282. The numerical modeling of surgical intervention in human pelvic bone

Antoni JOHN  
Silesian University of Technology, Faculty of Mechanical Engineering,  
Department for Strength of Material and Computational Mechanic,  
Konarskiego 18a, 44-100 Gliwice, phone +48 322372439,  
E-mail: antoni.john@polsl.pl

(Received 14 March 2007; accepted 15 June 2007)

Abstract. The numerical modeling makes it possible to prepare FE model of human pelvic bone after reconstruction. It is particular important when the THA operation is performed and the artificial acetabulum is fitted. Very often before and after operations the knowledge of the stress and strain distribution in the pelvic bone is needed. 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 bench-mark was proposed, with force acting on acetabulum by steel ball.

Keywords: finite element method, pelvic bone, artificial acetabulum, stress distribution.

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. 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.

Before numerical analysis of strain and stress distribution in human pelvic bone the numerical model of analyzed structure should be prepared. It is important step in numerical analysis because obtained results depend on it. Up to the present, in the most of examples, the creation of numerical model was done in simple but time-consuming way [12, 15, 16, 17, 18, 19, 22, 24, 26, 27, 28, 29, 30, 31, 50, 53].

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. A numerical routine (numerical code) was built to translate the geometrical data (the set of coordinate points) to Patran/Nastran code [32-43,56,59]. 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.

In the paper the following problems are discussed: 1. numerical model; 2. boundary conditions; 3. numerical analysis for different assumptions; 4. experimental verification (using ESPI); 5. numerical model with artificial acetabulum.

Commonly used and effective method for therapy of advanced degenerative changes of a hip joint is a mechanical reconstruction of joint destroyed co-operating surfaces by implantation of endoprosthesis. The implanted artificial joint is some foreign element in human body, which has worse mechanical properties in compare with natural one and never replace it [1, 3, 5-11, 15, 18, 22, 23]. In spite of all the early results of treatment are very good [2, 9, 15, 23]. After operation of a hip joint decrease some pain troubles and improve joint functions.
Numerical model

Before numerical analysis of strain and stress distribution in human pelvic bone the numerical model of analyzed structure should be prepared. It is important step in numerical analysis because obtained results depend on it. At the first step geometrical data should be taken into account and geometrical model is prepared. In the second step the boundary conditions are assumed [12,15,17, 23,24,27,28,30,50,51]. Next we put loads and assumed material properties [1-5,9,11,13,14,16-21, 44-49,55,57,58].

Up to the present, in the most of examples, the creation of numerical model was done in simple but time-consuming way. The coordinates of set of points from measurement were prescribed manually in numerical code for using FE program. When the time of creation of geometrical model can be reduced the total time of numerical analysis can be reduced too because the creation of geometrical model is the most time-consuming step. In the paper the numerical routine, translating data from coordinate measuring machine to Patran code is presented. From measurement we obtain the set of points on outside surface of pelvic bone. As an input data, there is assumed a file *.igs (AUTOCAD format) from coordinate measuring machine. Translation from *.igs to Patran/Nastran code is done in few steps. At first an *.igs file is transformed to separate outside loops of points for each scanning level. An example of loops of points for different scanning level for human pelvic bone is shown in Fig.1. In the next step the inner surface, between cortical bone tissue and trabecular bone tissue is created for each level of scanning the set of points is generated. The schema of generation of inner loop of points and obtained result is presented in Fig. 2.

On the ground of thickness in normal direction, a point is moving in normal inner direction but minimal value of translation is assumed. It depends on real thickness value of cortical bone tissue for given kind of bone. This parameter can be changed arbitrary in program. To obtain more accurate results, the smoothing algorithm is applied. If the thickness of bone tissue in cross-section is smaller then assumed, double inner loop of points is created. In the last step the geometrical data is transformed to the set of commands in Patran/Nastran code. Output data creates a Session file in Patran code.

Boundary conditions

Stress and strain distribution of human pelvic bone is 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 (Table 1). Muscle forces are depicted in the numerical model as loads spread out on nods on insertions’ surfaces. The load slants to surface of pelvic bone under angle determined by directive cosines of muscle tensions effect line. Muscle tensions load does not take components caused by passive fiber stretch into consideration.

Table 1. Maximum values of active muscle forces, muscle tensions interactions on pelvic bone

