ENGINEERING TRANSACTIONS • Engng. Trans. • 51, 2-3, 215–226, 2003
Polish Academy of Sciences • Institute of Fundamental Technological Research
10.24423/engtrans.487.2003

# COMPUTER AIDED MODELLING OF HUMAN PELVIC BONE

A. John, P. Orantek

**Silesian University of Technology**
Konarskiego 18 A, 44–100 Gliwice Poland,
e-mail:ajohn,orantek@rmt4.kmt.polsl.gliwice.pl

Numerical modelling of human pelvic bone makes it possible to determine the stress and strain distribution in bone tissue. Before numerical analysis, the geometrical model of analyzed structure should be prepared. It is an important step in numerical analysis because the obtained results depend on it. Up to the present, in the most of examples, the creation of geometrical model was done in a simple but time-consuming way. In the paper the numerical routine, translating data from the coordinate measuring machine to the Patran code is presented. At the first step, the geometrical model with the layer structure of bone tissue is created. The geometrical data is the basis to create an FE mesh. After meshing, the boundary conditions and load should be assumed. When the load and boundary conditions are known, the strain and stress distribution can be calculated using the Nastran or Advanced FEA code. The presented program enables to reduce the time of creation of numerical model.

## 1. Introduction

Pelvic bone is an element of the bone system, which is liable to suffer an injury (break, crush). When it needs surgical intervention, the surgeons want to know what will change in the pelvic joint (stress and strain distributions) after operations. It is very difficult or impossible to measure the strain and stress *in vivo* because safety of the patient should be taken into account. There are only two possibilities: model testing and numerical calculations [1, 10]. Complex geometry and material structure of bone tissue as well as its state of load or physiological reactions complexity, cause huge variety of acceptable assumptions in numerical models.

There are two main problems in preparing the numerical model. The first problem is – how to translate geometrical features from the really existing human pelvic bone to numerical model, and the second – how to model the boundary conditions and load. The former investigations are based on geometrical data prepared manually from the clinical specimen. Currently, the geometrical data are assumed on the basis of external measurement (scanning) using the coordinate measuring machine ([8, 9]). A numerical routine (numerical code) was built to translate the geometrical data (the set of coordinate points) to the Patran/Nastran code. From the measurements we obtain the data on the external surface of the pelvic bone only. When the layer structure of bone tissues is taken into account, it is necessary to use the knowledge of bone tissue density from the X-ray photo or CT. Using an in-house numerical code, the inner surface in numerical model is created (between the cortical bone tissue and the trabecular bone tissue). It can be done automatically or manually. Next, the finite element meshing is done. 3D solid elements were used for meshing. Separate solid elements layers are modelled by cortical and trabecular bone. At present, the homogeneous elastic properties within a certain group of tissues are assumed. Cortical and trabecular bones are modelled by one or more layers of elements, depending on model's bone tissues thickness. Next, when loads and boundary conditions are assumed, the strain and stress distribution can be calculated using the Nastran or Advanced FEA code. Boundary conditions and loads are implemented manually.

## 2. Geometrical model

Before numerical analysis of strain and stress distribution in a human pelvic bone, the numerical model of analyzed structure should be prepared. It is an important step in numerical analysis because the obtained results depend on it. At the first step, the geometrical data should be taken into account and the geometrical model is prepared. In the second step, the boundary conditions are assumed. Next we introduce the loads and the assumed material properties.

Up to the present, in the most of examples, the creation of numerical model was done in a simple but time-consuming way. The coordinates of the set of points from measurements were prescribed manually in a numerical code for using FE program [2–7]. When the time of creation of the 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 the coordinate measuring machine to the Patran code is presented. From measurement we obtain the set of points on external surface of pelvic bone. As an input data, there is assumed a file *.igs (AUTOCAD format) from the coordinate measuring machine. Translation from *.igs to the Patran/Nastran code is done in a few steps. At first, an *.igs file is transformed to separate the external loops of points for each scanning level. An example of loops of points for different scanning level for a human pelvic bone is shown in Fig. 1.

