# OPTIMIZED FE MESH GENERATION PROCEDURE BASED ON A USER-DEFINED SPATIAL REFINEMENT GRADIENT. APPLICATION TO A MOTION SEGMENT

**S. Cortez[1], M. Pinheiro[2], J. C. Pimenta Claro[3] and J. L. Alves[3]**

## 1. ABSTRACT

Finite Element (FE) meshes are usually highly refined and dense and, consequently, computationally very expensive. Therefore, after the preliminary FE mesh generation, it is necessary to decrease its size by diminishing the total number of nodes and elements while maintaining both geometrical accuracy and a physically-meaningful FE mesh refinement. The aim of this work is to describe an optimized FE mesh simplification procedure based on edge contraction and on a user-defined spatial refinement gradient criterion. The main idea is that, for normal mechanical loadings applied to a motion segment, the vertebrae shall behave almost as an incompressible medium, and only the intervertebral disc (IVD) should undergo relevant strains. In this case, it is acceptable (and desirable) to attain a problem-functional FE mesh, in which the FE mesh should be more refined at the IVD and coarser at vertebrae. On the other hand, an optimized FE mesh should be more refined at the annulus fibrosus than at the nucleus pulposus. In summary, the proposed FE mesh generation and simplification procedure, being based on a user-definable spatial FE mesh refinement gradient, allows the user to define precisely the required refinement and thus to reduce drastically the number of elements in the final FE mesh and consequently to diminish the computation time required for the FE analysis, while keeping the necessary physical meaning on the FE mesh. The aforesaid procedure will be applied to the FE mesh generation (based on [1]) of a motion segment initially characterized by medical imaging.

## 2. INTRODUCTION

In general, the starting point for the 3D geometrical modelling by finite elements of an anatomical structure is the generation of a 3D voxel-based geometrical model, obtained after denoising, smoothing and segmentation of a set of 2D medical images. After the 3D voxelized model has been defined, a specific tetrahedral FE meshing procedure is applied and, generally, a too dense and highly refined FE mesh is obtained. This FE mesh is computationally very expensive and one needs to decrease its size appling a FE mesh simplification procedure. This procedure consists in reducing the number of tetrahedra in the initial FE mesh while keeping the overall shape, volume and boundaries preserved. Generally, after this procedure, the smaller elements are located at the internal and external boundaries, while larger elements are located inside the FE mesh. However, this is not always acceptable. There may be situations where this accuracy may be required simultaneously in structures outside and inside the FE mesh.

[1] Researcher, Mechanical Engineering Department, University of Minho, 4800-058 Guimarães, Portugal (saracoutocortez@gmail.com)
[2] PhD Candidate, Mechanical Engineering Department, University of Minho, 4800-058 Guimarães, Portugal (manuelspinheiro@gmail.com)
[3] CT2M - Centre for Mechanical and Materials Technologies, University of Minho, 4800-058 Guimarães, Portugal.

For many applications, the need for accuracy is greater near the surface of the domain, such as for studies of the IVD. In a motion segment, the FE mesh should be more refined at the IVD and coarser at the vertebrae (nearly incompressible medium). On the other hand, since the annulus fibrosus (AF) is a stiff ring-shaped structure made up of concentric lamellae [2], an optimized FE mesh should be more refined at the annulus fibrosus than at the nucleus pulposus (NP). Another point is that these concentric lamellae are not uniformly defined in the annulus and it is necessary to develop a refinement gradient. In this case, the elements should be smaller in the outer annulus (where lamellae are denser and combined) than the ones in the inner annulus (less dense lamellae) (Fig. 1).

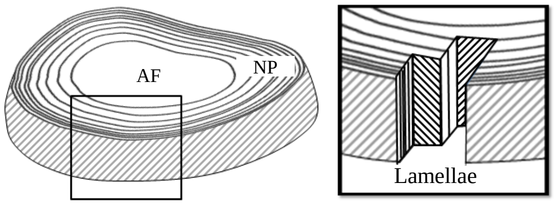



*Fig. 1 Illustration of the intervertebral disc and the lamellar structure in the annulus.*

## 3. METHODOLOGY

From the initial FE mesh generation procedure, a dense tetrahedral FE mesh of a Human lumbar motion segment (Fig. 2a) with 465 644 elements and 86 181 nodes was obtained. Three different domains are contained within this mesh: vertebrae (green), nucleus pulposus (blue) and annulus fibrosus (pink). The average element volume size is of 0.04 mm$^3$, while edge sizes range from 0.3 to 1.1 mm.

