Development of the software tool for generation and visualization of the finite element head model with bone conduction sounds

Cite as: AIP Conference Proceedings **1703**, 060017 (2015); https://doi.org/10.1063/1.4939372
Published Online: 15 January 2016

Dalibor Nikolić, Žarko Milošević, Igor Saveljić, and Nenad Filipović

Development of the Software Tool for Generation and Visualization of the Finite Element Head Model with Bone Conduction Sounds

Dalibor Nikolić*, Žarko Milošević*, Igor Saveljić* and Nenad Filipović*,†

*Bioengineering Research and Development Center – BioIRC, Kragujevac, Serbia
†Faculty of Engineering, University of Kragujevac, Kragujevac, Serbia

Abstract. Vibration of the skull causes a hearing sensation. We call it Bone Conduction (BC) sound. There are several investigations about transmission properties of bone conducted sound. The aim of this study was to develop a software tool for easy generation of the finite element (FE) model of the human head with different materials based on human head anatomy and to calculate sound conduction through the head. Developed software tool generates a model in a few steps. The first step is to do segmentation of CT medical images (DICOM) and to generate a surface mesh files (STL). Each STL file presents a different layer of human head with different material properties (brain, CSF, different layers of the skull bone, skin, etc.). The next steps are to make tetrahedral mesh from obtained STL files, to define FE model boundary conditions and to solve FE equations. This tool uses PAK solver, which is the open source software implemented in SIFEM FP7 project, for calculations of the head vibration. Purpose of this tool is to show impact of the bone conduction sound of the head on the hearing system and to estimate matching of obtained results with experimental measurements.

INTRODUCTION

Most of the (BC) studies focused on the mechanical properties and vibration transmission characteristics of the human cranial vault. In the study by Stenfelt et al. 2000 [4] there is measured skull vibration in three perpendicular directions close to the cochlea with stimulation at the ipsilateral and contralateral mastoid as well as at the forehead. But in this study only one dry human skull was used. Study design by [5] there is measurements of the cochlear promontory vibration in three perpendicular directions, using stimulation at numerous positions on the skull in the intact human heads. First goal of our software was to repeat the results obtained in this study using FE analysis, and then to analyze the same problem with several different material models

MODEL

Basically the process of solving a Finite Element (FE) model consists the three different phases: Preprocessing, Solving and Post processing phases. (Fig. 1).

Preprocessing Phase - Geometry

Input geometry for FE model is generated from DICOM images obtain from multiple slices CT scanner. Also on easy way with little modification software can work with MRI DICOM images. Firstly, we prepared the STL files (Fig. 2) for 3D meshing. Each STL file presents a different layer of human head with different material properties (brain, CSF, different layers of the skull bone, skin, etc.). In the next step from geometry files we created boundary files (Fig. 3):

- FORCE STL file represent position of input (vibration) sound – position of sound source
- BOUNDARY STL file present location of constrain - contact with the neck

Mechanics of Hearing: Protein to Perception
AIP Conf. Proc. 1703, 060017-1–060017-5; doi: 10.1063/1.4939372

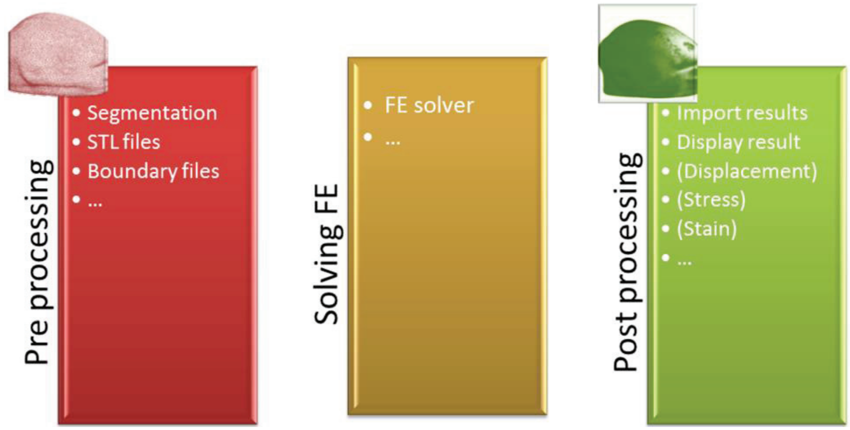

FIGURE 1. [Color version of figure available online] Schematic structure of FE model.

