

NUMERICAL COMPLEXITY OF SELECTED  
BIOMECHANICAL PROBLEMS

MARCIN WIERSZYCKI  
WITOLD KĄKOL  
TOMASZ ŁODYGOWSKI

*Institute of Structural Engineering, Poznań University of Technology*  
*e-mail: Marcin.Wierszycki@put.poznan.pl; Witold.Kakol@put.poznan.pl;*  
*Tomasz.Lodygowski@put.poznan.pl*

The paper describes various aspects of numerical modeling of biomechanical problems by the finite element method. The authors would like to present what they mean by the numerical complexity of modeling of biomechanical problems. The attention is focused on numerical simulation of dental implants and human lumbar spine motion segment (L4-L5). In both cases, acquisition and creation of geometry, number of DOFs, combining different types of elements, properties of the material, contact definitions, loads and boundary conditions are difficult tasks. The acquisition of geometric data of living body parts can be realized only by using noninvasive techniques like NMR or CT. The processing of these data requires specialized software and methods. The methodologies of defining mechanical parameters of human tissues are usually inaccurate and have to be used in practice on living people very carefully. The constitutive data in literature are usually grossly inconsistent. In numerical simulations, custom material formulations and modeling techniques should be used. It is difficult to describe real-world loads and boundary conditions since both are very complex and changing. Load scheme models are global and force values are very difficult to obtain. Boundary conditions are necessarily very simplified but they should reflect specific biological behaviors and conditions. Nevertheless, the numerical simulation by means of the finite element method can be helpful and useful during solving biomechanical problems like fatigue analysis of dental implants or estimating the stiffness of human lumbar spine segment.

*Key words:* biomechanical, finite element method, fatigue analysis, dental implant, spine segment

## 1. Introduction

Nowadays, the finite element method plays the key role in solving engineering problems in many fields of science and industry, and can be successfully applied also in simulations of biomechanical systems (Będziński, 1997; Geng *et al.*, 2001; Eberlein *et al.*, 2002; Kąkol *et al.*, 2003a; Sakaguchi and Borgersen, 1993). This method has been a well established one, used in biomechanical simulations for over 20 years so far. It allows taking into consideration the key features like material inhomogeneity and anisotropic mechanical properties of tissues as well as a very complicated geometry of human body parts (Eberlein *et al.*, 2002; Ogurkowska, 1992; Skaggs and Weidenbaum, 1994; Swartz and Wittenberg, 1991). Furthermore, it is proved (Dierich *et al.*, 1992; Eberlein *et al.*, 2002; Hędzulek *et al.*, 2003; Kąkol *et al.*, 2003c) that function, failure, prediction of changes and remodeling of biostructures are related to stress and strain fields in tissues which may be calculated with the help of FEA. Thus, FEA is an efficient tool for testing biomechanical sets, but it is still often very difficult to obtain useful and valuable results for these kinds of problems. The main reason for this is the complexity of biostructures and the resulting complexity of numerical simulations. These complexities are meant as difficulties of modeling as well as overcoming them by means of specific simulation techniques. These complexities are significant within each step of a numerical analysis.

### 1.1. Geometry creation

The first step of carrying out virtual simulations of biomechanical systems is correct definition of model geometry. Besides topology, additional data such as volume density, can also be interesting. The acquisition of geometric data of living body parts can only be realized by using a noninvasive technique such as Nuclear Magnetic Resonance (NMR) or Computer Tomography (CT). Other techniques, such as 3D scanning may also appear to be helpful but only for validation purposes (Rychlik *et al.*, 2004).

The preparation of geometric models with the use of NMR or CT consists of two stages. In the first stage, the rows of two dimensional pictures for subsequent slices of the analyzed parts of body are obtained. In the second step, the data must be processed with a specialized code that converts it into a 3D model in a format readable with the CAD software. In the case of a complex geometry, the above procedure is not a trivial task. In fact, it enables us to prepare quite accurate geometric descriptions of biomechanical models (Ogurkowska *et al.*, 2002a).

A more difficult procedure takes place in the transfer from a geometric (CAD) model to a discrete (FEA) model. The geometric data usually cannot

be used directly in FEA codes. Advanced preprocessors of FEA can import and mesh the surface of a solid geometric model prepared in the CAD software. In practice however, this procedure is limited only to objects with not really simple but regular shapes. If the geometry is complex, a lot of errors may occur while it is being imported, and the creation of mesh of finite elements may not succeed. The most significant sources of errors are: loss of geometry creation history (parent-child relations between geometry features), number of details (e.g. number of triangles) and precision of geometrical descriptions (e.g. order of the polynomial). Because of complexity of the biostructure geometry and errors occurring during the geometry transfer, it is often not possible to automate the process of FEA mesh generation. Finally, the finite element model must be created "by hand" on the basis of information from NMR, CT and contact scanners (Ogurkowska *et al.*, 2002b).

## 1.2. Material definition

Mechanical properties of tissues are extremely complex. They have usually nonlinear, anisotropic and viscous characteristics (Eberlein *et al.*, 2002; Skaggs and Weidenbaum, 1994). Moreover, these characteristics are always changing as a result of natural processes such as remodeling. Obtaining a reliable and representative value of these characteristics poses another important problem. Experimental methods of obtaining mechanical parameters of biomechanical materials are often not accurate enough and, first of all, cannot be used in research on living people. There is a body of literature on the assessment of physical parameters of tissues, but using such data is often difficult because the data are sometimes grossly inconsistent (Swartz and Wittenberg, 1991). Among others, the reason of this is the heterogeneity of tissues. This heterogeneity does not cease to become more conspicuous while testing materials taken from various donors (Skaggs and Weidenbaum, 1994). Practical application of FEA will be possible once the actual basic data describing an individual's traits are provided. The method meeting the requirements above is the one relying on the existing relation between the structure of osseous tissue and its mechanical properties and radiological density (Ogurkowska, 1992).