The paramount aim of this work is to simplify the previous mesh in such a way that a user-defined gradient can be achieved, i.e., one needs to define, in the FE mesh, a user-defined edge sizing field to drive the FE mesh simplification procedure. This so-called user-defined edge sizing field was implemented by the definition of a sizing field, in which the maximum size allowed to all edges sharing a node must be defined by the user.

Assuming the aforesaid FE mesh as an input file, a FORTRAN code was developed to generate a new FE mesh with a user-defined FE mesh refinement in the IVD. An optimized FE mesh shall be anatomically-based. The FE mesh must be more refined on the outside layers of the AF and have a progressive coarsening refinement radially until the inner layers of the AF. Finally, having in mind the biomechanical behaviour of the NP (mostly their incompressible behaviour), the FE mesh refinement of the NP is less relevant, and thus shall be as coarser as possible, as well vertebrae, which behave nearly incompressibility, for the anatomically normal loadings. The implemented FE mesh simplification procedure has two basic steps. The first one is the application of a user-definable sizing field, with a special emphasis on the AF. To define a user-definable refinement gradient within the annulus, and in order to mimic the radial evolution of the thickness of the fibres' layers, one needs to determine, firstly the nodes belonging to the annulus, and then their radial position within the annulus. So, each tetrahedron is classified as belong or not to the annulus, and then we select the annulus' outside boundary and each facet is classified between UP, DOWN, IN or OUT. The angle $\theta$ (see Fig. 2b) between the axial axis (which crosses the annulus centre of mass) and the outside normal of each facet is determined and contributes to the classification of the facet: the facet is UP and DOWN if $\theta$<60° or $\theta$>120°, and IN or OUT otherwise; secondly, in the facet is IN or OUT, angle $\alpha$, defined by the angle between the normal to the element and the vector (blue line in the Fig. 2b) that goes through the centre of the facet and it is perpendicular to the axial axis. Finally, if $\alpha < 90^{\circ}$, the facet is classified as IN, and OUT otherwise. Using a linear interpolation, the gradient edge size is calculated based on the input values (maximum edge sizes) defined at each node, for elements near the nucleus (IN region, green line in the Fig. 2b) to the periphery of the annulus (OUT region, orange line in the Fig. 2b). The default value is used to define the maximum edge size for other domains (vertebrae and nucleus). The sizing field can be absolute or relative, but in this study, only the absolute sizing field simplification was performed.

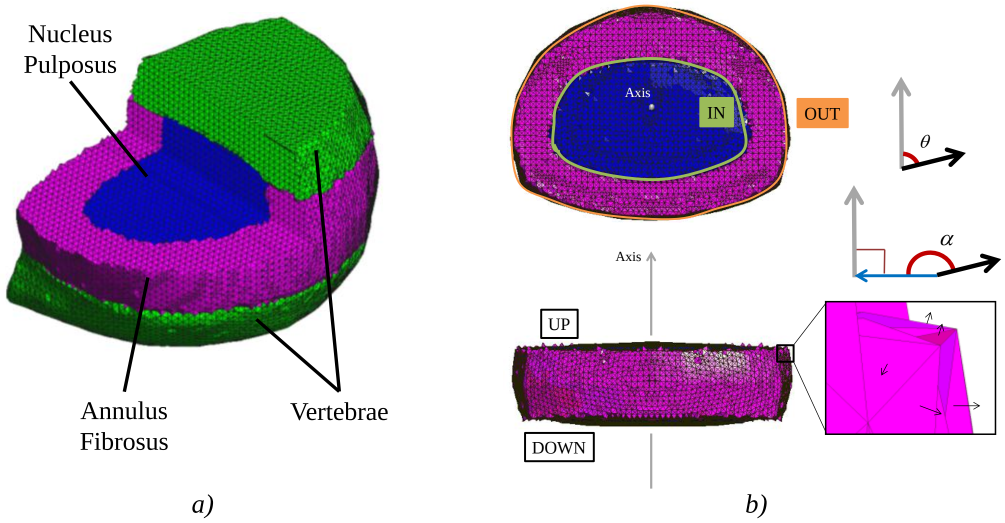



*Fig. 2 a) The initial FE mesh of a motion segment and b) the schematic of the inner and outer regions of the annulus fibrosus.*

Two different tests for FE meshes were performed and the values used for the absolute edge sizing field are presented in Table 1.