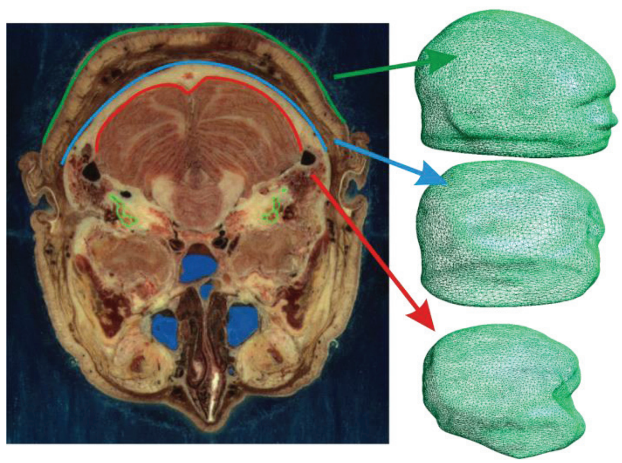

FIGURE 2. [Color version of figure available online] STL files generated by manually segmentation.

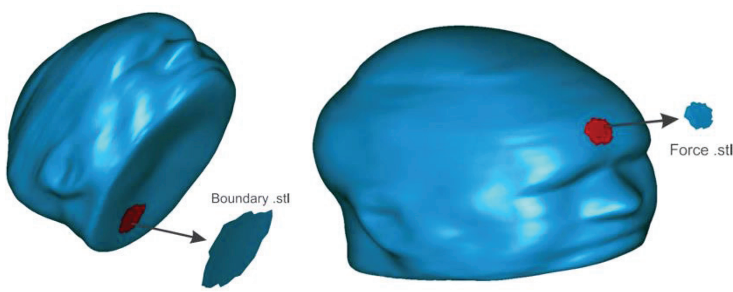

FIGURE 3. [Color version of figure available online] BOUNDARY STL file- FE constrains; FORCE STL file - position of input (vibration) sound.

Mesh Generation

After the surface mesh was generated, a 3D meshing was employed (Fig. 4). In order to have solution stability and accuracy in our finite element (FE) solver, we used 3D 4-node finite element for analysis. This procedure of the FE mesh generation was performed by a Tetgen algorithms [1] to create tetrahedrons from the surface triangles.

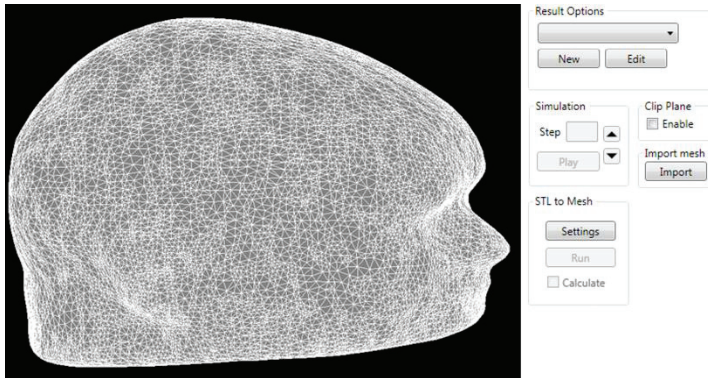

FIGURE 4. Tetrahedral mesh model of head.

Finite element formulation

We used linear tetrahedron finite element (Fig. 5) where displacement field over the tetrahedron element is defined by the three components u_x, u_y and u_z. These displacements are linearly interpolated over the element from their nodal values

$$\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} u_{11} & u_{12} & u_{13} & u_{14} \\ u_{21} & u_{22} & u_{23} & u_{24} \\ u_{31} & u_{32} & u_{33} & u_{34} \end{bmatrix} \begin{bmatrix} N_1 \\ N_2 \\ N_3 \\ N_4 \end{bmatrix} \tag{1}$$

where N_1, N_2, N_3, N_4 are the interpolation functions which are simply the tetrahedral coordinates; and $u_{11},\ldots,u_{34}$ are the nodal displacements.