The selection of the right types of material models and possible simplifications depends on prepared simulations, e.g. for a dynamic process, the viscoelastic behaviour will play the key role, and for a long term analysis the remodeling phenomena will be most significant. Generally, in the case of static or quasi-static stress-strain analyses for "hard" tissues, like a bone, linear, iso- and orthotropic elastic materials are used. For "soft" tissues, like ligaments, a hyperelastic material is used (Eberlein *et al.*, 2002). The next very important aspect is choosing and assigning the appropriate type of a finite element. In

order to describe the behaviour of very complex tissues, a combination of a few different types of elements is often necessary.

### 1.3. Load and boundary conditions

Load schemes and boundary conditions are very complex and difficult to define for biomechanical numerical models. For simulations of biostructures which always constitute a part of a greater entity – a living body – the loads are applied to the parts of body with muscles, ligaments and by contact between them. Furthermore, directions and values of these loads are constantly changing and are very difficult to obtain. Similarly, characteristics of boundary conditions are specified by surrounding tissues and organs. The changes in boundary condition characteristics are caused by the phenomena of tissue remodeling. Because of these reasons, it is not possible to describe in full details the loads and boundary conditions, and in a numerical model they usually must be globalised and simplified. The theoretical models of global load schemes for some biomechanical system can be found in literature (Adams *et al.*, 2002; Będziński, 1997).

### 1.4. Efficiency of simulation

The last but not least group of complexities are technical difficulties of numerical calculations. For the sake of the simulation complexity, only the size of numerical models, convergence of iteration algorithms and time of calculations may be a problem. The models that describe individual features of living body parts or very complex biomechanical structures must contain a very detailed geometric description. These models can have a few hundred thousands elements and degrees of freedom. It is possible to calculate such large problems with the present-day FEA codes by means of computers. Still, if the size of a model is near to one million DOFs, it may turn out not to be so easy to carry out such an analysis. In the case of large problems, already common workstations and also low-class servers have significant limitations. The source of these problems can be computer architecture limitations, memory addressing, CPU time needed or size of temporary and result files.

If the main goal of numerical calculations of biomechanical structures is their practical application, the efficient methods of simulations in medical treatment and rehabilitation are needed. For such complex structures and numerical models, this efficiency requires many simplifications and a hierarchical approach (Kąkol *et al.*, 2003c).

In the next sections, more detailed aspects of the complexity of numerical models of biomechanical problems will be discussed. Selected problems, such as fatigue analysis of dental implants and evaluating stiffness of the human lumbar spine segment are considered.

## 2. Example I – numerical analysis of dental implants

Implants are a commonly applied treatment method of dental restorations. Unfortunately, numerous clinical observations point to the occurrence of both early and late complications. In many cases, these problems are caused by mechanical fractures of the implants themselves (Hędzelek *et al.*, 2003; Kąkol *et al.*, 2002). The most frequent complications are loosening of the connecting screw, fracture and cracking of dental implant parts. While the loosening of the connecting screw causes mostly patient's discomfort in the implant usage, the cracking leads to much more serious complications and makes further treatment extremely difficult. To understand the reasons of observed mechanical complications, it is necessary to know stress and strain fields in implant components as well as changes in boundary conditions. Cyclic loads and a character of fracture indicate material fatigue as the basic cause of this fracture (Kocańda, 1985). The authors would like to confirm this proposition by means of numerical simulations of a dental implant system (Kąkol *et al.*, 2003d). The calculation was carried out with the commercial code ABAQUS/Standard. For fatigue calculations, the *fe-safe* Works was used.

In the analysis of a dental implant, the implant structure is not trivial, however it is a purely mechanical problem. The loads and boundary conditions have already more complex, biomechanical reasons. The most significant complexities of finite element stress-strain and fatigue analyses are geometry and mesh preparations, fatigue material characteristics, implant assembly, physiological changes of loads and, finally, definition of boundary conditions describing it as a bone.

### 2.1. Geometry

Numerical models were created on the basis of technical documentation of the commercial implant system OSTEOPLANT. It is a commonly used system consisting of an implant and abutment with a non rotational hexagonal connection assembled by a screw. For full simulation of the implant structure behaviour, a geometrically complex three dimensional model is necessary. This model, which includes a spiral thread, enables taking into consideration a few important aspects such as full simulation of kinematics of the implant, description of the multiaxial state of stress and, the most important, the possibility of simulation of screw loosening. The most interesting result will be the relationship between the torque, friction coefficient and loosening or fatigue life of the screw under cyclic loads. Unfortunately, a three-dimensional model is very large (ca. 500 000 DOF). Due to the fact that most parts of the implant are axisymmetric, connecting the axisymmetric and fully three-dimensional concept of modeling is a good idea of simplification. The ABAQUS/Standard

offers axisymmetric solid CAXA elements that are capable of modeling nonlinear asymmetric deformation. The CAXA elements are intended for analysis of hollow bodies such as pipes, but they may also be used to model solid bodies with some limitations. These elements use standard isoparametric interpolation in the axi-symmetric plane, combined with the Fourier interpolation with respect to the angle of revolution. The asymmetric deformation is assumed to be symmetric with respect to the axi-symmetric plane at an angle  $0$  or  $\pi$  [1, 2].

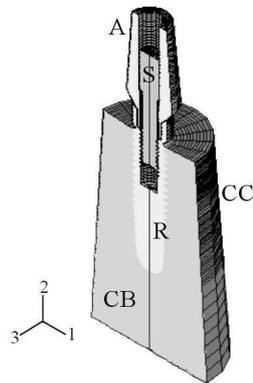


Fig. 1. Axi-symmetric model of a dental implant with a part of the jaw bone: CB – cancellous bone, CC – cortical bone, R – implant, S – screw, A – abutment

An axi-symmetric model of an implant was created with the application of CAXA elements (Fig. 1). This approach reduces geometry description of the implant model from three dimensional to two dimensional. The threads of the screw and the implant body were simplified to axi-symmetric parallel rings. This assumption results in the size of the problem ca. 75 000 DOF and the cost of calculation being significantly reduced (Table 1).