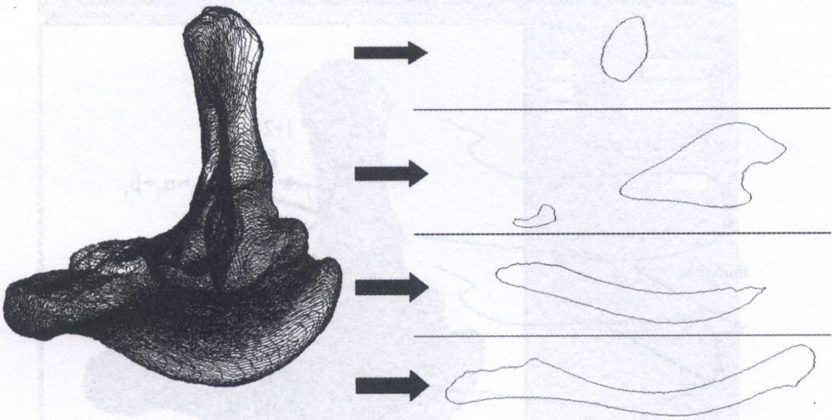

FIG. 1. Outside loops of points for different scanning level for human pelvic bone.

In the next step, the inner surface, located between the cortical bone tissue and the trabecular bone tissue is created. For each level of scanning the set of points is generated. The schema of generation of inner loops of points and the obtained results are presented in Fig. 2. On the grounds of thickness in normal direction $\Phi_i$, a point is moved in normal inner direction, but minimal value of translation is assumed. It depends on real thickness value of the cortical bone tissue for the given kind of bone. This parameter can be changed arbitrarily in the program. To obtain more accurate results, the smoothing algorithm is applied. If the thickness of bone tissue in cross-section is smaller than assumed, double inner loop of points is created (Fig. 3). There are also possibilities to put identical constant value of cortical bone tissue in the whole model. When the loop of points surface is parallel to the boundary surface, inner loop of points is not created. Notations assumed in Fig. 2:

| | |
|---|---|
| $i$ | – number of the given point, |
| $\Phi_i$ | – normal vector at point $i$, |
| $\varphi$ | – coefficient from the interval (0; 0.5), |
| $\alpha_i$, $\beta_i$ | – normal vectors of adjoining elements of point $i$, |
| thickness | – distance between the given point and the boundary, measured in normal inner direction, |
| distance | – value of translation of the given point in normal inner direction. |

In the last step, the geometrical data are transformed to the set of commands in the Patran/Nastran code. Output data creates a Session file in the Patran code. The geometrical model is shown in Fig. 4.

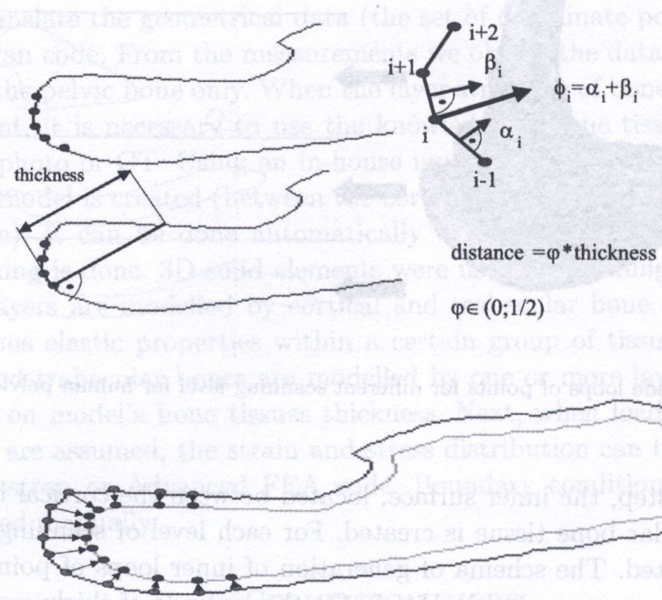


FIG. 2. The schema of generation of inner loop of a point.

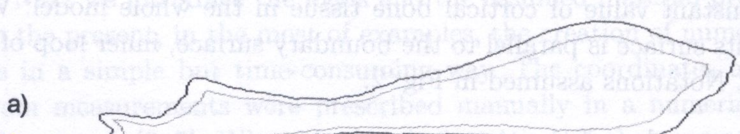


FIG. 3. Inner loop of points: a) single, b) double.

