# AUTOMATIC 3D FINITE ELEMENT MESH GENERATION : DATA FITTING FROM AN ATLAS

Luboz Vincent*, Payan Yohan*, Couteau Beatrice**, Swider Pascal**

## 1. ABSTRACT:

Finite Element (FE) analysis starts to be largely used in the field of computer assisted surgery, from bones model in orthopaedic surgery to facial tissues modelling in craniofacial surgery. In those frameworks, models have to be rebuilt and adapted to each patient, which means the use of automatic 3D mesh generators. In practice, to maintain a strong patient-oriented specificity, such automatic generators often make some compromises in terms of homogeneity, symmetry or mesh refinement. For those reasons, a new mesh generating technique, namely the Mesh-Matching algorithm, has been introduced. Five steps need to be driven to generate a patient FE model: (1) the atlas generation, (2) the segmentation of the patient data, (3) the automatic computation (following a matching process) of the transformation T between the atlas and the patient data, (4) the application of T to the atlas to generate a patient mesh and (5) the regularization of the patient mesh to allow FE analysis. This paper presents a first evaluation of the Mesh Matching algorithm, validated on femur and face 3D FE meshes.

## 2. INTRODUCTION

In biomechanical studies, Finite Element (FE) customized meshes start to be more and more interesting since they can integrate both geometry and mechanical properties of the patient. In commercial products, automatic 3D meshing methods are frequently based on the Delaunay criterion [1] followed by the advancing front technique [2]. Those techniques produce tetrahedral meshes, which are less efficient than hexahedral meshes [3]. So far, common software are not able to perform hexahedral meshes on complex structures, but many scientists focus on that problem (see [4] for a review).

Keywords: Finite Element Modelling, Automatic Meshing, Mesh Regularization

* Laboratoire TIMC/IMAG, UMR CNRS 5525, Faculté de Médecine Domaine de la Merci, 38706 La Tronche, France
** INSERM, Biomécanique, C.H.U. Purpan, BP 3103, 31026 Toulouse cedex 3, France

In a previous paper [5], the Mesh Matching (M-M) algorithm was suggested to automatically generate customized hexahedrons and wedges 3D meshes of proximal femora from an existing 3D mesh. Though this method has been validated for this structure, the application to a more complex model, namely a FE model of the human face [6], showed mesh irregularities that make the mechanical simulation impossible.

To eliminate those irregularities, a regularization step has been developed and applied to the mesh generated by the M-M. Many regularization techniques are based on a reconnection method [7] or Laplacian smoothing. The method used in this study is based on a node adjustment, respecting the connectivity. This technique uses the jacobian determinant and can thus be applied to every elements types.

In this paper, the Mesh Matching algorithm and the regularization technique will be present in the materials and method part. Then, results on femora and face meshes will be shown and discussed.

## 3. MATERIALS AND METHOD

### 3.1. M-M algorithm application

The first step of our FE mesh generation method is the application of the M-M algorithm to an atlas FE mesh. The matching algorithm was originally proposed for applications in computer-assisted surgery [8]. It was recently applied to FE analysis to transform an existing FE bone model in another FE mesh adapted to a new patient geometry [5].

The different steps of the M-M algorithm are :

1. The definition of the mesh which is called the "generic" mesh for the studied anatomical structure. This mesh is also called the atlas since every meshes produce using the M-M will be derived from this one. This FE atlas is manually designed to provide a good starting point (in terms of mesh structure, homogeneity, refinement) for the M-M.
2. A set of 3D points, located on the external surface of the patient structure, is extracted from CT or MRI data using an automatic contours segmentation process.
3. The M-M algorithm is then applied to compute the elastic transformation **T** that match the nodes located on the external surface of the atlas FE model with the 3D set of patient data points previously segmented. This transformation is the combination of a rigid body transform *RT*, a global warping *W* and a local displacement function *S* built on an octree displacement grid.

