THE NUMERICAL MODELING OF HUMAN PELVIC BONE WITH ARTIFICIAL ACETABULUM

Antoni John*, Piotr Orantek*, Jozef J. Telega†

* Department for Strength of Materials and Computational Mechanics
44-100 Gliwice, Konarskiego 18A, Silesian University of Technology, Poland
e-mails: ajohn@rmt4.kmt.polsl.gliwice.pl, orantek@rmt4.kmt.polsl.gliwice.pl

† Institute of Fundamental Technological Research Polish Academy of Science
00-049 Warszawa, Świętokrzyska 21, Poland
e-mail: jtelega@ippt.gov.pl

Key words: Pelvic bone, Numerical model, Artificial acetabulum, FEM, Numerical simulation.

Abstract. Bioengineering concerns many important problems apply to human body. The pelvic joint and its correct working is one of them. The pelvic bone is one of the most important supporting elements in human pelvic joint but it is liable to suffer an injury. Very often before and after operations the knowledge of the stress and strain distribution in the pelvic bone is needed. It is particular important when the THA operation is performed and the artificial acetabulum is fitted. Because the safety of patient should be taken into account there are only two possibilities: model testing and numerical calculations. Before numerical calculations the numerical model should be prepared. In the paper the numerical model is prepared on the ground of the geometrical data from 3D scanning or CT. For checking the numerical model a simple benchmark was proposed, with force acting in acetabulum by ceramic ball. The results for selected load cases are presented.
1 INTRODUCTION

Pelvic bone is an element of bone system, which is liable to suffer an injury (break, crush). When it needs surgical intervention surgeons want to know what will change in pelvic joint (stress and strain distributions) after operations. It is very difficult or impossible to measure the strain and stress „in vivo” because the safety of patient should be taken into account. There are only two possibilities: model testing and numerical calculations [1,2,3]. Complex geometry and material structure of bone tissue as well as its state of load or physiological reactions complexity, cause huge variety of acceptable assumption in numerical models.

There are two main problems during preparing numerical model. The first problem is – how to translate geometrical features from real existing human pelvic bone to numerical model and the second – how to model the boundary conditions and load. The former investigations, base on geometrical data preparing manually from clinical specimen. Currently, geometrical data are assumed on the base of outside measurement (scanning) using coordinate measuring machine [4, 5]. A numerical routine (numerical code) was built to translate the geometrical data (the set of coordinate points) to Patran/Nastran code. From measurement we obtain the data on outside surface of pelvic bone only. When the layer structure of bone tissues is taking into account there is necessary to use the knowledge of bone tissue density from X-ray photo or CT. Using own numerical code the inner surface in numerical model is implemented (between cortical bone tissue and trabecular bone tissue).

It can be done automatically or manually. Next, the finite element meshing is done. Solid elements illustrating 3D stresses distribution were used for modeling. Separate solid elements layers are modeled by cortical and trabecular bone. At present homogeneous elastic properties within a certain group of tissue as well as continuum are assumed. One or more layer of elements, depending on model’s bone tissue’s thickness, models cortical and trabecular bone. Next, when loads and boundary conditions are assumed, the strain and stress distribution can be calculated using Nastran or MARC code. Boundary conditions and loads are implemented manually ([6]).

When the THA is performed and artificial acetabulum is fitted it is important to know the stress and strain distribution near acetabulum after surgeon’s intervention. Numerical simulation is one of the easier and cheaper way to determine the stress and strain distribution in complex objects but only the numerical models verifying in experiment can be apply [7].

2 NUMERICAL MODELLING

In the aim to creating an artificial acetabulum a few procedures were done. All procedures were written in the C++ language. The procedures create the flange (width), the spherical cap (radius), and the bolts of artificial acetabulum (2 angles in spherical coordinates, width, height). On the basis of the above parameters the whole geometry of the structure is created (Fig. 1). Next on the ground of the geometry, the finite element model is created and put into the bone finite element model (Fig. 2). The surfaces are modeled using triangular elements and the solids are modeled using tetrahedral elements.
There is possible to model cemented and cement less acetabulum, with contact elements and without. At present MSC.Patran/Nastran and MSC.Patran/Marc systems are applied.
Here, boundary conditions are given in two areas: in contact area with sacral bone and in pubic symphysis. Stress and strain distributions of human pelvic bone are a result of external load coming from upper body part’s weight and muscles forces. Referring to earlier works, the model takes up 23 muscle tensions influencing through pelvic bone and tendons on insertions’ surfaces ([3,5,6]).

