on the first CT. For making the geometry model of the acetabular cage and the positions of the screws, the second CT was used.

With the control CT, it was possible to validate the simulation results.

The CT data is a three-dimensional scalar field, where each voxel has a so-called Hounsfield Unit value, which is related to the small volume-parts X-ray attenuation. If the attenuation is great, then the Hounsfield value is higher as well. Hence it is possible to represent the different density regions throughout the CT data.

Geometry Models

The CT data were imported to Slicer 3D for the geometric reconstruction. With this software, one can volume render the relevant areas from the CT and export it in “.stl” file format. After that procedure, the CAD model can be generated with a reverse engineering workflow.

Firstly, the CAD model of the pelvis was made. For this, the pre-operative CT was used.

With a global threshold-based segmentation, the pelvis bone was highlighted. For the accurate geometry model, a manual selection of the relevant parts was required. Then the “.stl” file could be saved.

In MeshMixer, the small defects of the “.stl” file and the holes were eliminated. With a smoothing procedure, it became a manifold hemipelvis model.

For the surface reconstruction, a SolidWorks 2018 CAD system with Scanto3D module was used.

After importing the “.stl” file, an automated surface reconstruction method was chosen, with manually adjusted feature lines. Then, a closed surface model was generated, which could be transformed into a solid body. The pelvis areas which were close to the acetabular defect were separated from the intact parts of the pelvis, because there a different material models were used. The pelvis model can be seen in Figure 2.

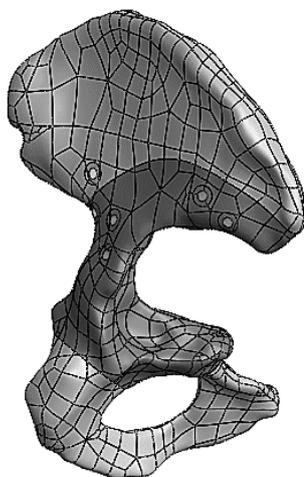


Figure 2: The CAD model of the pelvis

The acetabular cage is a surface model. The post-operative CT data was used to make the CAD model for the cage and its screws. These metal parts were highlighted with a global threshold-based segmentation method. In this case, it was not necessary to use manual segmentation. Then the “.stl” file was exported. The mid surface of the acetabular cage was modeled with reverse engineering and surface modeling tools, using SolidWorks 2018.

The segmented .stl file and the CAD models of the acetabular cage and its screws can be seen in Figure 3.

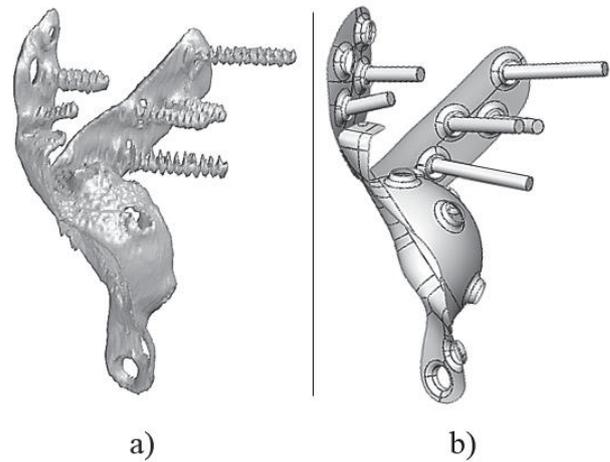


Figure 3: The .stl file of the acetabular cage and its screws (a) and the CAD model (b)

The screws have simplified geometry. Instead of modeling the threads, cylindrical geometries were made. The diameter of the cylinder was the nominal size of the thread (4.5 mm). The head of the screw was made according to the relevant standard (ISO 5831:1991). This was simplified spherical geometry, without the hexagonal drive hole.

For the assembly, the post-operative CT data was used, which helped to position the center of rotation, and the positions of the screws.

The liner and the balls were made as a revolved bodies. It can be seen on Figure 4 the parts and their names in an exploded views.

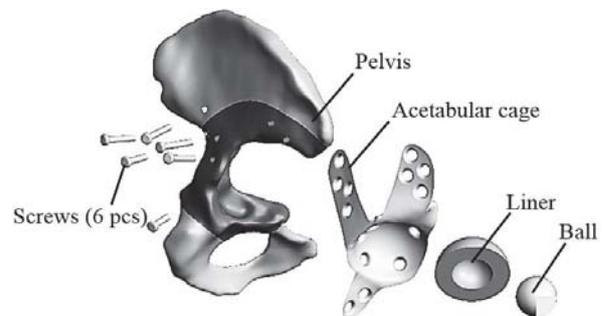


Figure 4: The parts of the fixation in an exploded view

Finite Element Model

For the preprocessing of the finite element model, HyperMesh 2017.2 was used.

