Stress computations of the hip joint replacement using the finite element method

M. Čertíková, J. Tuzar, B. Sousedík, J. Novotný

Czech Technical University, Dept. of Mathematics, Prague,
Institute of Thermomechanics, Czech Academy of Sciences, Prague

Abstract. *We describe computations of stresses in the replacement of hip joint using the finite element method. The construction of the replacement consists of several parts, made of titanium and is loaded by the weight of human body during walk and tension of the screws, which are used for mounting to pelvis bone and the construction together. We explain method of mesh generation, describe modelling of physical data and results of linear elasticity computations using different loads.*

1 Introduction

The replacement of hip joint is a complicated construction made of titanium, which consists of three parts, see Figs. 1 and 2.

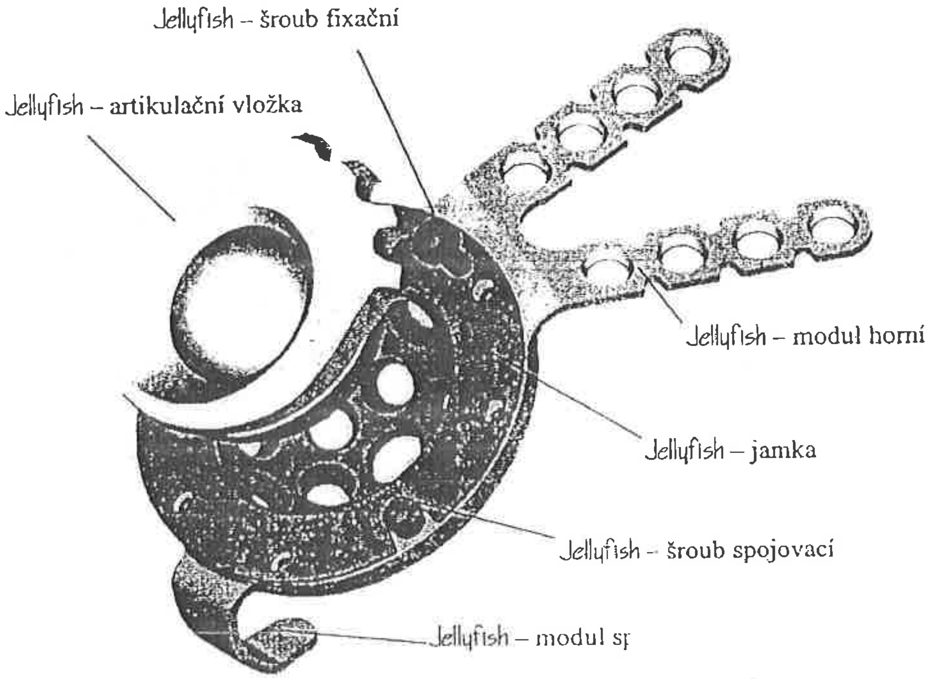

Fig. 1: *Scheme of the replacement construction*

There are cup, upper module with two holders, which contain holes for screwing to the pelvis bone, and lower module with one holder for supporting the pelvis bone. These components are mounted together using screws. Spherical enclosure of polyethylene fits in the cup. In this enclosure can move the head of the hip joint.

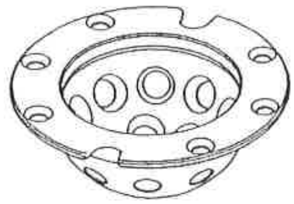

Fig. 2a: *Cup*

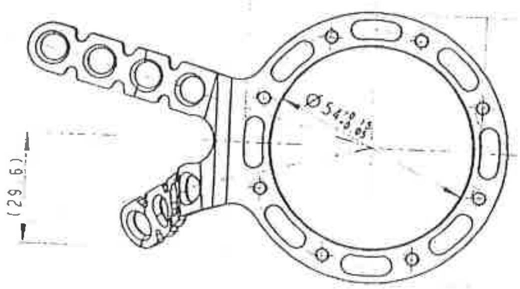

Fig. 2b: *Upper module*

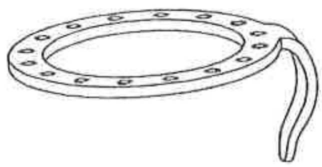

Fig. 2c: *Lower module*

The problem is durability of the construction from load during walk. Therefore we deal mainly with computation of stresses. In reality, the problem should be studied as a contact problem with time-dependent loads during walk, however, it would be too complicated. Stress analysis of the pelvis bone, to which the replacement is mounted, should be considered too.

