EXPERIMENTAL METHODS USED FOR VERIFICATION OF THE COMPUTATIONAL MODEL OF SILICON RUBBER IMPLANT

Jaroslav Kult[1], Daniel Vavřík[2]

Abstract

The contribution from the field of biomechanics reports on a complex analysis of Swanson's prosthesis of interphalangeal joint and its interaction with phalangeal and metacarpal bone. The implant material (silicon rubber with extremely high ductility) is numerically modeled by using hyper-elastic element. Material properties were measured by stress-strain test with optical strain identification. To verify the FE model a comparative photoelastic measurement was done.

KEY WORDS: interphalangeal joint, computational modeling, optical identification, photoelastic measurement, silicon rubber, Swanson's implant

1 Introduction

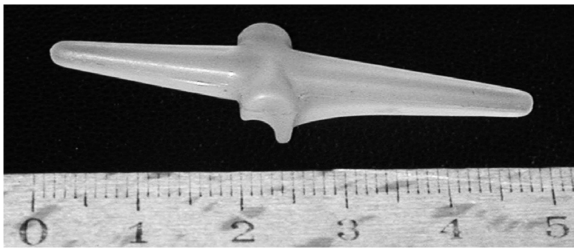

Figure 1: View of the implant

Swanson's endoprosthesis is used for replacement of all joints of human hand except distal interphalangeal (IP) joints. Using the Swanson's implant is a modern treatment method in the human hand reconstruction surgery [5], this type of implant can replace wrist, metacarpal or IP joint destroyed by rheumatic arthritis [3]. The flexible implant is made of silicon rubber and based on different principals than rigid implants. The prosthesis is a single-piece silicon rubber cast; it consists of a central transverse flexural zone and two slender conical stems in the distal and proximal direction that are placed (but not fixed) in the phalangeal bones. The stress magnitude in the bone tissue is low comparing to the bone resistance [4]. The bone has a strong influence on the implant as a means of load. The aim of the research was to create an appropriate model of the Swanson's IP joint implant

[1]Jaroslav Kult, Czech Technical University in Prague, Faculty of Transportation Sciences, Department of Mechanics and Materials, Na Florenci 25, 11000 Praha 1, Czech Republic, phone: +420-2-24 21 46 05, email: xkult@fd.cvut.cz

[2]Daniel Vavřík, Institute of Theoretical and Applied Mechanics, Academy of Sciences of the Czech Republic, Prosecká 76, 190 00 Praha 9, Czech Republic, phone: +420-2-86 88 21 21, email: vavrik@itam.cas.cz

manufactured by Czech producer Rubena Náchod and to evaluate the stress state in basic load cases recommended by surgeons.

2 Methods

2.1 FE model

The stress state analysis is based on a 3D numerical model created using FEM software ANSYS version 5.6. The implant-bone interaction was modeled by rigid contact pair. The solution with geometrical and material nonlinearities was obtained for the following load cases (recommended by orthopedists):

- flexion 15°, 30°, 45°, 60°, 75°
- extension 10°
- radioulnar flexure 10°

The hyper-elastic element HYPER158 [1] was used to describe the behavior of rubber material.