For the meshing of the solid models, 10-node tetrahedral elements were used. The finite element mesh of the pelvis has an average 2 mm edge length.

The intact areas of the pelvis are covered by shell elements, representing the cortical bone layer. The thickness of the shell elements was 1 mm. These were linear triangular elements. (Plessers and Mau 2015)

Previously mentioned, the acetabular cage was a surface model. Hence, shell elements were used there as well. The thickness of the shell elements was 1.5 mm, which was the real thickness of the sheet metal. Four node linear quads were used for the meshing of the acetabular cage.

The Optistruct solver does not support the quadratic shell element for large displacement analysis, that was the reason for using linear shell elements.

Manual mesh refinement was only used on the acetabular cage because this part is on the focus of the investigation. The data of the mesh can be seen in Table 1.

Table 1: The data of the FE mesh

| | |
|---------------------------------------|--------|
| Number of nodes | 244331 |
| Number of elements | 176428 |
| Number of 10 node tetra elements | 145562 |
| Number of 4 node quads | 14504 |
| Number of 3 node trias | 16362 |
| Maximum Aspect ratio (solid elements) | 5.28 |
| Maximum Aspect ratio (shell elements) | 4.46 |

The material properties were homogenous, linear elastic and isotropic.

It was previously mentioned that the pelvis had different material models. The healthy pelvis is a so-called irregular bone. On its surface, there is a thin layer of compact bone, called cortical bone, and inside this sandwich structure, there is the trabecular bone.

Due to the acetabular component migration, it can be seen on the pre-operative CT data, that near by the bone defect it cannot be separated easily the cortical and the trabecular bone. For this case, a homogenous part is modeled with averaged Young's modulus. (Anderson et al 2004, Ravera et al. 2015)

The acetabular cage is a cold worked sheet metal part. There was no annealing after the forming. Due to the strain hardening effect, the yield strength is increased in the formed regions. It is a challenging task to calculate the residual stresses and the modified yield strengths for further simulations, thus, linear elastic material properties were used.

The summary of the material properties can be seen in Table 2.

Table 2: Material properties

| | Young's modulus [MPa] | Poisson's ratio [-] |
|-------------------|-----------------------|---------------------|
| Steel (AISI 316L) | 192000 | 0.3 |
| XLPE | 1000 | 0.4 |
| Cortical bone | 17000 | 0.3 |
| Trabecular bone | 100 | 0.3 |
| Homogenous bone | 7000 | 0.3 |

The material of the ball and the screws were steel, homogenous, linear elastic and isotropic properties were used for the analysis with the steel's Young's modulus and Poisson's ratio.

There were bonded contact between the liner and the acetabular cage because these have a glued connection. The screws with the pelvis had bonded contact as well. The threaded connection was simplified with this modeling procedure.

The other metal – bone interfaces had frictional contact with a 0.3 frictional coefficient, as well as the metal-metal interfaces, but there 0.23 frictional coefficient was used. (Chih-Wei Chang et al. 2014)

There is small sliding friction between the ball and the liner, for numerical stability, a small (0.02) frictional coefficient was used.

According to the literature research, there are only a few publications where the muscular forces were modeled. This study is focusing on the acetabular cage, so the loading model and the boundary conditions were a bit simplified but consistent with the literature. It can be seen in Figure 5.

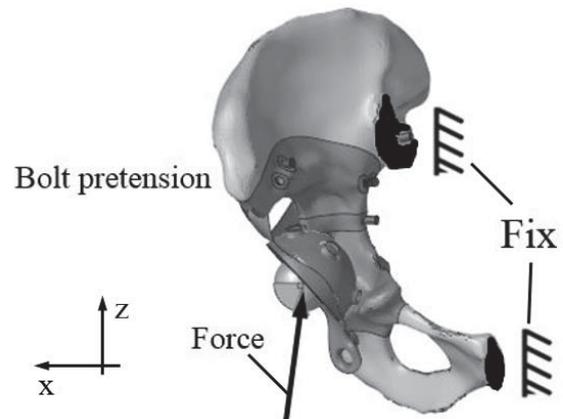


Figure 5: Loading model and boundary conditions

The main load was the maximum of the gait cycle because this is the most common load for this kind of implant. The components of this load were from the literature research. The coordinate system of the pelvis was the same what Bergmann et al. used. (Bergmann et al., 2001) The peak magnitude of the load is 233% Bodyweight. The patient's weight was approximately 75 kg. The components of the main load can be seen in Table 3.