*Table 1 Different input parameters for two simplification tests.*

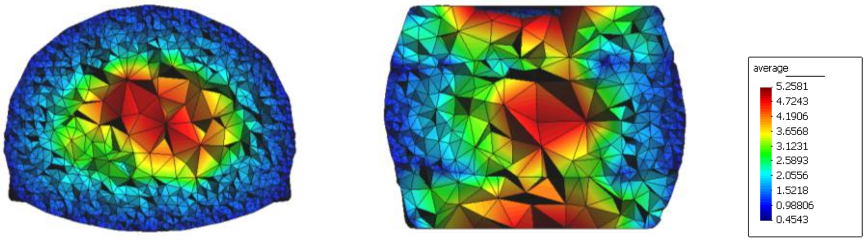
| **Test** | **Maximum Sizing Edge OUT** [mm] | **Maximum Sizing Edge IN** [mm] | **Maximum Sizing Edge (Default)** [mm] |
|---|---|---|---|
| 1 | 1.5 | 3.0 | 20 |
| 2 | 2.0 | 4.0 | 20 |

After the conversion of the FE mesh into the VTK format [www.vtk.org] and the definition of a sizing field, the second stage was performed. This VTK file was used as input file for the simplification process. The FE mesh simplification procedure applied is based on a step by step edge contraction algorithm, which can generate finite elements with a better geometrical shape, decreasing locally the FE mesh refinement (excepted in annulus fibrosus). In this method, using a cost function, local simplification operators are applied iteratively, which contract the edges of the elements taking into account the elements' shape quality and their geometric error. The optimum location of the new node depends primarily on the angles formed by adjacent elements when the edge is contracted. Sometimes, the contraction of some edges is not allowed in order to avoid violating the initial geometry of the boundaries between FE mesh domains.

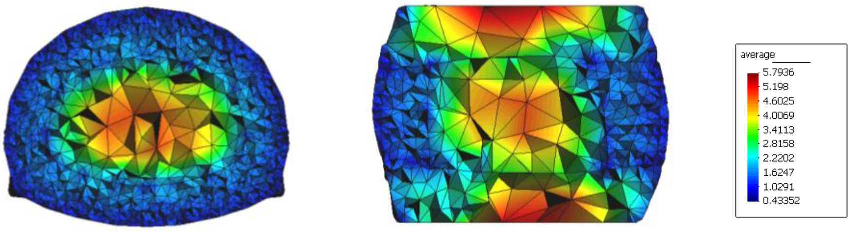
However, the geometrical constrain linked with the internal boundary between annulus and nucleus can be relaxed, even because the definition of this transition is, in the reality, difficult to be defined. In order to understand the role of this boundary, two types of results were obtained after simplification: with and without the preservation of an accurate geometrical definition of the internal boundary between AF and NP.

## 4. RESULTS

In order to compare and evaluate the geometrical accuracy and the influence of different edges sizes applied in the FE meshes, axial and sagittal cross sections were realized. Fig. 3a shows the FE mesh (with a non-accurate boundary between NP and AF) obtained in the simplification test 1 and Fig. 3b shows the FE mesh without a non-accurate boundary between NP and AF obtained with the same simplification test.



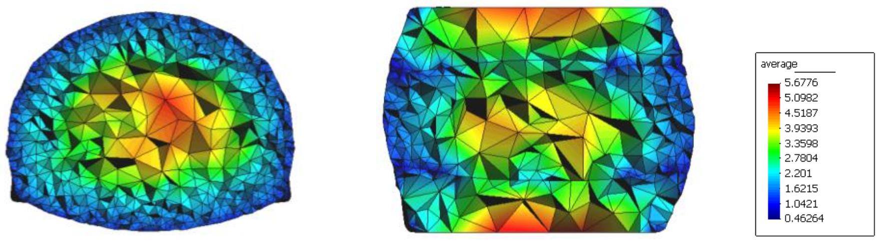

*a)*



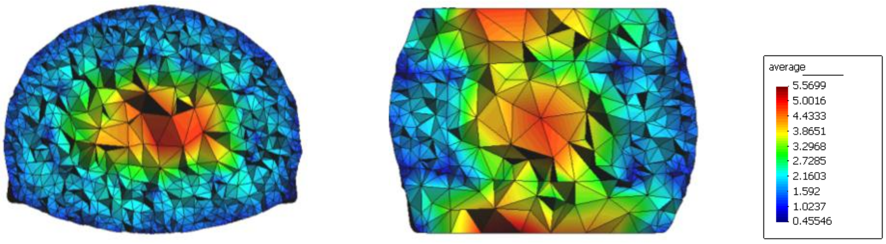

*b)*

*Fig. 3 Axial (left) and sagittal (right) cross sections of the FE meshes obtained by the simplification test 1 a) with and b) without a non-accurate boundary between NP and AF.*

FE meshes, with and without a non-accurate boundary between NP and AF, obtained by the simplification test 2 are shown in Fig. 4a and Fig. 4b, respectively.