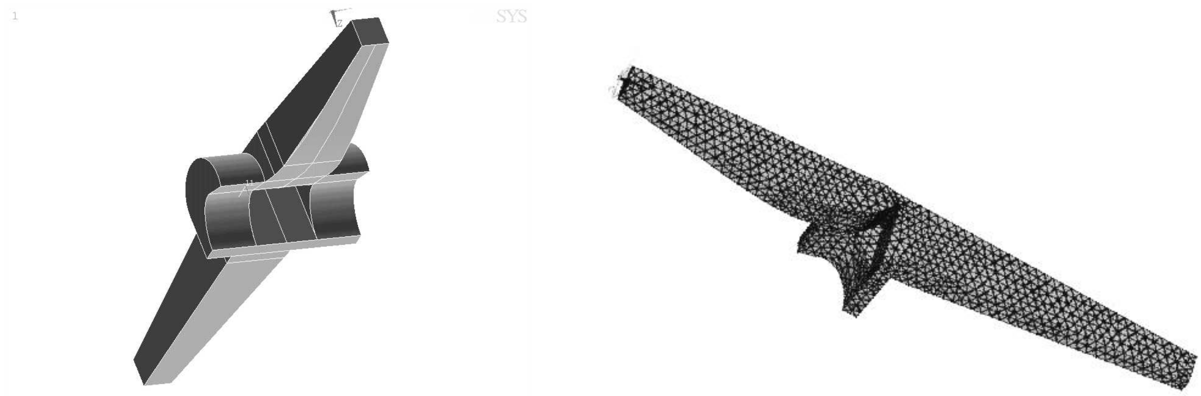

Figure 2: View of the Finite Element model

2.2 Material properties measurement

The implant is made of silicon rubber Q7-4550 Dow Corning - material with extremely high elasticity, strength and large strain. The guaranteed value of strain is 300% and tensile strength 6.0 MPa. Detailed relation between σ and ε was learned by uniaxilar stress - strain test with 3 specimens.

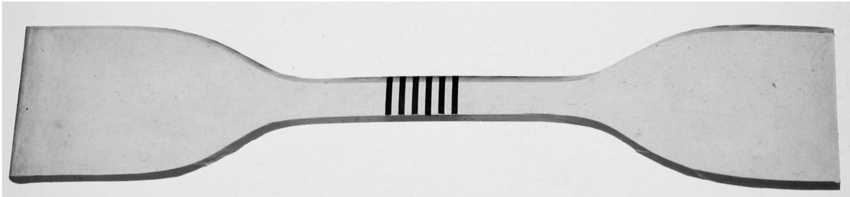

Figure 3: The specimen

The specimen geometry (fig. 3) respects the international ISO 37 standard [2]. Load with constant speed of 5 mm/min was applied by testing machine INSTRON 4301. This load can be considered as the quasi-static one (this was proved by comparative tests).

The force was measured by the testing machine and recorded on a PC in the time intervals of 5 sec. The deformation was simultaneously recorded by Black and White CCD camera with resolution of 1200 x 1000 pixels. Lens with telemetric properties (i.e. without optical error) were used. The images were processed by automated **optical identification method** (specifically **line grid method**) [6]. This method evaluates absolute and relative deformation of lines printed on the specimen surface in both longitudinal and transverse direction. The longitudinal deformation represents strain, the transverse deformation represents contraction.

The uniaxilar stress was calculated from the measured values of force, initial cross-section area and contraction. The table of data representing $\sigma - \varepsilon$ relation was used as an input to Mooney-Rivlin hyper-elastic material model [1].

2.3 Photoelastic model

In order to verify the proper way of boundary conditions (implant - bone interaction) a comparative photoelastic model was made of epoxy resin. Load was applied by loading panel simulating the implant placement in the bone. Respecting the linear and elastic nature of the photoelastic method only selected load cases were measured (flexion 15°, 30°; extension 10°; radioulnar flexure 10°).

3 Results

The photoelastic measurement shows isolines where the difference of principal stresses is constant. Similarly ANSYS can plot isolines of the stress intensity. The differences of principal stresses can be easily compared (fig. 5). Evaluation of stresses obtained from photoelastic measurement is pretty laborious and quantitative comparison wasn't done because the qualitative one showed good correspondence of the FE model with the experimental one.

The stress - strain diagram (fig. 4) and table (result of the stress - strain uniaxilar test using optical identification method) was used as input data to the numerical model.