Table 3: Components of the main load

| Force component | Value |
|-----------------|----------|
| X | -213.8 N |
| Y | -194.5 N |
| Z | 1690.8 N |

There was a fix boundary condition at the sacroiliac joint and the pubic symphysis. (Plessers and Mau 2016) For all simulations, the large displacement analysis was selected.

Bolt pretension was applied on the screws before the main load, for closing the contacts and modeling the post-operative situation. The magnitude of the bolt pretension was 50N on each screw.

RESULTS

Finite Element Results

The displacement field was as expected, in the direction of the load. The z-direction displacement field after the main load, can be seen in Figure 6.

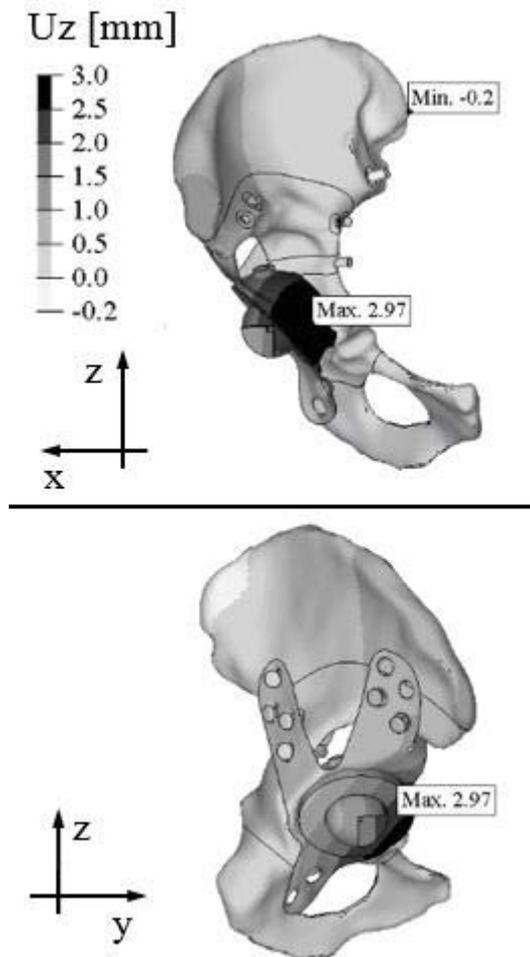


Figure 6: z-direction displacement field of the fixation

The von Mises stresses of the acetabular cage can be seen in Figure 7.

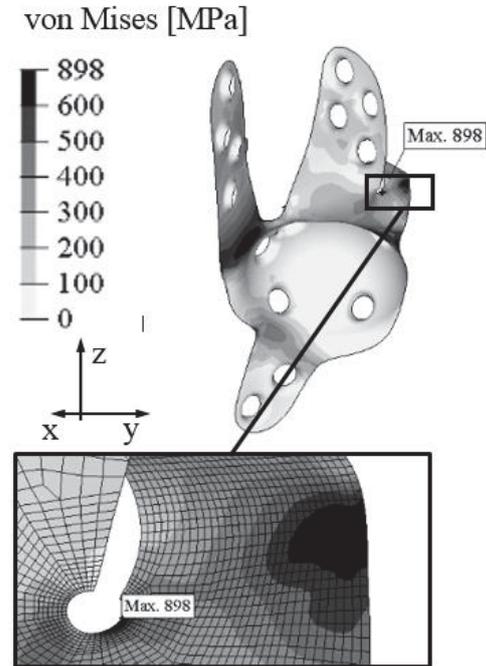


Figure 7: The von Mises stress on the acetabular cage

For investigating the tensioned and compressed zones, the first and third principal stresses were represented. In Figure 8 can be seen the different areas.