**Table 1.** Dimensions of models used in FEA of the implant

	Axi-symmetric model	3-dimensional model
Elements	5 891	76 345
Nodes	54 160	166 125
Degrees of freedom	74 820	498 376

## 2.2. Material properties

The part of the jaw is composed of two kinds of bones, the cancellous bone and the cortical one (Fig. 1). The problem of describing mechanical constitutive law of bones is very complex. Mechanical characteristics and the internal

microstructure of cortical and cancellous bones are nonhomogeneous, anisotropic and variable in time. Changes of bone characteristics are caused by the phenomenon of tissue remodeling. It is very difficult to take these aspects into consideration in implant models. In finite element analysis, many concepts of description of mechanical properties of bones could be applied – starting from very simple linearly elastic isotropic, going through more complicated, transversely isotropic or orthotropic and ending up with very complex, nonlinear anisotropic ones. The assumed material characteristics of the jaw bones are linearly elastic, homogeneous and isotropic (Table 2). This simplification is justifiable due to the role which the bone plays in fatigue analysis of an implant. The most important here is the influence of bone loss around an implant as well as bone flexibility on implant boundary conditions and implant fatigue life (Hędzulek *et al.*, 2003). The implant is made of a medical titanium alloy the mechanical properties of which are nonlinear. Their description was based on the certificate of conformity.

**Table 2.** Stiffness of jaw bone

Bone	Young's modulus $E$ [MPa]
Cortical	
all schemes	13 000
Cancellous	
1st scheme	9 500
2nd scheme	5 500
3rd scheme	1 600
4th scheme	690

### 2.3. Mechanical assembly

The implant system is seemingly simple, but in fact it is quite a complex mechanical system (Kąkol *et al.*, 2002; Merz *et al.*, 2000; Sakaguchi and Borgersen, 1993). An important aspect of the implant assembly is the modeling of tightening. For this purpose, it is necessary to define the contact surfaces between the root, abutment and screw. The friction characteristic on these surfaces is one of the key parameters influencing the preload axial force, reduction of implant components mobility, resistance to screw loosening but also fatigue life of the whole implant. For the friction coefficient, a value ranging between 0.1 (as in a specially finished surface) and 0.5 (as in dry titanium to titanium friction) may be found in literature. In the present analysis, three different friction coefficients (0.1, 0.2 and 0.5) were considered. The friction characteristic is one of the key parameters influencing the preload axial

forces  $V$ , reduction of mobility of implant components and screw loosening resistance.

The first step of simulation was the tightening of the screw. To simulate this, the middle part of the screw was subjected to temperature loading. The thermal expansion property of the screw material was orthotropic. It was defined in such a way that the shrinking occurred only in the direction of the screw axis. The axial force in the tightened screw was calculated from an empirical equation. It depends on the friction coefficient and torque. The effect of this force (ranging from 80 to 850 N) was replaced by a temperature field (Kąkol *et al.*, 2003d). In the case of a three-dimensional model, simulation of the tightening can be defined in such a way that it describes a real physical process. If this analysis is performed in an implicit code, a lot of iterations are required, and nevertheless, it is often very difficult to obtain a convergent solution. For simulations of the screw tightening, quasi-static analysis in an explicit code is recommended. In this case, the mass scaling is one of the most useful techniques [1, 2]. In the next stage, the simulation was carried out in an implicit code.

#### 2.4. Loads

The external loads of the implant model were applied in the second step of simulation. The values and directions of forces were assumed according to a physiologically proven scheme. To estimate the least favorable distribution of stress, only the maximal realistic occlusal forces were taken into account. The loading of the implant was never axial. The vertical component of it is estimated at 600 N and the horizontal at 100 N. For fatigue calculations, it was necessary to define the character of load changeability in the shape of a load-time curve, the so-called load signal. In the applied low-cycled scheme of 24-hour loads, the average values were 60 N (Hędzelek *et al.*, 2003; Kąkol *et al.*, 2003d).

#### 2.5. Boundary conditions

In the first stage of implant analysis, all degrees of freedom at the bottom part of the implant body were fixed. This assumption seemed to have its explanation in dental practice where no movements of implants under physiological loads are acceptable. However, the difference between infinitely stiff fixing and even low flexibility is significant, especially in the cyclic loading and fatigue damage context. In the next stage of implant analysis, the boundary conditions of implants are modeled as a small part of the jaw bone. The geometry of a small part of the jaw surrounding the implant is very simplified but it enables us to take into consideration the changes in implant fixing conditions. The

changing flexibility of the bone and bone loss phenomena are also very important, especially because the bone loss has significant influence on the implant behaviour, stress distribution and, therefore fatigue damage. The degree of encasement and osseointegration of the implant may not be 100%. It depends on the bone quality, stresses developed during healing and function as well as the location of the implant in the jaw. This percentage may decrease to as low as 50%. This is caused by phenomena of bone remodeling. In these analyses, three levels of osseointegration were considered. In the case of the first level, the implant body was fully fixed in the jaw bone. In the next two, the degree of the implant body embedding decreased to 75 and 50%, respectively (Fig. 2) (Hędzulek *et al.*, 2003).

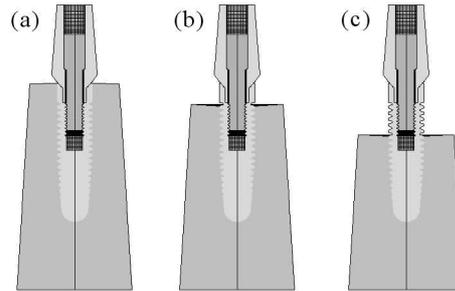


Fig. 2. Levels of dental implant osseointegration in the jaw bone: (a) 100%, (b) 75%, (c) 50%

## 2.6. Fatigue analysis

Fatigue calculations were carried out by the *fe-safe* program which uses algorithms incorporating the multiaxial plasticity model to estimate the fatigue life. These algorithms are based on the stress results obtained from the finite element analysis (Fig. 3), variations in loading, hysteresis loop cycle closure, and material properties. Elastic stresses from the FEA model are translated into elastic-plastic stresses by means of biaxial Neuber's rule and cyclic material properties. In order to estimate the service life of an implant, a designed life is defined. *fe-safe* calculates the factor (FOS – Factor Of Strength) by which the stresses at each node can be increased or reduced to give the required life. The above is the most interesting and vivid for our case. During a single analysis concerning each node separately, a 6-stress tensor is used to calculate the principal stresses and strains and their orientation. Stress concentration and scale factors are applied at this stage. A rainflow cycle counting algorithm is used to extract fatigue cycles. For biaxial fatigue methods, the critical plane procedure is used to calculate the orientation of the most damaged planes at nodes (3, Bishop and Sheratt, 2000; Draper, 1999).