<table>
<thead>
<tr>
<th>Muscle</th>
<th>Direct cosines</th>
<th>Max force F[N]</th>
<th>Components of muscle forces Fx[N] Fy[N] Fz[N]</th>
<th>L</th>
</tr>
</thead>
<tbody>
<tr>
<td>RF</td>
<td>0.999 -0.018 -0.999</td>
<td>852</td>
<td>0.00</td>
<td>-15.2</td>
</tr>
<tr>
<td>S</td>
<td>0.1826 0.1716 -0.968</td>
<td>148</td>
<td>27</td>
<td>25.4</td>
</tr>
<tr>
<td>IP-1</td>
<td>0.031 0.1972 -0.979</td>
<td>503</td>
<td>-16</td>
<td>99.2</td>
</tr>
<tr>
<td>IP-2</td>
<td>-0.738 -0.039 -0.672</td>
<td>503</td>
<td>-371</td>
<td>-19.9</td>
</tr>
<tr>
<td>IP-3</td>
<td>-0.016 -0.145 -0.989</td>
<td>503</td>
<td>-8.2</td>
<td>-73.2</td>
</tr>
<tr>
<td>IP-4</td>
<td>0.7129 0.0382 0.7002</td>
<td>503</td>
<td>358.6</td>
<td>19.2</td>
</tr>
<tr>
<td>GMu</td>
<td>0.2006 -0.449 -0.868</td>
<td>2339</td>
<td>-490.2</td>
<td>-203.1</td>
</tr>
<tr>
<td>ST</td>
<td>0.0263 0.0527 -0.998</td>
<td>236</td>
<td>5.9</td>
<td>11.9</td>
</tr>
<tr>
<td>SM</td>
<td>0.0262 -0.176 -0.984</td>
<td>159</td>
<td>-131.2</td>
<td>-73.4</td>
</tr>
<tr>
<td>BCL</td>
<td>0.0101 -0.216 -0.971</td>
<td>155</td>
<td>-11.1</td>
<td>-35.8</td>
</tr>
<tr>
<td>ADM</td>
<td>0.0866 -0.383 -0.921</td>
<td>1771</td>
<td>-117</td>
<td>-697.9</td>
</tr>
<tr>
<td>ADL</td>
<td>0.433 -0.455 -0.777</td>
<td>593</td>
<td>-257</td>
<td>269.8</td>
</tr>
<tr>
<td>ADB</td>
<td>0.650 -0.643 -0.467</td>
<td>452</td>
<td>-273</td>
<td>-291</td>
</tr>
<tr>
<td>PC</td>
<td>0.666 -0.572 -0.478</td>
<td>188</td>
<td>-125</td>
<td>-107.6</td>
</tr>
<tr>
<td>GMd-1</td>
<td>0.505 0.1466 -0.856</td>
<td>425</td>
<td>-214.2</td>
<td>-63.1</td>
</tr>
<tr>
<td>GMd-2</td>
<td>0.095 0.1781 -0.979</td>
<td>425</td>
<td>-40.8</td>
<td>75.7</td>
</tr>
<tr>
<td>GMd-3</td>
<td>0.0895 -0.096 -0.991</td>
<td>425</td>
<td>38</td>
<td>41</td>
</tr>
<tr>
<td>GMu-1</td>
<td>0.392 0 -0.919</td>
<td>249</td>
<td>-97.8</td>
<td>0</td>
</tr>
<tr>
<td>GMu-2</td>
<td>0.119 0.1535 -0.980</td>
<td>249</td>
<td>-29.7</td>
<td>38.2</td>
</tr>
<tr>
<td>GMu-3</td>
<td>0.2436 -0.470 -0.848</td>
<td>249</td>
<td>60.7</td>
<td>-117.1</td>
</tr>
<tr>
<td>TFL</td>
<td>0.0833 0.0263 -0.996</td>
<td>286</td>
<td>-23.8</td>
<td>7.5</td>
</tr>
</tbody>
</table>


102 © Vibromechanika. Journal of Vibroengineering. 2007 April/June, Volume 9, Issue 2, ISSN 1392-8716
There is also an important question: how to model the boundary conditions in pelvic bone? It causes the next questions: How to model the contact with others element of bone system? What we know about the stiffness of support? How to model the load? Few models can be taken into account.

It is possible to model boundary conditions in acetabulum using axial elements (in radial co-ordinate). The first ends of rods connect with nodes on outer surface of finite element in acetabulum and second ends are fixed in center of acetabulum curvature. In contact area with sacral bone boundary conditions are given using axial elements in two co-ordinates, respectively. In pubic symphysis boundary conditions are given in symmetry plane as restraints in selected co-ordinates (selected components in nodes) or by using axial elements in two co-ordinates. Here, boundary conditions are given in two area: in contact area with sacral bone and in pubic symphysis. For checking the influence between material coefficients and stress distribution the boundary conditions are given as restraints in selected co-ordinates (selected components in nodes) or by using axial elements in two co-ordinates.

### Numerical calculations

Numerical results are obtained for selected models of human pelvic bone. Displacement and stress distribution changing when different material properties are applied in selected regions for cortical and trabecular bone tissue. The stress distribution depend on load case, too. There is assumed that Young modulus is changed from 100 MPa to 200 MPa for trabecular bone tissue and from 10 GPa to 20 GPa for cortical bone tissue. Poisson’s ratio is changed from 0.3 to 0.4 and from 0.25 to 0.35 for trabecular and cortical bone tissue, respectively. For model 1 it is assumed the load acting on acetabulum surface (total 2 kN). Model 2 is loaded by concentrated force (200 N – 400 N). Model of load assumed here coming from real existing conditions in measurement station.