For checking the influence of the forces acting in acetabulum on the stress and strain distribution in the surroundings of the artificial acetabulum a simple benchmark was proposed, with force acting in acetabulum by ceramic ball. Fig. 3 shows the components of model with artificial acetabulum (separated for test) and Fig. 4 shows the model of pelvic bone with fitted artificial acetabulum.

![Figure 3: The components of model with artificial acetabulum.](image)

### 3 NUMERICAL CALCULATIONS

The calculations were performed for 5 schemes of acting forces (Fig. 5). The scheme 1 represents force acting perpendicularly to base plane of artificial acetabulum. The schemes 2 – 5 represent forces acting at an angle of 30° to force from scheme 1, in given direction. All these forces acting in the center of ceramic ball, inwards of acetabulum. For every load cases assumed total value of acting force equals to 400N. The highest effort of constituent elements was obtained for the 3rd scheme of load (the right force). The stress distribution shows the Figures 6 – 9 for the artificial acetabulum, cement layer, trabecular bone and cortical bone, respectively (the down force, scheme 5).
Figure 4: Model with fitted artificial acetabulum.

Figure 5: The scheme of acting forces.

Figure 6: The reduced stress distribution in artificial acetabulum (scheme 5).

Figure 7: The reduced stress distribution in cement layer (scheme 5).

Figure 8: The reduced stress distribution in trabecular bone (scheme 5).

Figure 9: The reduced stress distribution in cortical bone (scheme 5).
Figures 10 – 13 show the strain distribution for the 3rd scheme of load (right force).

Figure 10: The reduced strain distribution in artificial acetabulum (scheme 3).

Figure 11: The reduced strain distribution in cement layer (scheme 3).

Figure 12: The reduced strain distribution in trabecular bone (scheme 5).

Figure 13: The reduced strain distribution in cortical bone (scheme 5).

The highest differences appear for the reduced stresses (von Mises) and reach 40%. The highest value of reduced stresses appears in artificial acetabulum and polyethylene cup. The region of the stress concentration is characteristic for the given load case. The maximal and minimal values of the reduced stress, the reduced strain and the resultant displacement for each part of numerical model (for the 3rd scheme of load) are presented in Table 1.

Presented results were obtained for model without friction, adhesion and wear. In preparing is a model with every features mention above. Theoretical ground for that investigations is assumed from the literature [8,9,10,11].
<table>
<thead>
<tr>
<th>Reduced stresses [MPa]</th>
<th>Min.</th>
<th>2.77-001</th>
<th>8.99-002</th>
<th>8.46-002</th>
<th>3.48-002</th>
<th>0.00+000</th>
<th>0.00+000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max</td>
<td>1.83+000</td>
<td>6.36+000</td>
<td>1.05+001</td>
<td>5.40+000</td>
<td>5.00+000</td>
<td>6.32+000</td>
<td></td>
</tr>
<tr>
<td>Reduced strains</td>
<td>Min.</td>
<td>1.20-012</td>
<td>1.99-011</td>
<td>1.02-012</td>
<td>4.07-012</td>
<td>0.00+000</td>
<td>0.00+000</td>
</tr>
<tr>
<td>Max</td>
<td>4.17-010</td>
<td>5.50-010</td>
<td>3.27-010</td>
<td>9.21-010</td>
<td>9.21-010</td>
<td>7.01-010</td>
<td></td>
</tr>
<tr>
<td>Resultant displacement [mm]</td>
<td>Min.</td>
<td>5.12-009</td>
<td>3.07-009</td>
<td>2.54-009</td>
<td>2.49-009</td>
<td>0.00+000</td>
<td>0.00+000</td>
</tr>
<tr>
<td>Max</td>
<td>7.45-009</td>
<td>7.34-009</td>
<td>4.07-009</td>
<td>4.06-009</td>
<td>4.01-009</td>
<td>3.90-009</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Selected results for the most effort load case (5).

4 CONCLUSIONS

- The numerical models, prepared on the ground of 3D scanning and CT were used to create model with artificial acetabulum.
- The numerical models applied to evaluation results of surgical intervention should be verify in experiment.
- The creation of the models after THA needs additional subroutines aided that process.
- The boundary conditions are results from the correct pelvic joint and changes from surgical intervention.
- Obtained results can be useful to planning and quality assessment of THA. The surgeons can observe which states of load are dangerous for the patients.
- When the contact with friction, adhesion and wear will be taken into account, it seems that it is closer to real existing conditions.

ACKNOWLEDGEMENT

The work was done as a part of research project 4T11F00325 sponsored by Polish Ministry of Science and Information Technology.

REFERENCES