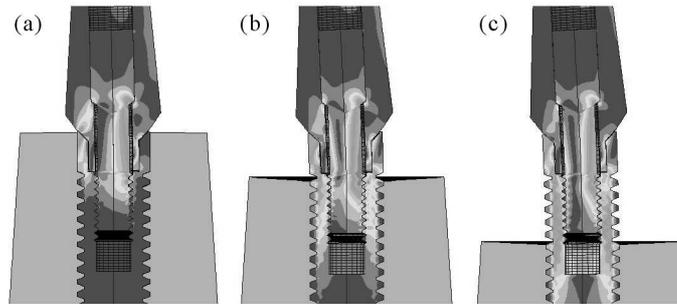


Fig. 3. Huber-Mises equivalent stress distribution after bending for three levels of osseointegration: (a) 100%, (b) 75%, (c) 50%

The fatigue life of the implant screw was calculated for nine separate cases of loading, three cases of boundary conditions and three cases of the jaw cancellous bone density. For all of these cases, the same cyclic scheme of loading was assumed. A twenty-four-hour changeability scheme was assumed as the signal (Fig. 4), while the number of days corresponding to four years was assumed as the number of cycles.

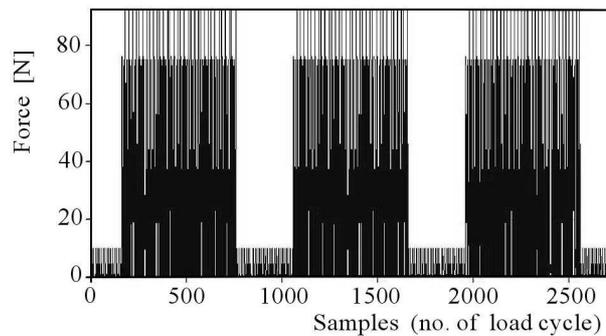


Fig. 4. Cyclic scheme of loading – load signal

The FOS distribution analysis for particular cases indicates the axial forces in the screw and the changes in the scheme of boundary conditions which have the greatest influence on fatigue changes. For different bone densities and at the same time, divergent stiffness of boundary conditions and significant differences of stress distributions in the screw are noticeable. Yet, it does not lead to serious fatigue changes. For axial forces above 600 N, there is a noticeable increase in the areas endangered by fatigue failure. The degree of required stress reduction reaches ca. 30%. In the most unfavorable load case, the maximum axial force results from a high torque and a very small friction coefficient on the screw thread. In two-part implants, this high tightening force is motivated by biological and medical aspects. However, the increase

in torsion and decrease in friction coefficients entail reduction of the implant components fatigue life (Hędzelek *et al.*, 2003; Kąkol *et al.*, 2003b,d).

### 3. Example II – numerical analysis of human spine motion segment

The main goal of this study was to conduct mechanical analysis of the human lumbar spine segment which can be helpful in medical treatment, diagnosis and rehabilitation (Glema *et al.*, 2004b; Kąkol *et al.*, 2003a). Special attention was focused on the role of intervertebral disc in the characteristic of motion segment stiffness (Glema *et al.*, 2004a). The function, failure, prediction of pathological changes and the remodeling of the motion spinal segment are all related to stress and strain fields in its tissues and may be calculated with the FEA. To calculate the stress and strain fields in the case of such a complex biostructure, computational models should base on experimentally determined material properties, realistic geometry, appropriate boundary conditions and load schemes (Ogurkowska *et al.*, 2002a,b).

In the case of a practical application of such a simulation of surgery or analyses of spinal equilibrium and stability, simplified models of the lumbar spinal motion segment should be used. The simplification of the lumbar spinal motion segment is based on a hierarchical approach. The intervertebral disc is replaced with a connector type element. The kinematic and kinetic relationships describing behaviour of the connector element may be very complex (Glema *et al.*, 2004a). This behaviour is defined on the basis of results obtained from the FEA simulations which were carried out with the ABAQUS/Standard commercial code. In the next stage, validation of the simplified model of the lumbar spinal motion segment is performed (Glema *et al.*, 2004a).

The anatomy of the motion segment is very complex and is of the utmost importance at each stage of numerical simulation of the motion segment.

#### 3.1. Geometry

The shape of all parts of the motion segment is shown in Fig. 5. Their individual anatomical characteristics may be quite significant. In order to construct a finite element model of the spinal motion segment, geometric data of the real object are required. In the case of acquisition of such complex geometric data, special methods are used to scan the 3D geometry of the real object and transform it into a CAD system. Nowadays in medicine, the most popular techniques of scanning are Computer Tomography (CT) and Nuclear

Magnetic Resonance (NMR) (Ogurkowska *et al.*, 2002a). Both of these methods have some limitations and their results may not always be used directly in FEA codes.