2 Formulation of the mixed problem of linear elasticity in 3D (small displacements)

Let Ω be a bounded domain, f_i, $i = 1, 2, 3$ are densities of volumetric loads, Γ_τ and Γ_u are disjoint parts of the boundary of Ω, $\partial\Omega = \Gamma = \Gamma_\tau \cup \Gamma_u$. In **classical formulation** we search displacements $u = (u_1, u_2, u_3)$ satisfying Lame equations

$$(\lambda + \mu)\frac{\partial^2 u_j}{\partial x_i \partial x_j} + \mu \Delta u_i = f_i, \quad i = 1, 2, 3 \quad \text{in } \Omega \text{ (summation over index } j)$$

with boundary conditions

$$\tau_{ij}\nu_j = T_{0i}, \quad i = 1,2,3 \quad \text{on } \Gamma_\tau$$
$$u_i = u_{0i}, \quad i = 1,2,3 \quad \text{on } \Gamma_u$$

Here u_{0i} denotes prescribed diplacements and T_{0i} stress vector on the boundary, τ_{ij} represents Cauchy stress tensor, given by the Hook's law, λ and μ are Lame coefficients. We suppose, that they are constant in our problem.

From this formulation **weak formulation** can be derived. This represents the basis for the finite element method. Existence and uniqueness of the *weak solution* follows from Korn's inequality, see for instance [1]. Problem, discretized using the finite element method leads to the solution of a large sparse system of linear equations in the form

$$KU = F,$$

where K is the stiffness matrix, which is positive definite, F is discretized right hand side and U is the vector of nodal displacements.

In our code we used discretization by isoparametric hexahedrons with 20 nodes. For numerical integration of stiffness matrices and right hand sides we implemented Gaussian integration of order 3. Therefore we integrate polynomials of fifth order in three dimensions exactly.

3 Mesh generation

The construction of the replacement has complicated geometry, consisting of three parts. We describe in more details mesh generation of the most complicated part - the cup. First, we created inner and outer surface of one block containing one circular hole, see Fig. 3.

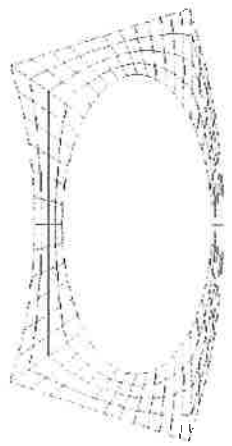

Fig. 3a: *Inner surface*

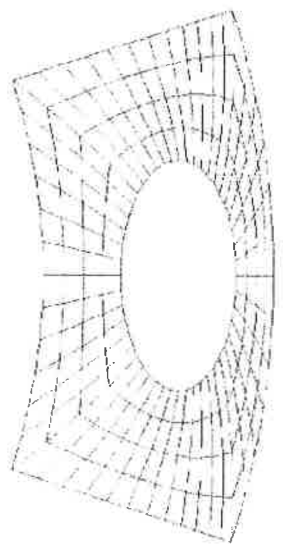

Fig. 3b: *Outer surface*

Then, the volume between these surfaces is meshed using the generator and we obtain finite element mesh of the block with one circular hole, Fig. 4.