$$\mathbf{T}_p = RT o W o S \quad (1)$$

where $p$ is a vector gathering 6 parameters that define *RT*, the 12 to 30 parameters that define *W* and the thousands of local displacement vectors that define *S*. Let $M = \{M_i,\ i = 1 \ldots N_1\}$ and $P = \{P_i,\ i = 1 \ldots N_1\}$ be the sets of atlas and patient features, obtained by the previous segmentation step. The elastic registration algorithm minimizes a least square criterion *E(p),* given by :

$$E(p)=\sum_{i=1}^{N_1}\frac{1}{\sigma_i^2}\left[dist(P,T_p(M_i))\right]^2+R_p \quad (2)$$

where *R* defines a regularization term which is applied to *S* in order to obtain a smooth displacement function, $\sigma_i$ is the variance of the noise of the measurement *i* and *dist* is the distance between the set $P$ and a point $M_i'$ ($M_i$ transformed by **T**). The optimisation of $E(p)$ is performed by using the Levenberg-Marquardt algorithm.

4. The FE patient mesh is automatically generated by applying the transformation **T** to every nodes of the FE atlas mesh. The mesh thus obtained is adapted to the patient geometry with a topology similar to the atlas (same number and types of elements).

At the end of these four steps, a FE mesh is obtained. It has been shown, for proximal femora [5] and entire femora [9] that these resulting meshes allowed satisfying FE analysis. But for a more complex mesh, namely a FE model of the human face, the M-M algorithm provides some irregular elements. The regularization step, presented in the next sub-part, proposes to correct those irregular elements to allow Finite Element Analysis.

### 3.2. Regularization of the mesh

The second step of our mesh generation method is the correction of the irregular elements that may be produced by the M-M algorithm.

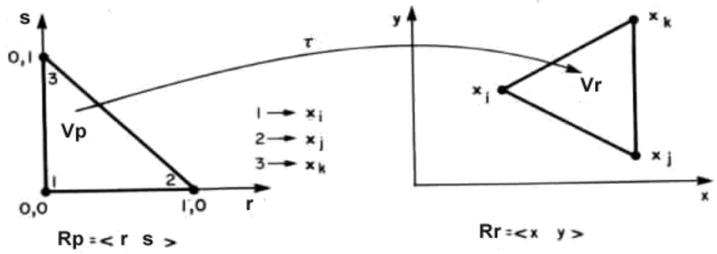


Fig. 1 : Transformation between the reference space, with a reference triangle, and the actual space, with an actual triangle. If this jacobian matrix of this transformation is not singular, FE analysis is allowed.

The notion of regularity for FE meshes is function of the jacobian determinant computed from the transformation between the reference space and the actual space (Fig. 1). The jacobian **J** of this transformation (for more explanation, see [10]) can be written as :

$$\mathbf{J} = \begin{bmatrix} \mathbf{J}_{\mathbf{xx}} & \mathbf{J}_{\mathbf{xy}} & \mathbf{J}_{\mathbf{xz}} \\ \mathbf{J}_{\mathbf{yx}} & \mathbf{J}_{\mathbf{yy}} & \mathbf{J}_{\mathbf{yz}} \\ \mathbf{J}_{\mathbf{zx}} & \mathbf{J}_{\mathbf{zy}} & \mathbf{J}_{\mathbf{zz}} \end{bmatrix} = \begin{bmatrix} \sum_i \frac{\partial N_i}{\partial r} x_i & \sum_i \frac{\partial N_i}{\partial r} y_i & \sum_i \frac{\partial N_i}{\partial r} z_i \\ \sum_i \frac{\partial N_i}{\partial s} x_i & \sum_i \frac{\partial N_i}{\partial s} y_i & \sum_i \frac{\partial N_i}{\partial s} z_i \\ \sum_i \frac{\partial N_i}{\partial t} x_i & \sum_i \frac{\partial N_i}{\partial t} y_i & \sum_i \frac{\partial N_i}{\partial t} z_i \end{bmatrix} \quad (3)$$