The internal virtual work can be expressed as [2]

$$\delta W^{\text{int}} = \int_V \delta \mathbf{e}^T \boldsymbol{\sigma} dV = \delta \mathbf{U}^T \int_V \mathbf{B}^T \mathbf{C} \mathbf{B} dV \mathbf{U} = \delta \mathbf{U}^T \mathbf{K} \mathbf{U} \tag{2}$$

where we have employed the relation for strain components:

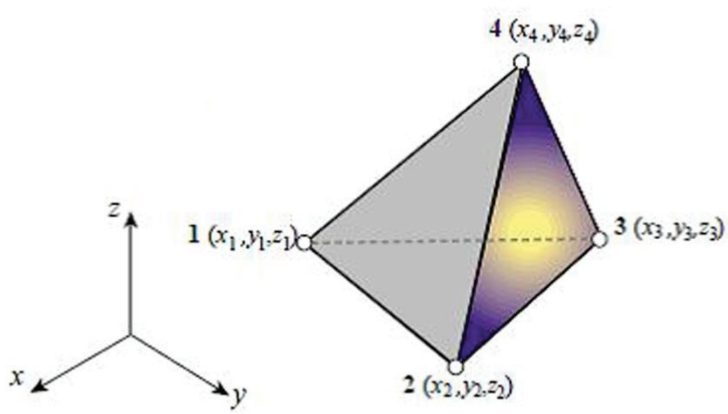

FIGURE 5. [Color version of figure available online] The linear tetrahedron finite element.

$$\mathbf{e}=\begin{Bmatrix} e_{xx} \\ e_{yy} \\ e_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}=\begin{Bmatrix} u_{1,1} \\ u_{2,2} \\ u_{3,3} \\ u_{1,2}+u_{2,1} \\ u_{2,3}+u_{3,2} \\ u_{1,3}+u_{3,1} \end{Bmatrix}=\begin{bmatrix} N_{1,1} & 0 & 0 & \ldots & N_{N,1} & 0 & 0 \\ 0 & N_{1,2} & 0 & \ldots & 0 & N_{N,2} & 0 \\ 0 & 0 & N_{1,3} & \ldots & 0 & 0 & N_{N,3} \\ N_{1,2} & N_{1,1} & 0 & \ldots & N_{N,2} & N_{N,1} & 0 \\ 0 & N_{1,3} & N_{1,2} & \ldots & 0 & N_{N,3} & N_{N,2} \\ N_{1,3} & 0 & N_{1,1} & \ldots & N_{N,3} & 0 & N_{N,1} \end{bmatrix}\begin{Bmatrix} U_1^1 \\ U_2^1 \\ U_3^1 \\ \vdots \\ U_1^N \\ U_2^N \\ U_3^N \end{Bmatrix}=\mathbf{BU} \tag{3}$$

from which $\delta\mathbf{e}^T = \delta\mathbf{U}^T\mathbf{B}^T$, and the constitutive relationship $\boldsymbol{\sigma} = \mathbf{Ce}$; here, **e** is the strain (used here in the form of the engineering strain vector), **U** is the vector of nodal displacements**,** **B** is the strain-displacement relation matrix, and **C** the material constitutive matrix. Clearly, the stiffness matrix **K** is

$$\mathbf{K}=\int_V \mathbf{B}^T\mathbf{CB}dV \tag{4}$$

and the element internal force $\mathbf{F}^{\text{int}}$ is given by the expression $\mathbf{F}^{\text{int}} = \mathbf{KU}$. The stiffness matrix is symmetric and has dimensions $3N$x$3N$ (in our case 12x12) and the force vector $\mathbf{F}^{\text{int}}$ is of size $3N$, $\mathbf{F}^{\text{int}}\left(F_x^{(\text{int})1}, F_y^{(\text{int})1}, F_z^{(\text{int})1}, \ldots\ldots, F_x^{(\text{int})N}, F_y^{(\text{int})N}, F_z^{(\text{int})N}\right)$.

The external nodal forces resulting from the pressure on an element surface are calculated by employing again the equivalence of virtual work. A simple approximation for the 4-node tetrahedron element is to calculate the total force as $F_p = pA$ (where p is the mean pressure and A is the area of the element side) and use F_p / 3 at each node in the normal surface direction.

RESULTS

To create a user-friendly presentation of the results, we developed a set of post-processing features in our software tools. Originally our PAK [3] solver makes output UNV file format, and we used an in-home software for 3D drawings based on the Open GL library. In one FE model calculation 50 GB of data were written in a number of UNV files (usually one UNV file per time step). A problem with this UNV file is that all physical quantities are written in one file which can slow visualization. Software can visualize different physical quantity very easy. We created specific feature for easy cross-sections (user can saw inside of the FE model). The distribution of the displacements has been shown in the Fig. 6 and displacements at cross-section are presented in the Fig. 7. Also, for this cases Von Mises Stress are shown in the Fig. 8.