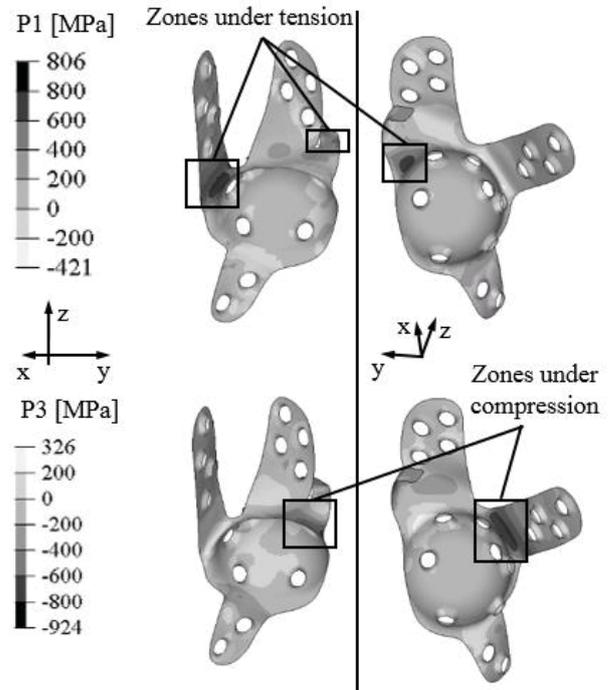


Figure 8: The first and the third principal stresses on the acetabular cage

Investigating the von Mises stresses on the screws, it can be seen which had the largest von Mises stress. It is way below the material's Yield strengths, but it is important that the screws had simplified cylindrical geometry, which did not have so special stress concentrating areas as a threaded geometry, and they had a larger section as well. The loading of the screws was bending, this can be seen after viewing the first and the third principal stresses (Figure 9).

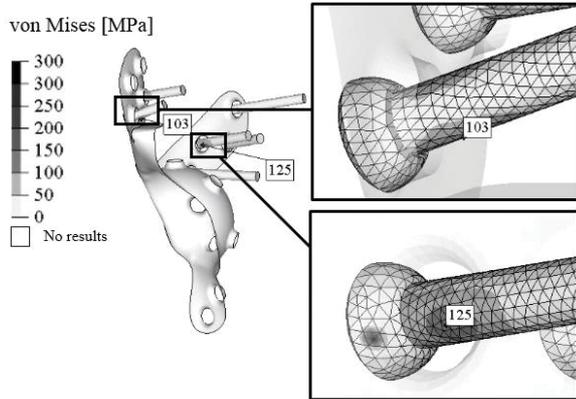


Figure 9: The most loaded screws and its von Mises stresses (cage's visibility transparent)

Validation

After registering the pre-operative and the post-operative stl meshes, there can be seen the main deformation areas. (Figure 10). These are the same, where the large stresses had developed.

The results are qualitatively accurate because the model can predict where are the tensioned or compressed zones. The sheet metal acetabular cage's main loading is bending. Therefore, it can be seen where the sheet metal deform inward or outward (dent or protrusion and deflections). On the side where protrusion occurred, there was the tensioned part, where dent occurred, there was the compressed area.

The lighter gray stl mesh is the deformed cage (from control CT), and the darker one is the undeformed (from post-operative CT).

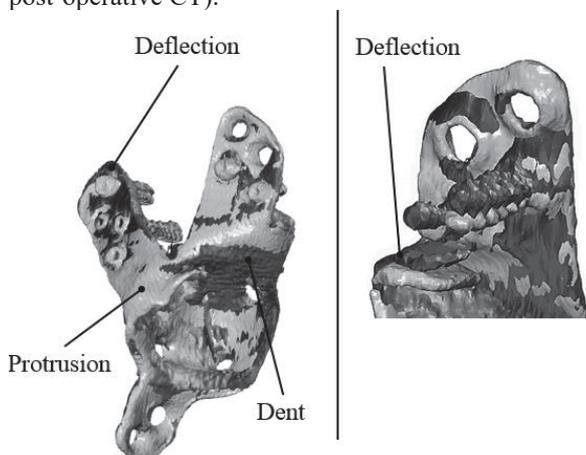


Figure 10: The deformations of the acetabular cage

Another good option for validation is that the fractured screw is the same, which has a large von Mises stress. It can be seen in Figure 11.

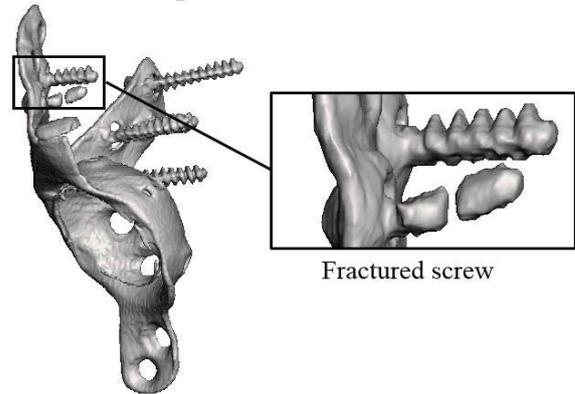


Figure 11: Fractured screw

DISCUSSION

The FE simulations of the acetabular cage provide results which are consistent with observed implant failure. The deflections, dents, and protrusions occurred in the same areas, where were the maximum von Mises stresses. Analyzing the first and the third principal stresses, it could be seen that the results are qualitatively quite accurate because the tensioned and compressed side could be separated, and the characteristic of the deformations could represent the similarity.