where $N_i$ are the interpolation shape functions, ($x_i$, $y_i$, $z_i$) the actual coordinates of the element and ($r$, $s$, $t$) the coordinate system of the reference space.
FE analysis is possible if **T** exists for each element of the mesh, which is equivalent to say that the FE simulation is available if the jacobian matrix is not singular for each element. This singularity is pointed out by the sign of the determinant of **J**. If this determinant **detJ** is negative or null for a node of an element, there is an irregularity in this element. Given this criterion; every irregular elements can be detected in the FE mesh generated by the M-M algorithm.

The next step concerns the regularization of those elements. Many authors [11], [12] have studied this problem, providing smoothing solution or local optimization. The method presented here [13] considers the elements in their globality through an iterative process. Each iteration computes the jacobian determinant **detJ** to find irregular elements. Then irregular elements are corrected by slightly moving their nodes until every elements become regular. The direction of each node displacement is given by the gradient of **detJ**, computed for every nodes of the irregular elements.

For a distorted element $k$, the displacement $Disp_{k,j}$ for the node $j$ can be written as :

$$Disp_{k,j}=\sum_{i}(\nabla_j \mathbf{detJ}_i)=\begin{bmatrix} \sum_{i}\left(\frac{\partial \mathbf{detJ}_i}{\partial x_j}\right) \\ \sum_{i}\left(\frac{\partial \mathbf{detJ}_i}{\partial y_j}\right) \\ \sum_{i}\left(\frac{\partial \mathbf{detJ}_i}{\partial z_j}\right) \end{bmatrix} \quad (4)$$

where $i$ is the node index of element $k$ that has a negative or null determinant value.
The total displacement $Disp_j$ of the node $j$ is computed by adding each $Disp_{k,j}$ obtained for every irregular elements $k$ where the node $j$ appears. This displacement vector is normalized following :

$$Disp_j=\frac{\sum_{k} Disp_{k,j}}{\left|\sum_{k} Disp_{k,j}\right|} \quad (5)$$

Given this $Disp_j$ vector, nodes of irregular elements can be moved according to the following equations :

$$P_j' = P_j + Disp_j(y)*W_j \quad (6)$$

where $P_j$ and $P_j'$ are the previous and the new coordinates of the node $j$ and $W_j$ is a weight factor chosen to constrain the node displacement.
To get a resulting mesh geometry close to the patient morphology, an additionnal constraint maintains the surface nodes in the near neighborhood of their initial position (obtained by the M-M matching).

## 4. RESULTS

The application of the M-M method to femora [5] was a success since it generated automatically FE meshes fitting to the patient geometry and providing a mesh topology similar to the atlas FE mesh one (the atlas mesh is only composed of hexahedrons), despite significant geometry changes (Fig. 2). It takes 2 minutes to generate a femora FE mesh with the M-M. It is interesting to note that in the case of this bone structure; no irregularity was detected and consequently that the Mesh Matching algorithm is able to provide regular FE meshes for structures with quite simple shapes (Fig. 3).

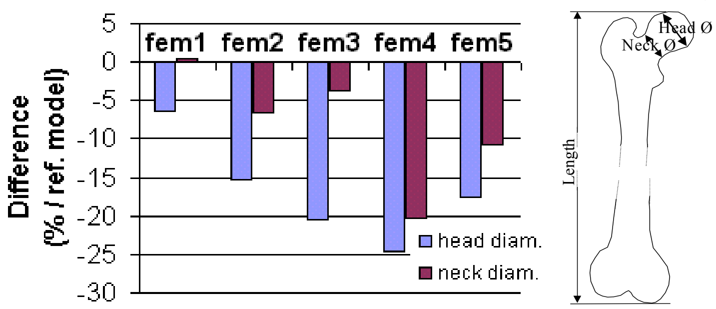

Fig. 2 : Difference of the head and neck diameter with respect to the reference model, on 5 FE meshes generated by the M-M algorithm