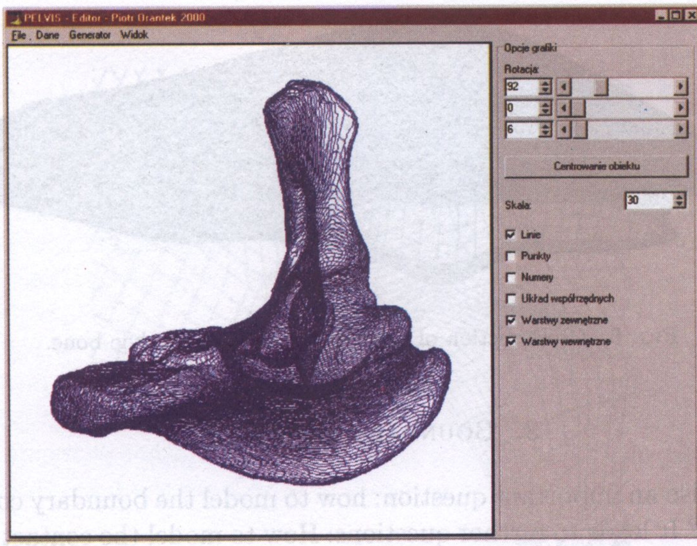


FIG. 4. Geometrical model of a human pelvic bone.

On the grounds of geometrical features (curves and surfaces), finite element meshing is done. The FEM model of pelvic bone and cross-section of the pelvic plate is shown in Figs. 5 and 6, respectively.

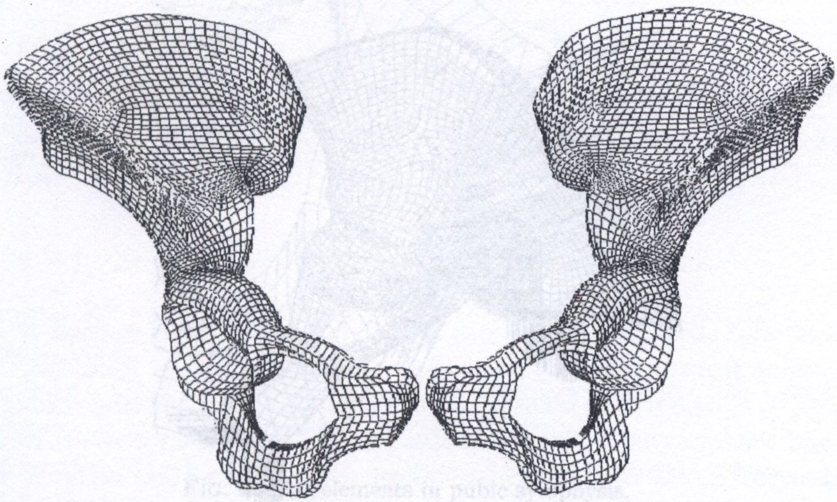

FIG. 5. Solid model of human pelvic bone.

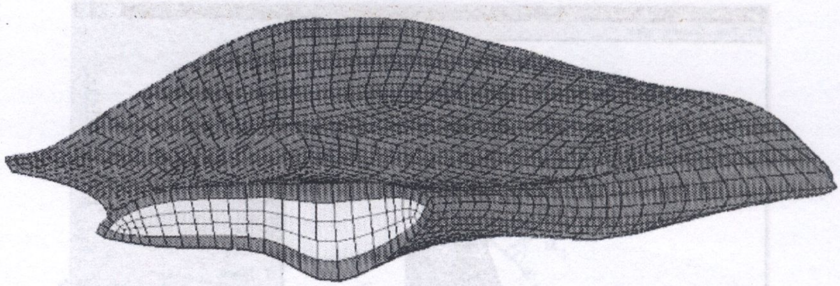

FIG. 6. Cross-section of solid model of human pelvic bone.

## 3. BOUNDARY CONDITIONS

There is also an important question: how to model the boundary conditions in a pelvic bone? It leads to further questions: How to model the contact with other elements of the bone system? What do we know about the stiffness of support? How to model the load? Several models with different boundary conditions were analyzed [2, 6, 7, 11].