There were made geometry simplifications to modeling the pelvis, the screws and the cage. No one of these could provide large effect for the results because the geometries were quite accurate. Because of the larger diameters on the threads, the mechanical stresses were obviously inaccurate, which could provide qualitative results. With the simplification of the cage and the screws, the computational cost was reduced, but with these models, the local stresses, for example near the contacts between the bolt heads and the cage could not be investigated, only the stresses far from these zones.

The more accurate modeling of the material properties of the bone was not a trivial task, but the authors think that this had not a large impact for the stresses of the cage.

Next, the authors want to make more analyses for other cages, and checking the sensitivity of the results for different parameters, for example different material properties, bolt pretension etc.

ACKNOWLEDGMENT

The research reported in this paper was supported by the Higher Education Excellence Program of the Ministry of Human Capacities in the frame of Artificial intelligence research area of Budapest University of Technology and Economics (BME FIKP-MI), and by the National Research, Development and Innovation Fund (TUDFO/51757/2019-ITM, Thematic Excellence Program).

REFERENCES

- Ahmad, A. and Schwarzkopf, R. 2015. "Clinical evaluation and surgical options in acetabular reconstruction: A literature review." *Journal of Orthopaedics* 12 (2): S238-S243
- Anderson, A. et al. 2005. "Subject-Specific Finite Element Model of the Pelvis: Development, Validation and Sensitivity Studies." *Journal of Biomechanical Engineering* 27 (3): 364-373
- Bergmann, G. et al. 2001. "Hip contact forces and gait patterns from routine activities." *Journal of Biomechanics* 34 (7): 859-891
- Chih-Wei Chang et al. 2014. "Role of the compression screw in the dynamic hip-screw system: A finite-element study." *Medical Engineering & Physics* 37 (112): 1174-1179
- ISO 5831:1991 "Implants for surgery - Metal bone screws with hexagonal drive connection, spherical under-surface of head, asymmetrical thread -Dimensions"
- Paprosky, W., Perona, P. and Lawrence, J. 1994. "Acetabular defect classification and surgical reconstruction in revision arthroplasty: A 6-year follow-up evaluation." *The Journal of Arthroplasty* 9 (1): 33-44
- Plessers, K. and Mau, H. 2016. "Stress Analysis of a Burch-Schneider Cage in an Acetabular Bone Defect: A Case Study." *Reconstructive review*. 6 (1): 37-42
- Ravera, E. et al. 2015. "Combined finite element and musculoskeletal models for analysis of pelvis throughout the gait cycle." *Conference: 1st Pan-American Congress on Computational Mechanics and XI Argentine Congress on Computational Mechanics*
- Szody, R. et al. 2017. (in hungarian) "Csípőprotézis revízióikor alkalmazott „custom made” vápakosár tervezése és készítése, három esetben alkalmazott eljárás." In *7. Hungarian Conference of Biomechanics* (Szeged, HU, okt 6-7) *Biomechanica Hungarica* 10(2): 20

AUTHOR BIOGRAPHIES

MARTIN O. DÓCZI is a Ph.D. student at the Budapest University of Technology and Economics Department of Machine and Product Design, where he studied mechanical engineering and obtained his degree in 2019. His research area is numerical biomechanics and implant development. His e-mail address is: doczi.martin@gt3.bme.hu and his Web-page can be found at <http://www.gt3.bme.hu>.

RÓBERT SZÓDY is an orthopaedic and traumatology physician. He got his degree at the Semmelweis University in 1995. He made a traumatology professional examination in 2000 and an orthopaedics professional examination in 2005. He works as a surgeon at Péterfy Hospital and Manninger Jenő National Institute of Traumatology. His e-mail address is: robert.szody@gmail.com.

PÉTER T. ZWIERCZYK is an assistant professor at Budapest University of Technology and Economics Department of Machine and Product Design where he received his M.Sc. degree and then completed his Ph.D. in mechanical engineering. His main research field is the railway wheel-rail connection. He is member of the finite element modelling (FEM) research group. His e-mail address is: z.peter@gt3.bme.hu and his web-page can be found at: <http://gt3.bme.hu>