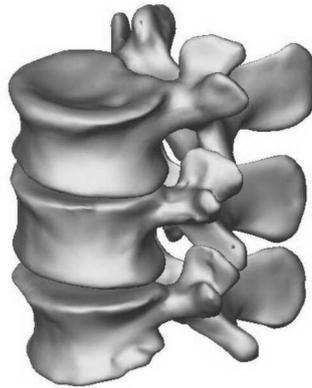


Fig. 5. Lumbar vertebra (L3-L5) – 3D geometrical model

In this study, a CAD model is constructed with the use of the 3D-Doctor software (obtainable from Able Software Corp.) that converts the data from CT to 3D geometric model. Unfortunately, this model cannot be used directly for structural analysis. The geometric model is imported into the SolidWorks CAD system. Following a lot of repairing and rebuilding operations, the geometric model is imported into the ABAQUS/CAE preprocessor and meshed in the final step. Special techniques are required to perform the meshing step successfully. Partitioning and smoothing of the geometry are indispensable here. At the same time, it is important to maintain the most critical geometric parameters. In fact, it is done in hundreds of small steps, and the intermediate results are compared with some existing available data. The final geometry, which is a basis for meshing, has a very complex description of topology (Ogurkowska *et al.*, 2002b).

### 3.2. Material properties

Each part of the motion segment needs a different approach with respect to material modeling. The anatomy and function of individual parts of the motion segment are of great importance in the case of selection of finite element formulations, modeling techniques and constitutive laws. The motion segment of a human spine consists of two vertebral bodies, intervertebral disc, posterior elements and spinal ligaments (Fig. 6) (Adams *et al.*, 2002; Będziński, 1997).

Basically, the vertebra is composed of the cancellous bone and cortical bone. For modeling of the vertebra, solid and shell elements were used with linear elastic orthotropic material properties. Two kinds of bone tissue were taken

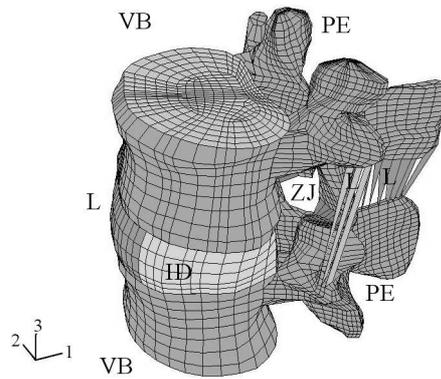


Fig. 6. FEA model of the motion segment: VB – vertebral bodies, ID – intervertebral disc, L – ligaments, PE – posterior elements, ZJ – zygapophysial joints

into consideration: cortical and cancellous. For the cortical bone of the vertebral body, which is a very thin sheet in fact, shell elements were used. For the cancellous part, in turn, solid 3D hexahedral elements were applied (Table 3). The intervertebral disc is composed of three different parts: annulus fibrosus, nucleus pulposus and vertebral endplates. This complex structure enables movements between the vertebral bodies. These are lateral and sagittal bending, twisting, and small sliding. The annulus fibrosus consists of 10-20 layers of collagen fibers oriented  $+30^\circ$  and  $-30^\circ$  with respect to the circumferential axis. The nucleus pulposus is a hydrated gel located at the center of the disc.

**Table 3.** Types of elements and materials in FEA model

Anatomical part of the motion segment	Element		Material model
	type	number	
vertebra body	shell	6 400	elastic, orthotropic
	solid	13 966	elastic, orthotropic
intervertebral disc	solid	1250	elastic, isotropic
	surface with rebars	1004	elastic or hyperelastic, isotropic, no compression
	fluid	564	incompressible fluid
vertebral endplates	shell	814	elastic, isotropic
ligaments	membrane	129	elastic or hyperelastic, isotropic, no compression
zygapophysial joint	shell	56	elastic or hyperelastic, isotropic
	solid	94	elastic, isotropic

Both components maintain the stiffness of the disc against the compression loading and allow for some degree of movement between the vertebral

bodies. The annulus fibrosus can be modeled as an anisotropic material or by structural elements which introduce this anisotropy.

In this study, the second approach was adopted. The ground matrix of the disc annulus was modeled with 3D solid elements. The collagen fibers were modeled with rebar type elements embedded into the 3D solid elements (Table 3). The rebar elements are defined as surface layers of uniaxial reinforcements in the solid elements with uniformly spaced reinforcing bars. Such layers are treated as smeared layers with a constant thickness which is equal to the area of each reinforcing bar divided by the reinforcing bar spacing. This technique allows for easy alterations in the number of layers, section properties and its orientation. What is more, it is independent of the element re-meshing mentioned above. In our study, we calibrated the 3D embedded elements with data published by Skaggs and Weidenbaum (1994). So, the number of rebars and their cross-sectional areas were calculated on the basis of the published data (Skaggs and Weidenbaum, 1994). The nucleus pulposus, assumed here to be incompressible, was modeled as a fluid-filled cavity using hydrostatic fluid elements with initial pressure 2 MPa. The hydrostatic fluid elements cover boundaries of the nucleus pulposus. They share the nodes at the cavity boundary with the standard elements of the annulus fibrosus (Table 3). The posterior elements control the position of the vertebral bodies such as a pair of stout pillars of a bone called pedicles, two transverse processes, two superior articular processes and two laminae endings with a spinous process. These several processes serve as sites of attachment for muscles that control the lumbar vertebral column.

All parts of the spinal segment are connected by ligaments. The most definite ligament is the ligamentum flavum which consists of an elastin that connects the lower end of the internal surface of one lamina to the upper end of the external surface of the lamina below and closes the gap between the consecutive laminae. The transverse processes are connected with thin sheets of collagen fibers. The opposing edges of spinous processes are connected with the collagen fibers referred to as the interspinous ligament and the supraspinous ligament. In addition to the ligaments of the posterior elements, the lumbar vertebral column is reinforced by ligaments that connect the vertebral bodies. These are the posterior longitudinal ligament and the anterior longitudinal ligament. The vertebral bodies, the disc and the posterior elements create a complete structure of the lumbar vertebral column which maintains stability and controls of movements. The vertebrae are jointed with zygapophysial joints which create a locking mechanism between them. They block axial rotation and forward sliding of the lumbar vertebrae. For seven spinal ligaments, membrane elements describing the material devoid of compression were used. For the zygapophysial joints, shell and solid elements were used (Table 3).