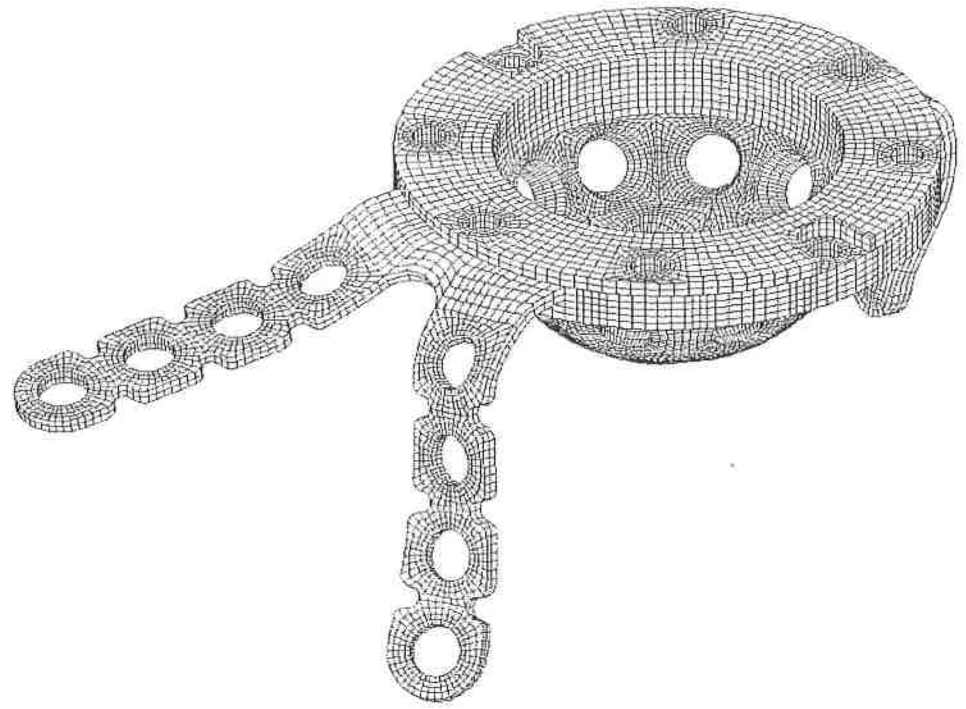

Fig. 6: *Mesh of the replacement*

Finally, all components were connected together, Fig. 6. In regions, where the components are screwed together, we needed to identify nodes on surfaces under the screwheads, Fig. 7.

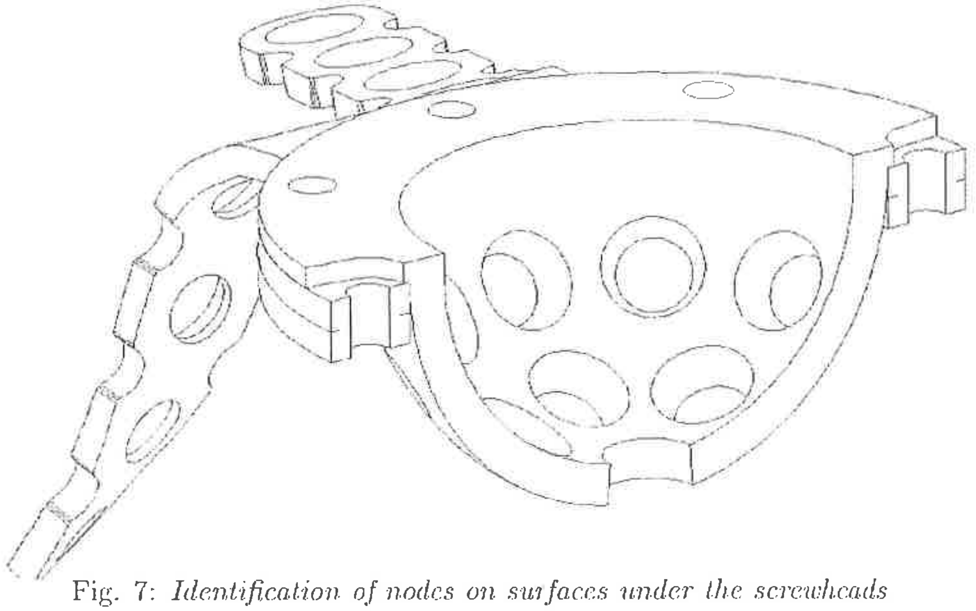

Fig. 7: *Identification of nodes on surfaces under the screwheads*

4 Physical data

Material properties of titanium are precsribed as follows: Young modulus E = 1.128 . 10^{11} Pa, Poisson ratio ν = 0.32. First, we considered only load from the weight of human body, F = 2000 N, Fig. 8. Orientation of the load force vector changes during walk, we tried to prescribe the worse orientation. Second, we added also surface loads from tightening of the screws. Total force acting on each screw was 500 N.