The examples of reduced stress and principal stress (major) distribution for model 1 show Fig. 3. The strain distribution and resultant displacement are presented in Fig. 4. Here, it is very important to apply the same boundary conditions and load in both: experimental testing and numerical calculation. Obtained results depend on material coefficient, so correct selection is needed. Results for displacement should be analysed and compare in the same coordinate frame only. Obtained results show that material coefficient have an effect on displacement (difference about 100%) and have a little bit effect on stress (difference about 30%). These results can be suitable in selection of material coefficient for checking experimental results from ESPI. When the ESPI method is applied the results, like displacement and strain, can be obtained without using material coefficient. Of course, when stresses are calculated, material coefficient are necessary. Quite different situation is during numerical simulation – the knowledge of the material coefficient is necessary from the beginning. Only geometrical model can be prepared without using material coefficient, but all calculations use them. Correct selection of material coefficient is very important step during numerical calculation.

### Experimental verification of numerical models

Pelvic bone is an important part of human bone system. For the sake of its function and work conditions, it is liable to suffer an injury. It is very difficult to measure the strain and stress distribution “in vivo” because the safety of patient should be taken into account. There are only two possibilities: experimental testing and numerical calculations. In both, experimental testing and numerical calculations it is necessary to simulate natural and pathological conditions or surgical intervention. Advanced model requires high fidelity of geometry and boundary conditions. Here, experimental testing and numerical analysis are performed. Two different methods have been used and next the results are comparing to decreasing probability of mistake (incorrect boundary conditions, incorrect finite element mashing, friction, etc.). Advantage of empirical research is possibility of avoid or restrict muscles, tendons, and ligament effect. It gives an opportunity of concentration on selected factor.

Here, in numerical models, simple boundary conditions (the same as in experiment) are assumed. The pelvic bone is restrained in two regions: on pelvic plate and near pubic symphysis, where the screws are mounted (in measurement station). A force acting in acetabulum in the same direction as in experiment. The measurement station and numerical...
model are presented in Figures 5 and 6. The way of optimal modeling is found.

Results of surgical intervention and reconstruction of damaged join are taking into account, too. On the ground of strain estimation, for given boundary conditions, comparison of obtained results is done.

Experimental verification is done using Electronic Speckle Pattern Interferometry – ESPI. It is advanced optical method, which enable to measure displacement and next account strain and stress on the outer surface of testing body. The experimental investigations are done in Institute of Machine Design and Operations of Wroclaw University of Technology. Three components of displacement are measure, and next resultant displacement is account.

There is a test of implementation real existing boundary conditions (on measurement station) to numerical model of human pelvic bone. The examples of the numerical results are presented in Figure 7. Figure 7 shows the resultant displacement for given load (one point acting force 200N), and Y components of displacement for the same load. When we compare the results it appears that we obtained very close distribution of displacement with a little difference in displacement value. It is necessary to check boundary conditions and load, and assumed material coefficient too.

The numerical modeling of human pelvic bone after surgical intervention

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. Here, the numerical model is prepared on the ground of the geometrical data from 3D scanning or CT.

When the THA is performed and artificial acetabulum is fitted data from 3D scanning or CT. For checking the numerical model a simple benchmark was proposed, with force acting in acetabulum by ceramic or steel ball. The results for selected load cases are presented.

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. 8). Next on the ground of the geometry, the finite element model is created and put into the bone finite element model (Fig. 9). The surfaces are modeled using triangular elements and the solids are modeled using tetrahedral elements. The model consists of two main groups. The first is artificial part and the second group is a biological part. In the artificial part there are: (i) – steel ball, (ii) – artificial acetabulum and (iii) – cement layer. All parts in this group are created in
the use of computer program (C++). The model depends on few parameters: size of the steel ball, and widths layers of the acetabulum and cement.

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 centre of 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). Here, the results for the 1 scheme with contact between ball and acetabulum are presented (Fig. 11-13). The reduced stresses (von Mises) increase in trabecular bone to 0.15 MPa and in cortical bone to 2 MPa.

In the biological part there are two kinds of bones: (i) trabecular bone and (ii) cortical bone. The preparation of this part is more difficult. First the position of inclusion the cement should be selected. Next, some fragment of surface should be deleted (in this stage model is as the surface). In the next step (the most time consuming step) the coupling of the cut surface (the edges) with the cements edge must be created. In the last step on the basis on the surfaces, the finite elements are generated.

Each elementary group has isolated nodes and it allows to analyse the contact problem. To perform a simple bench-mark the part of pelvic bone with artificial acetabulum is isolated. The boundary conditions are assumed on cutting planes. A force is acting in the centre of ball.

The calculations were performed for 5 schemes of acting forces (Fig. 10.). The scheme 1 represents force acting perpendicularly to base plane of artificial acetabulum.
The numerical model with bone wedge for Salter osteotomy shows Fig. 14.

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 scanning and CT were used to create model with artificial acetabulum.
- 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.

References