The superior and inferior vertebral endplates are the cartilage plates that cover the superior and inferior aspects of the discs and bind the disc to their respective vertebral bodies. The cartilaginous endplates were modeled with thin shell elements (Table 3). The intervertebral disc and ligaments play an essential role in motion of the spine and in the contribution to the general stiffness of the spinal segment.

As it can be seen, it is not sufficient for such a complex structure as the motion segment to use only material characteristics, even if they are very complex. In this case, it is necessary to mix different material descriptions and modeling techniques. Material complexities must be considered when choosing material constants for the finite element model. There is rich literature on the assessment of physical parameters of bones and soft tissues, but it is difficult to make use of the basic piece of information because the constitutive data of the same human bones are usually grossly inconsistent. The methods of defining biomechanical parameters are inaccurate and cannot be used in practice on living people. The osseous tissue of the spine is very heterogeneous in its repetition structure and its properties change depending on the sample placement in the vertebra. On the basis of the relationship existing between mechanical properties and radiological density of bones, Young's modulus of the FEA model of the vertebral cancellous bone of the motion segment can be determined (Ogurkowska *et al.*, 2002b). The rest of the data was taken from literature (Adams *et al.*, 2002; Eberlein *et al.*, 2002; Skaggs and Weidenbaum, 1994; Swartz and Wittenberg, 1991).

### 3.3. Load and boundary conditions

In reality, loads of the motion segment are applied with neighbouring vertebrae, muscles, ligament and other connecting tissues. It is very difficult to take into consideration all of them. Thus, it is necessary to use simplified models of load schemes with global equivalent forces and moments (Adams *et al.*, 2002; Będziński, 1997).

For the first stage of the motion segment, analysis in which the separated disc was taken into consideration, representative load schemes with the unit force/moments were applied. The concentrated force and moments were applied to the reference point. In this analysis, the boundary conditions were also global and reduced to one reference point which was fixed. These reference points couple all nodes on the upper and the bottom surfaces of the intervertebral disc.

In the second, validated stage, six load cases with loads and moments typical for the L4-L5 motion segment from literature were taken and applied to the whole spinal motion segment model. The load values were assumed as the half of the damage force. These loads were applied to the reference

point on the upper surface of the vertebral body. For a single motion segment, other motion segments, muscles and connecting tissues, etc., constitute the boundary conditions. Of course, it is not possible to take into account such a realistic description of the boundary conditions. In our model, the bottom surface of the vertebral body was constrained against any movement.

### 3.4. Analysis of stiffness of the motion segment

The whole complex model of the motion segment was validated for four basic load schemes: axial compression, lateral and sagittal bending and torsion. On the basis of the obtained results, it was possible to verify the validity and quality of the model definition. These results are the axial displacement and disc bulge for compression, rotation for bending and axial rotation for torsion. They were compared to values taken from literature (Adams *et al.*, 2002). The numerical model of the whole motion segment had 84 385 DOF and 24 277 elements (Table 3). The complete calculations lasted 22 hours on a standard PC workstation and needed 34 increments.

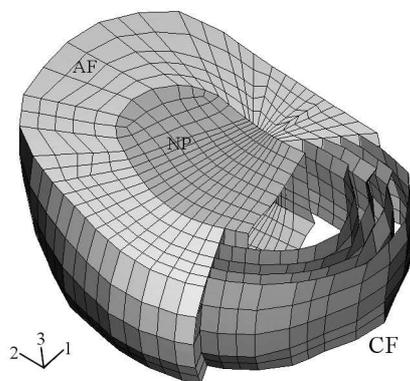


Fig. 7. FEA model of the intervertebral disc: AF – annulus fibrosus, NP – nucleus pulposus, CF – collagen fibers

If we take into consideration the time needed for the preparation of such a complex model, this approach is impossible to apply in future common medical examinations. The basic idea of simplification of modeling of the motion segment is the replacement of the complex structure of the intervertebral disc (Fig. 7) with one connector type element having a complex definition. The elastic behaviour of this type of element can be described as an equivalent stiffness matrix that is in a general case the relation  $\mathbf{F} = \mathbf{K}\mathbf{D}$ , where  $\mathbf{F}$  is the vector (12 components) of generalized forces that act on the segment,  $\mathbf{D}$  (12 components) is the vector of mutual displacements between bones and  $\mathbf{K}$  (144 components) is the stiffness operator of the segment. The values of these matrix components will be obtained from FEA simulations and calcu-

lated with a self made code in the SciLab (open source software similar to MatLab).

For the behaviour of the segment in the range of our interest, i.e. small rotations and small displacements, we can use a tangent stiffness matrix because of the linearity of load displacement relations. In order to follow this approach, we examine a separated disc, which is subjected to twelve loading schemes in the first place. By means of application of the unit loads (axial, shear and moments) and by recording displacements at appropriate points, where relative motion can be estimated, we build the compliance matrix and then the stiffness matrix for the model. Four concepts of connector elements were tested. The first three were based on two node, twelve DOF (six transverse and rotationall) elements the behaviour of which was described by means of a 12 by 12 stiffness matrix. Three definitions of this matrix were prepared for full matrix with 144 non-zero components, reduced matrix with 92 significant non-zero components and symmetrized with 52 non-zero components. The last one was based on a special type of the connector element. It is a two node element, the stiffness of which is defined by a  $6 \times 6$  matrix. This matrix describes relative motion and rotations of these two nodes (Table 4) (Glema *et al.*, 2004a).

The last stage was comparison of the numerical models of motion segments containing a one element model of the intervertebral disc with multi-element, complex disc definition. All four concepts of the equivalent element were studied. The six load cases discussed above were considered (Glema *et al.*, 2004b). The recorded relative displacements and rotations allow for validations of different concepts of intervertebral disc simplifications. Selected results are shown in Fig. 8. It is seen that the response of the simplified intervertebral disc model using connector elements is the closest to behaviour of the multi-element model. This element is recommended for the hierarchical modeling technique. Note here that the response results (Fig. 8) for other elements employed are nearly the same – the curves coincide with each other.