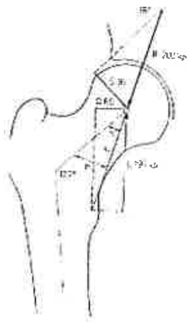

Fig. 8: *Load vector from the weight of human body*

Prescribed total load from the weigt of human body cannot be implemented in the model as nodal force. It is necessary to determine approximately its distribution on the surface of the cup. For simplicity, we considred circular region on the cup surface and cosine distribution of loads. In the centre of the spherical cap, the intensity of the load is maximal and on the boundary of the cap the intensity is zero, Fig. 9.

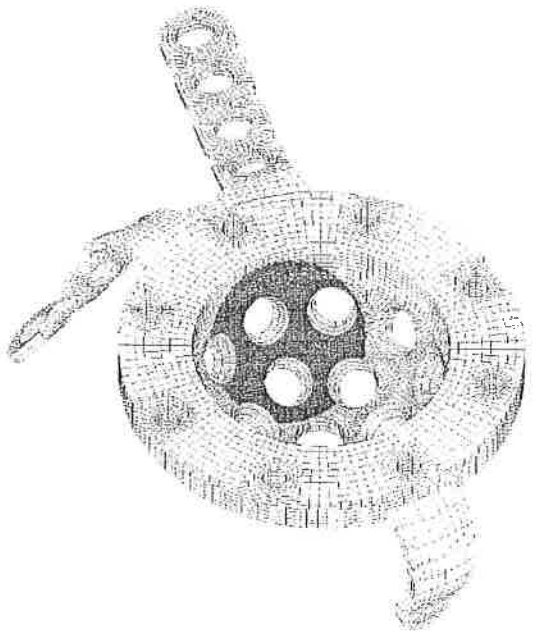

Fig. 9: *Distribution of surface loads in the cup*

Boundary condition for displacement are prescribed in the following way. The screws are placed only in last two holes of each holder of upper module. By the screws is the replacement mounted to the pelvis bone. Therefore we prescribe variables on surfaces of holders under the screwheads fixed, Fig. 10. The construction of the replacement is also supported by the pelvis bone on 2 small elliptic regions of lower module, where we consider only 1 component of displacement fixed, Fig. 11.

Fig. 10: *Fixed variables - screws*

Fig. 11: *Fixed variables - support*

5 Results of numerical computations

We considered the following variants:

1. Thickness of holders on upper module (which contain screws, mounting the construction to the pelvis bone) is 2 mm, loads only from the weight of human body, without titghtening the screws. Computation of the problem of linear elasticity lead to maximal von Mises stress $\sigma_{HMH}^{MAX} = 1500$ MPa. The highest stress is reached in the notches of the holders, Fig. 12.