*a)*




*b)*

*Fig. 4 Axial (left) and sagittal (right) cross sections of the FE meshes obtained by the simplification test 2 a) with and b) without a non- accurate boundary between NP and AF.*

Table 2 shows the number of elements and nodes of the initial FE mesh and the FE meshes obtained with the developed algorithm.

*Table 2 Number of elements and nodes of the initial and FE meshes obtained.*

| | **Initial Mesh** | **FE mesh with simplification test 1** | | **FE mesh with simplification test 2** | |
|---|---|---|---|---|---|
| | | **non-accurate AF/NP boundary** | **accurate AF/NP boundary** | **non-accurate AF/NP boundary** | **accurate AF/NP boundary** |
| **Number of Nodes** | 86 181 | 16 966 | 18 129 | 13 652 | 14 782 |
| **Number of FElements** | 465 644 | 85 381 | 92 018 | 65 807 | 72 183 |
| **Number of FElements (AF)** | 132 675 | - | 51 997 | - | 32 871 |
| **% FElements of the AF on the FE mesh** | 29% | - | 57% | - | 46% |

## 5. DISCUSSION

The results presented a noticeable difference in the overall elements size between the annulus and the other regions of the motion segment. The geometrical accuracy is well visible in the axial and sagittal cross sections of the FE meshes. The vertebrae surfaces and the nucleus central region are largely simplified, with elements edge sizes considerably larger than the annulus region. The refinement of the annulus does not present a well-defined gradient in all cases. For low edge sizes, there is an agglomerate of smaller elements in the outer annulus region, which for high values is not observed. To describe the geometry of the boundary between annulus and nucleus, elements could not be simplified which is why they present elements with the same size both in the inner and the outer annulus region. The same happens between the disc and the vertebrae. This is well visible in the FE meshes obtained with an accurate boundary between NP and AF. In the case of FE meshes with a non-accurate boundary between AF and NP, the refinement is more gradual because the elements are not required to describe any boundary. This boundary restriction also influences the maximum edge size for regions without interest, such as the vertebrae. All FE meshes showed edge sizes between 0.4 and 5 mm, approximately, and the maximum value (default) imposed for edge size was never reached because elements would be very thin (bad elements) and the FE mesh domain is too small to have elements with a large size.

However, by a simple analysis of these figures, it is not possible to conclude anything concerning the number of elements of the FE mesh. For that, Table 2 presents a resume of the number of elements and nodes of the final simplified FE meshes. The FE meshes obtained by the two tests studied present a small number of nodes and elements comparing with the initial and dense FE mesh. Regarding the number of elements in the annulus, the small size of the elements resulted in an increased percentage of the number in relation to the total number of elements in the initial mesh. In the FE mesh obtained by the first test, the number of elements in the annulus represents about 57% of the total elements. This shows that more than half of the elements are present on the annulus, which in the initial FE mesh only represented 29%. In the case of the FE mesh obtained from the second simplification, the number of elements in the annulus represents about 46% of the total number in the mesh.

## 6. CONCLUSION

The examples of application of the proposed FE mesh simplification procedure allow to demonstrate our ability to completely master the simplification process by defining a non-uniform and user-definable sizing field, and thus to obtain a user-defined FE mesh refinement gradient within all FE domain. The definition of a higher FE mesh refinement in regions of interest (like the annulus fibrosus) can be easily obtained using the proposed FE mesh simplification algorithm. Essentially, an absolute sizing field allows to manipulate edge sizes, and indirectly FE volume sizes, in different regions/structures of the global FE mesh. It is a powerful tool to decrease the size (number of nodes and number of elements) of a FE meshes, keeping geometrical accuracy and even improving FE mesh quality.

## 7. ACKNOWLEDGMENTS

The authors gratefully acknowledge the support of the European Project: NP Mimetic - Biomimetic Nano-Fiber Based Nucleus Pulposus Regeneration for the Treatment of Degenerative Disc Disease, funded by the European Commission under FP7 (grant NMP3-SL-2010-246351).

## 8. REFERENCES

1. Labelle, F. and Shewchuk, J. R., Isosurface stuffing: Fast tetrahedral meshes with good dihedral angles, ACM Transactions on Graphics (TOG), 2007, Vol. 26(3), 57.1-57.10
2. Holzapfel, G. A., Schulze-Bauer, C. A. J., Feigl, G. and Regitnig, P., Single lamellar mechanics of the humam lumbar annulus fibrosus, Biomechanics and Modeling in Mechanobiology, 2005, Vol. 3(3), 125-40