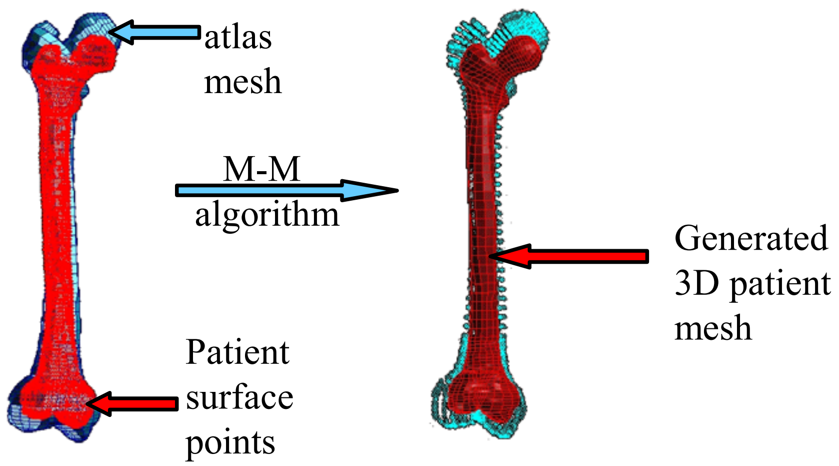


Fig. 3 : Application of the M-M algorithm to the femora FE mesh. The patient mesh is the smallest of the two meshes and is generated from the atlas mesh and the patient surface points.

For a more complex structure, namely a FE model of the human face, irregularities were detected after the M-M process. The regularization step has thus to be applied. Fig. 4 shows the atlas mesh, the patient mesh, an example of an irregular element and its correction. It took 2 minutes to generate the FE mesh of the patient face with the M-M algorithm and about 3 minutes to regularize it (on a DEC Alpha 500 MHz computer). The human face mesh is composed of 2884 elements and 4216 nodes. The M-M algorithm generates 149 irregular elements (about 5 % of the total number of elements) that were detected and corrected by the procedure.

As for the femora, the resulting patient mesh has been geometrically adapted to the patient morphology, with again the same topology that for the atlas. The preservation of this topology is important since the FE atlas mesh has been designed carefully to respect the biomechanical and morphological properties of the structure.

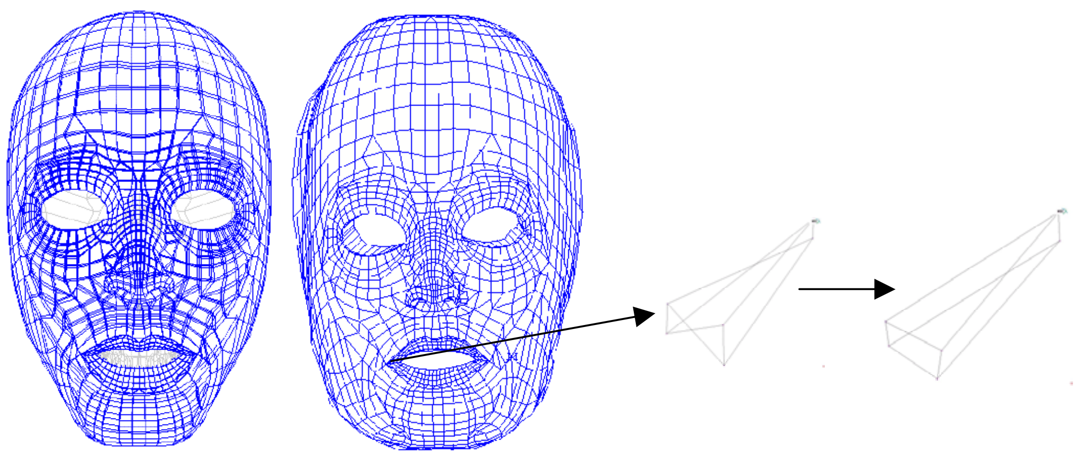

Fig. 4 : From left to right : the face FE atlas mesh; the patient FE mesh obtained with our method; an irregular element taken after the MM and regularized.