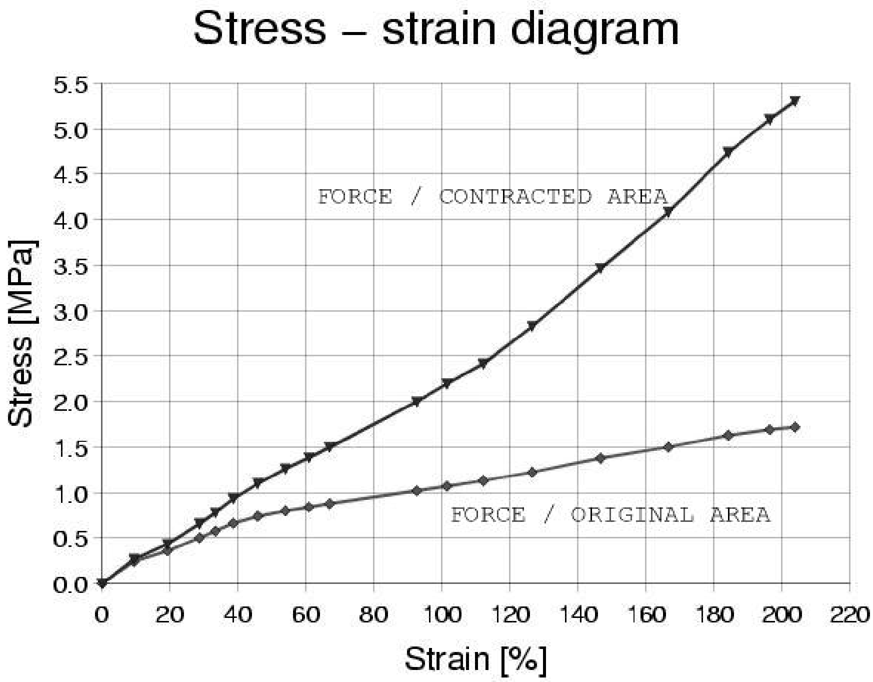

Figure 4: Stress - strain diagram of silicone rubber

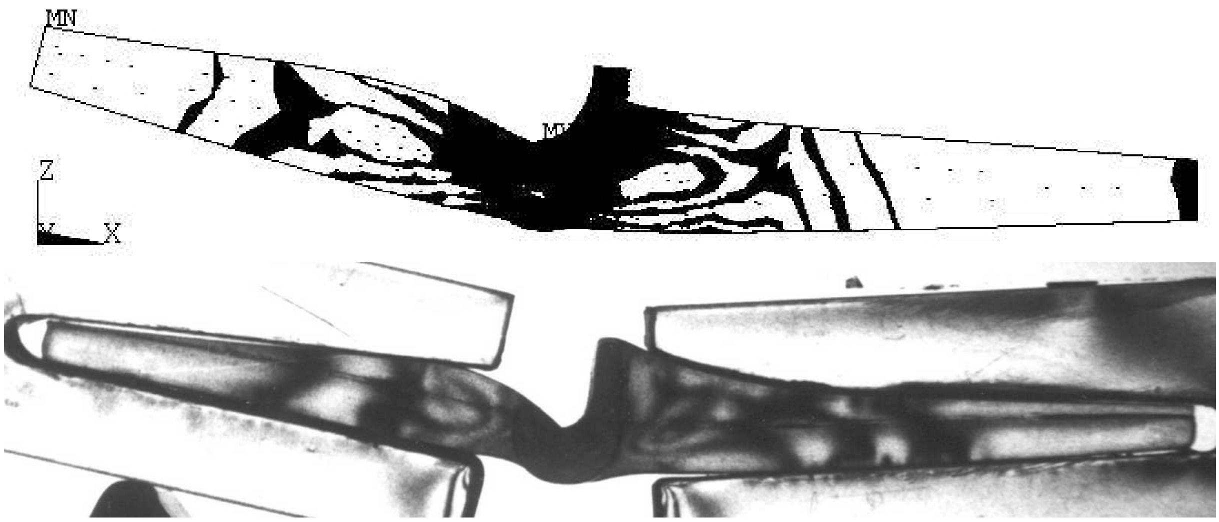

Figure 5: Comparison of principal stresses differences obtained from FEM and photoelastic model for flexion 15°

Numerical evaluation of the implant stress state was obtained from the Ansys FE model. The dangerous mode of fracture of silicon rubber is the tension failure. Therefore the values of the first principal stress σ_1 were analysed. The maximal stress value for flexion is located on the dorsal side of the implant where the stem is connected to the central flexural zone. On the contrary the maximal stress for extension is on the palmar side. The highest value σ_1=1.18 MPa results up from the load case flexion 75° . Stress values from extension and radioulnar flexure are much lower.