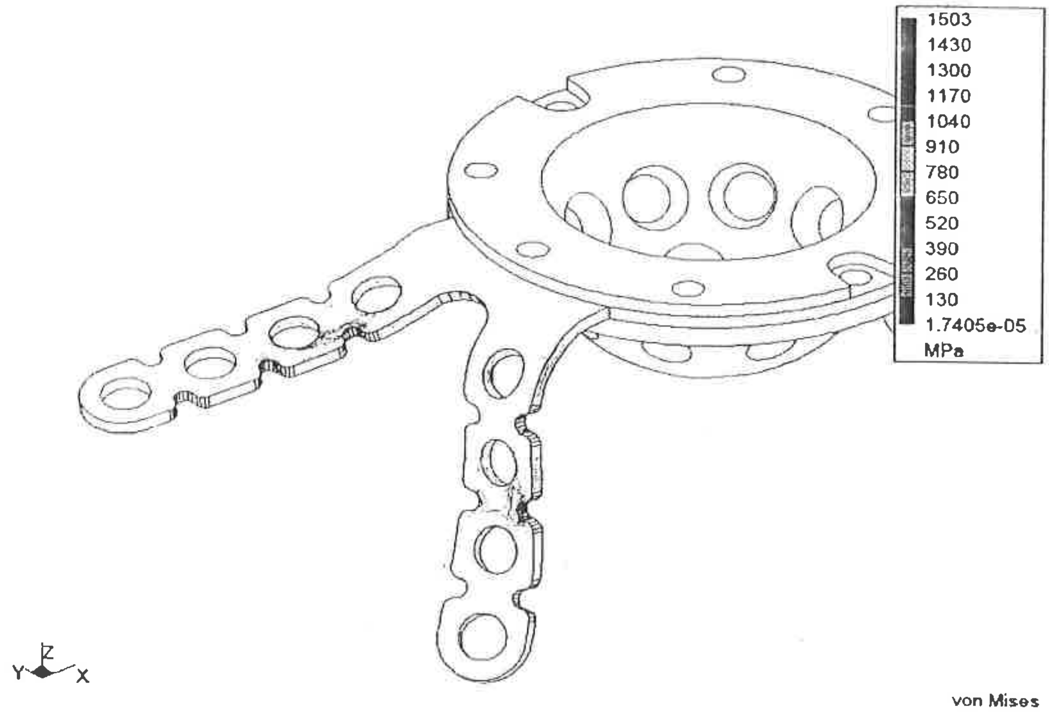

Fig. 12: *Stresses* σ_{HMH}, σ_{HMH}^{MAX} *= 1500 MPa*

The problem has 27 586 elements, 462 741 unknowns, maximum front width is 4 803. On AlphaServer ES47/1000 MHz the computation takes 9 hours of CPU time.

Because the yield point of titanium is about 800 MPa, this high stress is unacceptable and we had to modify the geometry of the construction.

2. Thickness of holders on upper module was increased to 3 mm and radiuses of the notches in the holders were increased and the notches were made smaller. Loads from the weight of human body, without titghtening the screws were considered. Therefore new mesh had to be created, Fig. 13.

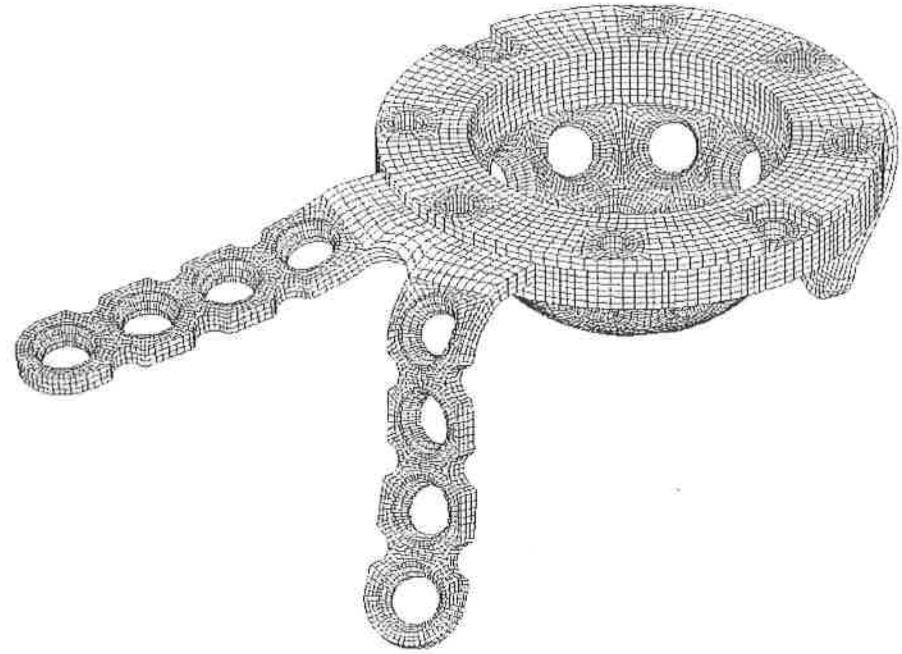

Fig. 13: *Modified mesh of the replacement*

Computation of the problem of linear elasticity lead to maximum von Mises stress only $\sigma_{HMH}^{MAX} = 540$ MPa, which satisfied the demands for the strength of the construction. Fig. 14.