## 5. DISCUSSION

The combination of the M-M algorithm with the regularization technique for the automatic generation of various meshes has given encouraging results. Starting from an existing regular FE mesh, it has generated regular FE patient meshes that respect the patient morphology and the repartition of the elements providing a good finite element analysis. However, the efficiency of this method must be more quantitatively evaluated on more meshes [13].

The main drawback of our method is the requirement of the atlas, which is very long to build (as it is manually meshed with respect of the patient morphology and the structure) compared to the voxel-based automatic method [14]. But our method generates meshes with smooth surface and a good element refinement along the mesh, which is really interesting for precise simulations.

Future works focus on other structures such as the orbit or the shoulder and on the integration, in the regularization method, of element quality criterions such as the warping factor or the aspect ratio.

REFERENCES:

[1] Delaunay B., 1934. Sur la sphère. Izvestia Akademia Nauk SSSR, VII Seria, Otdelenie Matematicheskii i Estestvennyka Nauk. 7:793-800.
[2] Lo S. H., 1991. Volume Discretization into tetrahedra – II. 3D triangulation by advancing front approach. Computers and structures, 39(5):501-511.
[3] Craveur J.C., 1996. Modélisation des structures : calcul par éléments finis. Masson.
[4] Owen S., 1998. A survey of unstructured mesh generation technology. Proceedings of the 7th International Meshing Roundtable, Sandia National Lab :239-267.
[5] Couteau B., Payan Y. and Lavallée S. (2000). The Mesh-Matching algorithm : an automatic 3D mesh generator for finite element structures, *Journal of Biomechanics, vol. 33/8, 1005-1009.*
[6] Chabanas M. & Payan Y. (2000). A 3D Finite Element model of the face for simulation in plastic and maxillo-facial surgery. Proceedings of the Third International Conference on Medical Image Computing and Computer-Assisted Interventions - MICCAI'2000, October 2000, Pittsburgh, US.
[7] Joe B., 1995. Construction of three-dimensional improved quality triangulations using local transformations. SIAM Journal on Scientific Computing, 16:1292-1307.
[8] Lavallée S., Sautot P., Troccaz J., Cinquin P., Merloz P., 1995. Computer Assisted Spine Surgery: a technique for accurate transpedicular screw fixation using CT data and a 3D optical localizer. J. Image Guided Surg., 1(1), 65-73.
[9] Luboz V., Couteau B., Payan Y., 2001. 3D Finite element meshing of entire femora by using the mesh matching algorithm. In Proceedings of the Transactions of the 47th Annual Meeting of the Orthopaedic Research Society, p. 522. Transactions Editor, B. Stephen
[10] Zienkiewicz O.C., Taylor R.L., 1994. The Finite Element Method, fourth edition. McGraw-Hill Book Company.
[11] Cannan, S., Stephenson, M., Blacker, T., 1993. Optismoothing : an optimization-driven approach to mesh smoothing. Finite Elements in Analysis and Design, 13, pp. 185-190.
[12] Freitag, L.A., Plassmann, P., 1999. Local optimization-based simplicial mesh untangling and improvement. ANL/MCS, 39, pp. 749-756.
[13] Luboz V., Estivalèzes E., Payan Y., Pascal S.. Regularization Of A Mesh Generated With The Mesh Matching Algorithm. Submitted to the Journal of Biomechanics.
[14] Keyak et al., 1993. Validation of an Automated Method Of The Three-Dimensional Finite Element Modelling Of Bone, J. Biomed Eng, 15:505-509.