General view of a numerical model of pelvic bone with the assumed boundary conditions is shown in Fig. 7. In the contact area with sacral bone, the boundary conditions are given using bar elements (rods) in directions of coordinate axes (Fig. 8). In pubic symphysis, the boundary conditions are given in the symmetry plane as restraints in selected co-ordinates (selected components in nodes), or

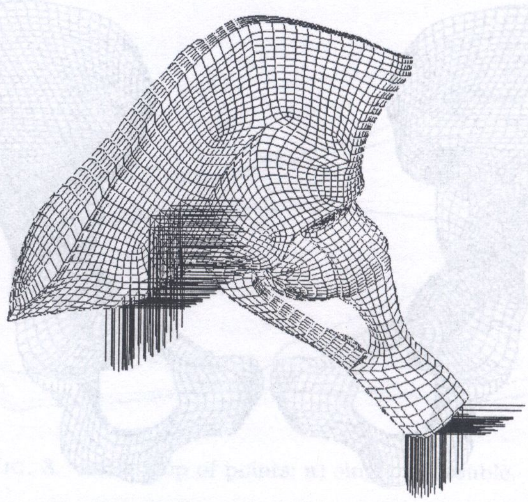

FIG. 7. General view of a numerical model of human pelvic bone with boundary conditions.

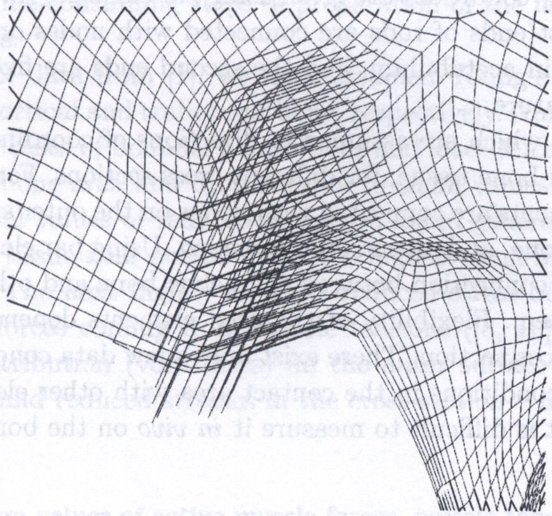

FIG. 8. Bar elements in the contact area with sacral bone.

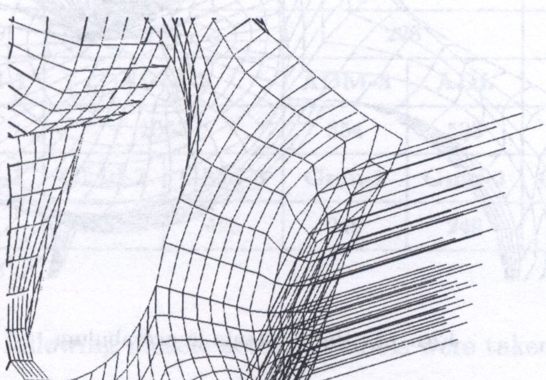

FIG. 9. Bar elements in pubic symphysis.

by using bar elements in directions of two coordinate axes (Fig. 9). In acetabulum, the boundary conditions are given using bar elements (in radial co-ordinate, Fig. 10). The first ends of rods are connected with nodes on the outer surface of finite element in acetabulum, and the second ends are fixed in the center of acetabulum curvature.

Bar elements, which are assumed in directions of coordinate axes, are modelled as a cortical bone tissue, for different cross-sections. For all cases, the first ends of axial elements are connected with nodes on the outer surface of numerical model of pelvic bone, and second ends are fixed. Using bar elements it is possible to model flexible connection between the pelvic bone and other elements of the human bone system. Flexibility of the used elements depends on the material properties and cross-section. There exist only a few data concerning the flexibility of boundary conditions in the contact area with other elements of the bone system because it is difficult to measure it *in vivo* on the bone tissue.