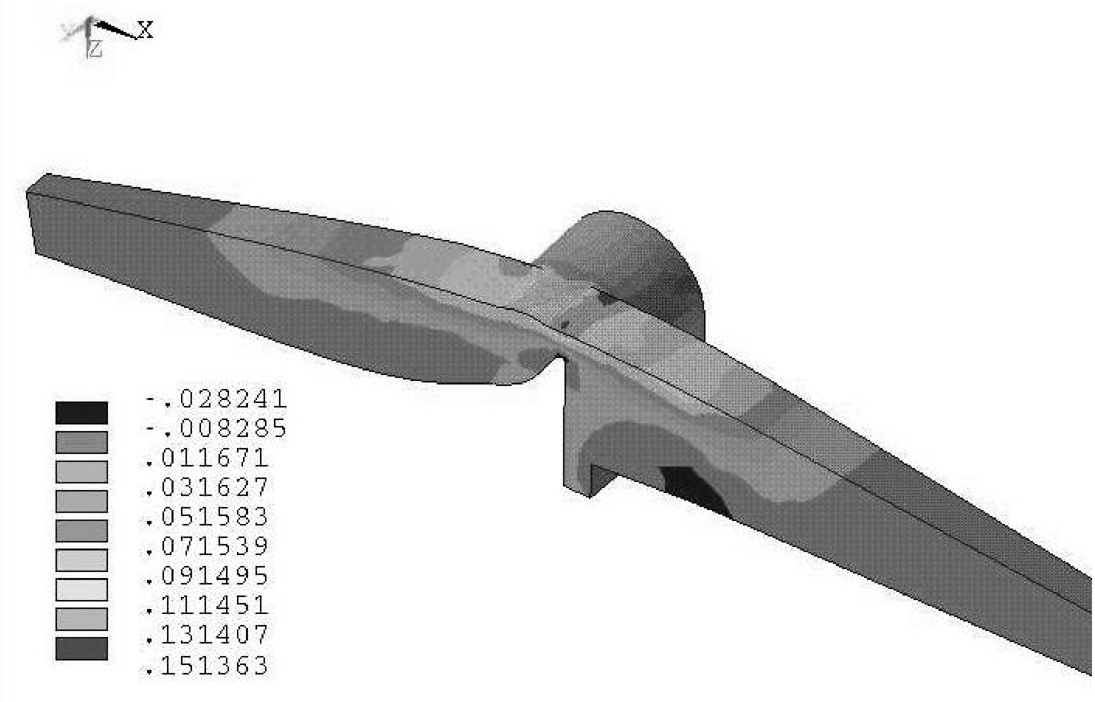

Figure 6: Stress field σ_1 for flexion 75° [MPa]

4 Discussion

The guaranteed tensile strength of the silicon rubber Q7-4550 Dow Corning is 6.0 MPa. The highest calculated value of the first principal stress for flexion 75° is 1.18 MPa. The stresses in the implant are much lower than the bearing capacity of the material.

We should also aim our attention to the place where the bone is in contact with the prosthesis. The stresses at this location are not maximal but they are still of high values. At different load cases (flexion vs. extension) there is a range of stresses varying from tension to compression. Contact with the resected bone edge can initiate a surface damage followed by a crack propagation. To avoid this we can recommend preventing sharp edges during the implant fixation.

5 Conclusion

- Optical Identification Method is a very good method for strain measurement of hyper-elastic rubber materials; using this method we can measure deformation in both directions with high accuracy
- qualitative comparison of the results from FEM analysis and photoelastic measurement shows good correspondence of the models in the range of linear deformation; the FE model can be trusted and used for analysis of other load cases
- the maximal calculated stresses for analysed load cases are much lower than the bearing capacity of the material
- the probable failure mode of the analysed implant is crack initialization caused by the contact with the bone edge; avoiding sharp bone edges or thin metal covers can eliminate this danger

References

[1] *ANSYS User's Manual for Revision 5.5*

[2] Czech standard *CSN ISO 37 Pryž z vulkanizovaných nebo termoplastických kaučuků - Stanovení tahových vlastností*

[3] Jíra J., Jírová J.: Computational modelling of stress state during hand treatment, in: *Computer Methods in Biomechanics & Biomedical Engineering - 2*, Gordon and Breach Science Publishers, 1998, edited by J. Middleton, M. L. Jones and G. N. Pande

[4] Jiroušek O.: Biomechanical Analysis of an Implant Used in Finger Reconstruction Surgery on the Basis of Data Obtained from Computer tomography, in: *Proc. Engineering mechanics 2000 - Volume III.* Svratka, 2000

[5] Pech J.: *Endoprotetika zápěstí se vztahem k revmatochirurgii*, Schola nova Comenium, 1996

[6] Vavřík D.: Experimental Determination of Constitutive Equation of Nonlinear Material on One Specimen Only, in: Proc. *Advances of Measurement Techniques and Experimental Methods in Engineering Sciences.* Bad Honnef, 2001

Acknowledgment: *The research has been sponsored by the project A2071003 of the Grant Agency of the Academy of Sciences of the Czech Republic and by research funds JD MSM 216200031 & JO MSM 212600025 of the Ministry of Education, Youth and Sports of the Czech Republic.*