#### 4. Conclusions

On the basis of analysis carried out, it was proved that the finite element method enables one to obtain useful and helpful results for such complex structures as dental implants or human lumbar spine motion segments. On each and every step of these analyses, many obstacles need to be overcome. These obstacles as well as the overcoming them in numerical modeling with specific techniques determine the numerical complexity of biomechanical problems of biostructures. If the main goal of numerical calculations of biomechanical

**Table 4.** Comparison of concepts of disc simplifications

Complex finite element model of disc	One element model with full matrix	One element model with reduced matrix	One element model with symmetrized matrix	Connector element model of disc
5 types of elements: solid, membrane, shell surface with rebars fluid	1 type of element $12 \times 12$ unsymmetric stiffness matrix 144 non-zero components	1 type of element $12 \times 12$ unsymmetric stiffness matrix 92 non-zero components	1 type of element $12 \times 12$ symmetrized stiffness matrix 52 non-zero components	1 type of element $6 \times 6$ symmetrized stiffness matrix 18 components
3747 elements	1 element	1 element	1 element	1 element
8797 dofs	12 dofs	12 dofs	12 dofs	6 dofs
136 s/iteration*	1 s/iteration*	1 s/iteration*	1 s/iteration*	1 s/iteration*

\* CPU time on P-IV, 2 GHz, 512 RAM

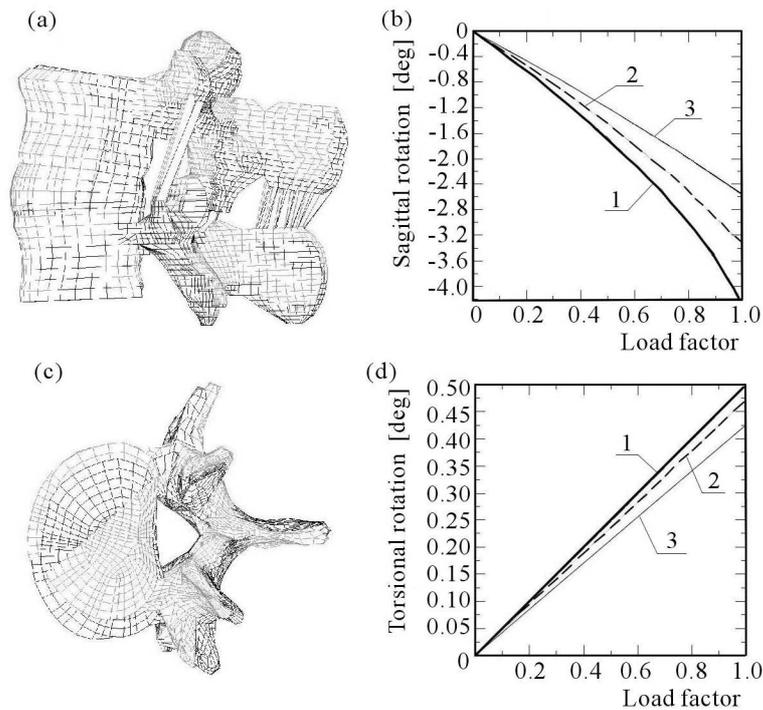


Fig. 8: Validation of different concepts of intervertebral disc simplifications: deformed shape (a) and rotation plot (b) for sagittal bending, deformed shape (c) and rotation plot (d) for axial torsion (1 – complex model of disc, 2 – connector element, 3 – all three others)

structures is the practical application of them, efficient methods of simulations in medical treatment and rehabilitation are needed. For such complex structures and numerical models, this efficiency requires many simplifications and hierarchical approaches. The complexity will still be quite significant, so we can suggest tools and methodologies to make them viable to be carried out.

In the case of dental implants, the use of CAXA elements allowed for a limited 3D analysis with the use of two dimensional geometric models. In the case of a full three dimensional model of an implant, the use of explicit codes enabled effectively simulation of the screw tightening. On the basis of the results of fatigue analysis, it can be claimed that the material fatigue is the basic reason of the observed complications.

For simulation of the motion segment, each step needs special techniques. The acquisition of geometry and the creation of the finite element model require the use of three different types of software and many special procedures to process geometric data. The modeling of soft and hard tissues needs different types of elements and material models. Preparation of a numerical model

of one motion segment and its analysis is so complex and needs so much time and computer resources that some simplifications are absolutely necessary. The idea of a simplified model of the intervertebral disc will enable mechanical analysis of much more complex models of human spine which can be helpful for simulation of surgery or analyses of spinal equilibrium and stability.

#### *Acknowledgements*

This research was financially supported the grant No. 3 T11F 026 28 within the budget resources for 2005 and 2006.

### References

1. 2005, *ABAQUS Analysis User's Manual*, ABAQUS, Inc., Pawtucket
2. 2005, *ABAQUS Theory Manual*, ABAQUS, Inc., Pawtucket
3. 2005, *fe-safe Works User's Manual*, Safe Technology Limited, Sheffield
4. ADAMS M.A., BOGDUK N., BURTON K., DOLAN P., 2002, *The Biomechanics of Back Pain*, Elsevier Science Limited, Churchill Livingstone
5. BĘDZIŃSKI R., 1997, *Biomechanika inżynierska*, Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław
6. BISHOP N.W.M., SHERRATT F., 2000, *Finite Element Based Fatigue Calculations*, NAFEMS, Glasgow
7. DIETRICH M., KĘDZIOR K., WITTEK A., ZAGRAJEK T., 1992, Non-linear finite element analysis of formation and treatment of intervertebral disc herniae, *Proc. Inst. Mech. Eng.*, 225-231
8. DRAPER J., 1999, *Modern Metal Fatigue Analysis*, HKS, Inc. Pawtucket
9. EBERLEIN R., HOLZAPFEL G.A., SCHULZE-BAUER C.A.J., 2002, Assessment of a spinal implant by means of advanced FE modeling of intact human intervertebral discs, *Fifth World Congress on Computational Mechanics*
10. GENG J.P., TAN K.B., LIU G.R., 2001, Application of finite element analysis in implant dentistry. A review of literature, *J. Prosthet. Dent.*, **85**, 585-598
11. GLEMA A., ŁODYGOWSKI A., KĄKOL W., WIERSZYCKI M., OGURKOWSKA M.B., 2004a, A simplified vertebrae disk model for computations of human lumbar spine segment, *Sixth World Congress on Computational Mechanics*
12. GLEMA A., ŁODYGOWSKI A., KĄKOL W., WIERSZYCKI M., OGURKOWSKA M.B., 2004b, Modeling of intervertebral discs in the numerical analysis of spinal segment, *ECCOMAS 2004*
13. HĘDZELEK W., ZAGALAK R., ŁODYGOWSKI T., WIERSZYCKI M., 2003, The effect of marginal bone loss and bone density on the risk of late implant components failures, *27th Annual Conf. of the European Prosthodontic Association*