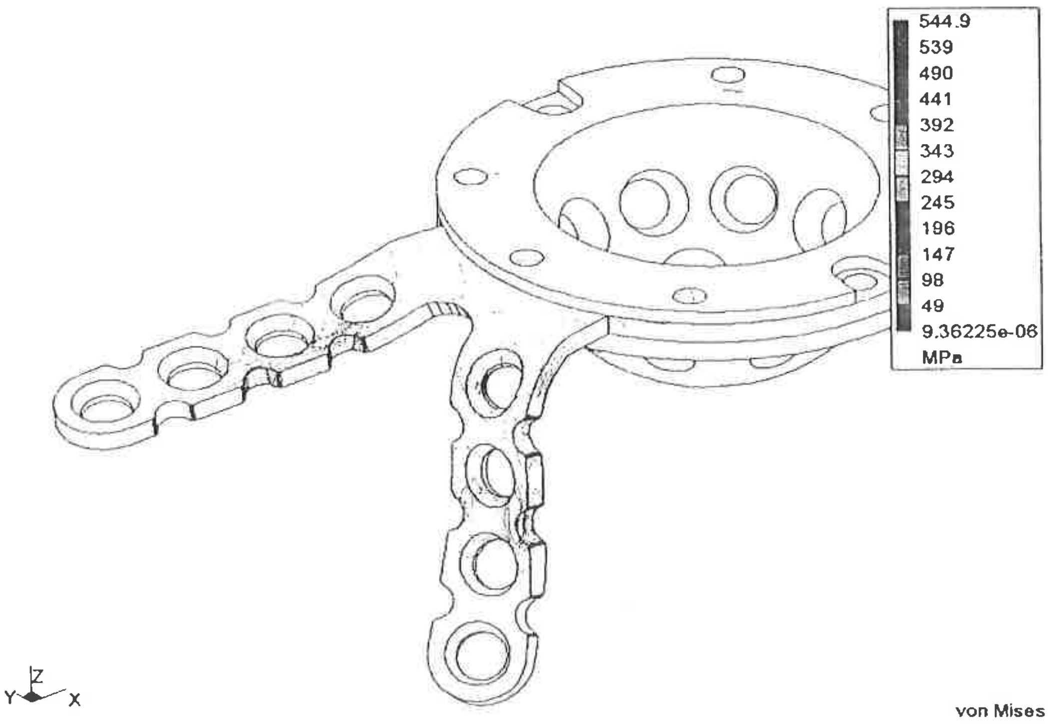

Fig. 14: *Stresses* σ_{HMH}, $\sigma_{HMH}^{MAX} = 540$ *MPa*

The problem had 33 186 elements, 544 734 unknowns, maximum front width was 8406. On AlphaServer ES47/1000 MHz the computation takes 18.5 hours of CPU time.

3. We considered same geometry as in variant 2, but we added loads from tightening of the screws. Maximal von Mises stress nearly didn't change and reached also $\sigma_{HMH}^{MAX} = 540$ MPa.

6 Conclusion

Linear elasticity computation of stresses of the replacement of hip joint lead to high von Mises stress and therefore the construction had to be modified. After doing the computations with the modified construction we succeeded in reaching much smaller stresses, which satisfy to our demands. Next step should be to incorporate stress computation of the pelvis bone together with the construction.

Acknowledgement. This work has been supported by the grant GA CR 106/05/2731.

References

[1] Nečas, J., Hlaváček, I: *Úvod do matematické teorie pružných a pružně-plastických těles.* SNTL, Praha, 1983.