CONCLUSIONS

Finite element analysis has been used extensively to predict the biomechanical performance of hearing instruments. The principal difficulty in simulating the mechanical behavior of bone conduction is generating accurate models of the living human bone tissue and its response to applied mechanical forces. This research has been conducted to the comparison of the biomechanical stresses formed in the cochlea for different positions of the sound sources.

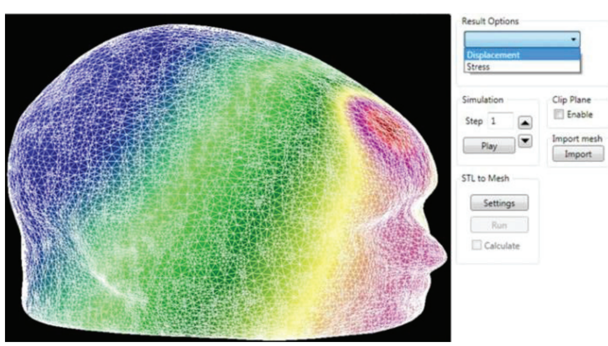

FIGURE 6. [Color version of figure available online] Displacement distribution.

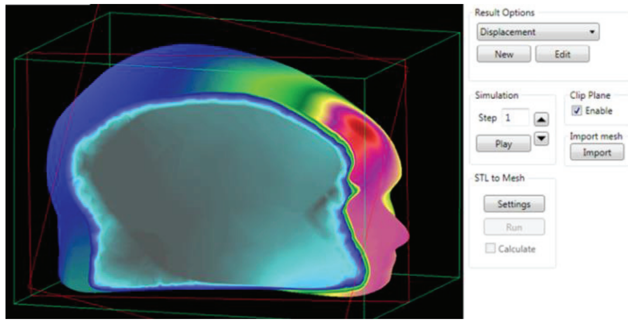

FIGURE 7. [Color version of figure available online] Displacement distribution - Cross section.

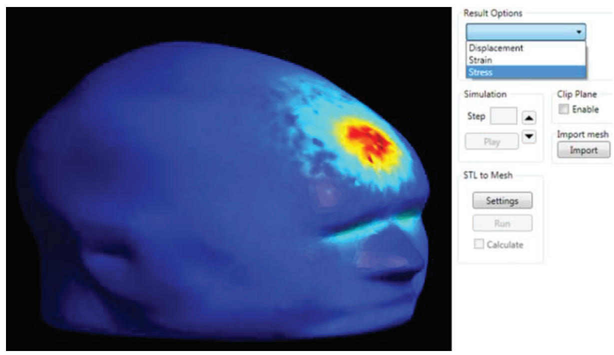

FIGURE 8. [Color version of figure available online] Effective stress distribution.

ACKNOWLEDGMENTS

This work is supported by the European Union under Grant FP7 No. 600933, SIFEM project.

REFERENCES

[1] Hang S (2007) Tetgen: A quality tetrahedral mesh generator and three-dimensional delaunay triangulator, Research Group: Numerical Mathematics and Scientific Computing Weierstrass Institute for Applied Analysis and Stochastics Mohrenstr. 39, 10117 Berlin, Germany. http://tetgen.berlios.de/

[2] Kojic M, Filipovic N, Stojanovic B, Kojic N (2008) Computer Modeling in Bioengineering: Theoretical Background, Examples and Software. New Jersey: J. Wiley & Sons

[3] Kojic M, Slavkovic R, Zivkovic M, Grujovic N, Filipovic N (1998) PAK - Program for finite element analysis of construction, Faculty of Mechanical Engineering, University of Kragujevac, Kragujevac

[4] Stenfelt S, Håkansson B, Tjellström A (2000) Vibration characteristics of bone conducted sound in vitro. J Acoust Soc Am 107: 422– 431

[5] Stenfelt S, Goode RL (2005) Transmission properties of bone conducted sound: Measurements in cadaver heads. J Acoust Soc Am 2118:2373–91

COMMENTS AND DISCUSSION

Jeremie Guignard: Interesting work. In figure 7, in which I suspect the displacement is coded from low (blue) to high (red), it seems like the displacement is propagating mostly in the external layer, while the lower structures show no/little displacement. Can you comment on that? Does this happen at various frequencies of the stimulus?