14. KĄKOL W., ŁODYGOWSKI T., OGURKOWSKA M.B., WIERSZYCKI M., 2003a, Are we able to support medical diagnosis or rehabilitation of human vertebra by numerical simulation, *15th Int. Conf. on Computer Methods in Mechanics*
15. KĄKOL W., ŁODYGOWSKI T., OGURKOWSKA M.B., WIERSZYCKI M., 2003b, Numerical modeling of spinal motion segment, *ABAQUS User's Conf.*
16. KĄKOL W., ŁODYGOWSKI T., OGURKOWSKA M., WIERSZYCKI M., 2003c, The support of medical diagnosis or rehabilitation of the human vertebra by numerical simulation, *Acta of Bioengineering and Biomechanics*, **5**, 1, 235-241
17. KĄKOL W., ŁODYGOWSKI T., WIERSZYCKI M., 2003d, Estimate of tooth implant fatigue under cycling loading, *15th Int. Conf. on Computer Methods in Mechanics*
18. KĄKOL W., ŁODYGOWSKI T., WIERSZYCKI M., HĘDZELEK W., ZAGALAK R., 2002, Numerical analysis of the behavior of dental implant, *ABAQUS Users' Conference 2002*
19. KOCAŃDA S., 1985, *Zmęczeniowe pękanie metali*, Wydawnictwo Naukowo-Techniczne, Warszawa
20. MERZ B.R., HUNENBART S., BELSER U.C., 2000, Mechanics of the implant-abutment connection: an 8-degree taper compared to a butt joint connection, *The International Journal of Oral and Maxillofacial Implants*, **15**, 4, 519-526
21. OGURKOWSKA M.B., 1992, *Application of radiology and rheology method for mechanical testing of the vertebral column*, Doctoral Thesis. University School of Physical Education, Poznań
22. OGURKOWSKA M.B., RYCHLIK M., STANKIEWICZ W., NOWAK M., ROSZAK R., GLEMA A., WIERSZYCKI M., MORZYŃSKI M., ŁODYGOWSKI T., 2002a, The interaction of the L4-L5 spinal segments by FEA analysis. Part 1. Methods of geometrical data acquisition and validation, *Acta of Bioengineering and Biomechanics*, *13th Conference of the European Society of Biomechanics*, **4**, 1, 98
23. OGURKOWSKA M.B., RYCHLIK M., STANKIEWICZ W., NOWAK M., ROSZAK R., GLEMA A., WIERSZYCKI M., MORZYŃSKI M., ŁODYGOWSKI T., 2002b, The interaction of the L4-L5 spinal segments by FEM analysis. Part 2. Virtual modeling of the structure, *Acta of Bioengineering and Biomechanics*, *13th Conference of the European Society of Biomechanics*, **4**, 1, 99
24. SAKAGUCHI R.L., BORGERSEN S.E., 1993, Nonlinear finite element contact analysis of dental implant components, *The International Journal of Oral and Maxillofacial Implants*, **7**, 1, 655-661
25. SKAGGS D.L., WEIDENBAUM M., 1994, Regional variation in tensile properties and biomechanics composition of the human lumbar annulus fibrous, *Spine*, **9**, 120-134
26. SWARTZ D.E., WITTENBERG R.H., 1991, Physical and mechanical properties of calf lumbosacral trabecular bone, *J. Biomechanics*, **24**, 11, 1059-1068

27. RYCHLIK M., MORZYŃSKI M., NOWAK M., STANKIEWICZ W., ŁODYGOWSKI T., OGURKOWSKA M.B., 2004, Acquisition and transformation of biomedical objects to CAD systems, *Strojnicka Casopis*, **3**, 4, 121-135

### Numeryczna złożoność wybranych problemów biomechaniki

#### Streszczenie

Praca opisuje wybrane aspekty modelowania numerycznego z wykorzystaniem Metody Elementów Skończonych (MES) zagadnień biomechaniki. Autorzy starają się podkreślić złożoność takiej analizy MES. Uwagę skupiono na dwóch przykładach: analizie wszczepu stomatologicznego oraz ruchomości segmentu kręgosłupa ludzkiego (L4-L5). W obu analizowanych i modelowanych przypadkach odtworzenie geometrii, liczba typów elementów i stopni swobody modeli, przyjęcie modeli i właściwości konstytutywnych materiałów biologicznych, definicje problemów kontaktowych czy warunków początkowo-brzegowych stanowią o poziomie trudności podejmowanych zadań. Pokonanie wymienionych trudności, a co za tym idzie zbudowanie modeli numerycznych odtwarzających zachowanie się rzeczywistych elementów biomechanicznych jest przedmiotem podejmowanej w pracy dyskusji. Pomimo tej złożoności zadań, zastosowanie MES do oceny zachowania się implantów bądź segmentu ruchowego prowadzi do wystarczająco zgodnej z eksperymentem oceny trwałości implantów lub sztywności segmentów kręgosłupa.

*Manuscript received February 2, 2006; accepted for print May 5, 2006*