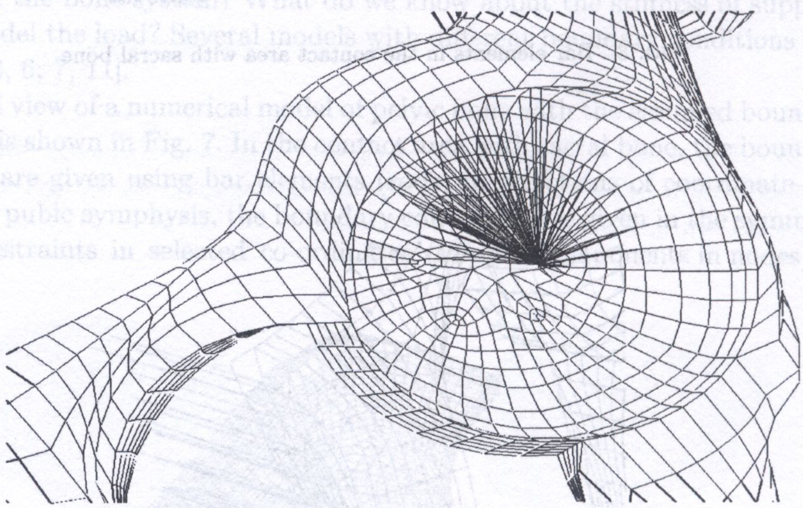

FIG. 10. Bar elements in acetabulum.

Stress and strain distribution in a human pelvic bone is a result of external load coming from the upper body weight and muscle forces. Referring to earlier works [3, 4, 5], the model takes up 23 muscle tensions influencing through pelvic bone and tendons the insertion surfaces (Table 1). Muscle forces are depicted in the numerical model as loads spread out on nodes on insertion surfaces. The load slant to the surface of pelvic bone at angle determined by directional cosines of muscle tension effect line. Muscle tension load does not include the components caused by passive fiber stretch into consideration.

Boundary conditions and load are implemented manually. Currently, it is one of the more time-consuming steps of preparation of the numerical model of human pelvic bone.

On the basis of data from [1], we have assumed Young's modulus 15 GPa and 100 MPa for cortical and trabecular bone, respectively. Numerical results are obtained for many cases of boundary conditions and load. Maximum stress value occurs in cortical bone area and drop sharply to small values at the trabecular bone area. Maximal values of reduced stress are found generally in two regions – in cortical bone tissue near acetabulum (the highest) and in the contact area with sacral bone. The next higher values of reduced stress occur at the points of application of forces coming from muscle tensions ([8, 10]). An example of reduced stress distribution (von Mises) on the outer surface of pelvic bone is shown in Fig. 11, and reduced stresses in the cross-section – in Fig. 12.

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

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| | **RF** | **S** | **IP-1** | **IP-2** | **GRA** | | |
| $F_{max}$[N] | 835 | 148 | 1006 | 1006 | 165 | | |
| | **GMx-1** | **GMx-2** | **ST** | **SM** | **BCL** | | |
| $F_{max}$[N] | 1559 | 780 | 226 | 1359 | 745 | | |
| | **ADM-1** | **ADM-2** | **ADM-3** | **ADL** | **ADB** | **PC** | |
| $F_{max}$[N] | 345 | 1063 | 354 | 593 | 452 | 188 | |
| | **GMd-1** | **GMd-2** | **GMd-3** | **Gmu-1** | **Gmu-2** | **Gmu-3** | **TFL** |
| $F_{max}$[N] | 425 | 425 | 425 | 249 | 249 | 249 | 286 |

In the Table 1, the following muscle actons symbols were taken:

- flexors: RF – rectus femoris, S – sartorius, IP – iliopsoas, IP-2 – psoas maior, GRA – gracilis,
- extensors: GMx – gluteus maximus, ST – semitendinosous, SM – semimembranosous, BCL – biceps femoris caput longum,
- adductors: ADM – adductor magnus, ADL – adductor longus, ADB – adductor brevis, PC – pectineus,
- adductor muscles and stabilising the pelvis: GMd – gluteus medius, GMu – gluteus minimus, TFL – tensor fasciae-latae.

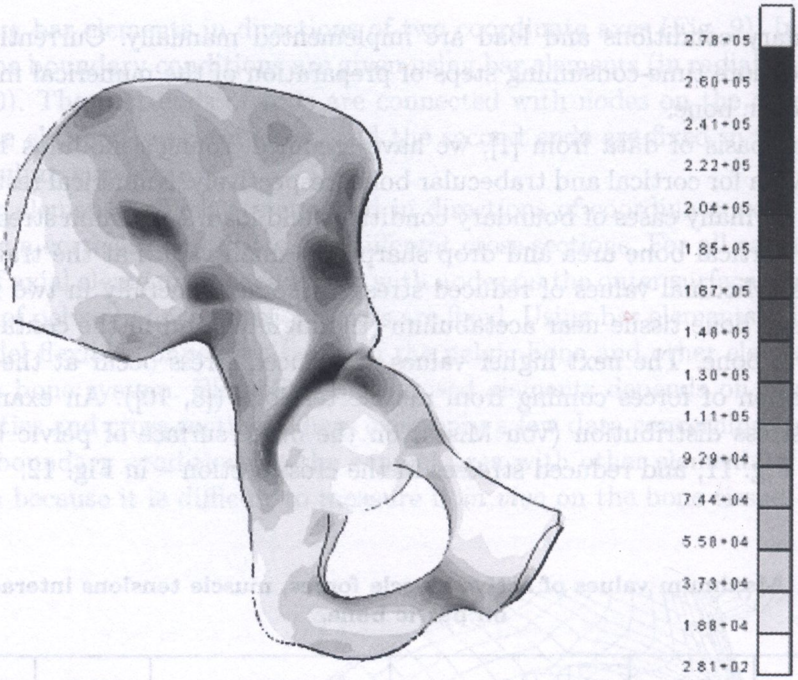


FIG. 11. Reduced stress distribution on outer surface of human pelvic bone (in kPa).

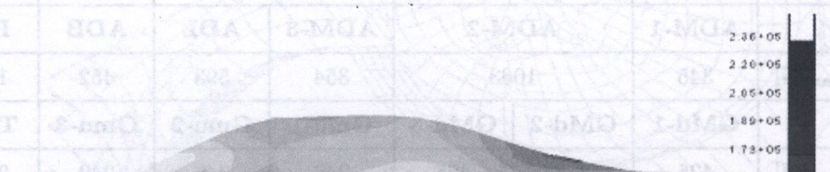


FIG. 12. Reduced stress in cross-section of human pelvic bone (in kPa).

## 4. Conclusions

- The presented numerical program prepares the geometrical model in the Patran code.
- All geometrical features read from the coordinate measuring machine are taken into account.
- The applied program enables to take into account layered structure of the bone tissue.
- The presented numerical code enables to while away the time of creation of numerical model of human pelvic bone.
- On the grounds of geometrical model, numerical model of a pelvic bone is performed using solid finite elements.
- Numerical analysis of a human pelvic bone shows that the stress distribution depends on the boundary conditions, e.g. on stiffness of the given restraints.
- There are also questions: how to model the contact with other elements of the bone system and what values of material coefficients should be assumed.

## Acknowledgments

The work was done as a part of the research project 8T11F02618 sponsored by the Polish State Committee for Scientific Research.

## References

1. R. Będziński, *Biomechanical engineering* [in Polish], Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław 1997.
2. M. Dalstra, R. Huiskes, *Load transfer across the pelvic bone*, J. Biomechanics, **28**, 6, 715–724, 1995.
3. A. M. Dąbrowska-Tkaczyk, A. John, *Numerical modelling of the human pelvic bone load*, Scientific Papers of Department of Applied Mechanics, **7**, 71–78, Gliwice 1997.
4. A. John, *The boundary conditions in numerical model of human pelvic bone*, Acta of Bioengineering and Biomechanics, **1**, Supplement 1, 918–222, 1999.
5. A. John, *The boundary conditions and yield criteria in numerical analysis of human pelvic bone*, Proceedings of the VIII-th International Conference Numerical Methods in Continuum Mechanics, 149–151, Žilina 2000.
6. A. John, *Numerical analysis of solid and shell models of human pelvic bone*, Lectures Notes in Computer Science, 1988, Numerical Analysis and Its Applications, 764–771, Springer-Verlag, Berlin Heidelberg 2001.

7. A. JOHN, *Yield criteria and stress concentration in numerical analysis of human pelvic bone*, Cax Techniques, Proceedings of the 4th International Scientific Colloquium, 289–296, Bielefeld 1999.

8. A. JOHN, P. ORANTEK, *Creation of numerical model of human pelvic bone*, Proceedings of the XL Symposium Modeling in Mechanics, 119–120, Wisła 2001.

9. A. JOHN, P. ORANTEK, *Numerical modeling of human pelvic bone*, Proc. of the 2nd European Conference on Computational Mechanics, Vol. of abstracts, **2**, 786–787, Cracow 2001.

10. A. SEIREG, *Biomechanical analysis of the musculoskeletal structure for medicine and sport*, Hemisphere Publishing Corporation, New York 1989.

*Received March 16, 2001.*