2024-1 | 49th congress of the Société de Biomécanique
29-31 octobre 2024, Compiègne, France

Multidisciplinary Biomechanics Journal

Automatic generation of patient-specific face finite element meshes

Marie-Charlotte Picard[a, b, c*], Pascal Perrier[c], Mohammad Ali Nazari[d], Michel Rochette[b], Rodolphe Lartizien[e], Georges Bettega[e], Yohan Payan[a]

a Univ. Grenoble Alpes, CNRS, Grenoble INP, TIMC, Grenoble, France
b ANSYS, Villeurbanne, France
c Univ. Grenoble Alpes, CNRS, Grenoble INP, GIPSA-lab, Grenoble, France
d School of Mechanical Engineering, University of Tehran, Tehran, Iran
e CH Annecy-Genevois, Epagny Metz-Tessy, France
* Corresponding author: marie-charlotte.picard@univ-grenoble-alpes.fr

Received date: 07/04/2024
Accepted date: 28/06/2024
Publication date: 31/01/2025

Keywords: human face, finite element modeling, image registration

1. Introduction

Our group is working on a project that studies the appearance and the mechanical behavior of a patient's face after orthognathic surgery. Osteotomy and bone realignment made during this surgery can indeed influence both the aesthetics of facial tissue and the motor functions upon them. Our aim is to predict these changes using a patient-specific nonlinear Finite Element (FE) model that represents main anatomical structures of the face, namely the bones and the soft tissues such as the hypodermis, fat, skin and muscles. The main challenge is the design of a nearly automatic method that requires no engineering expertise on the side of surgeons and being suitable with the time constraints of standard clinical procedures. Clinicians cannot spend weeks, days or even hours designing a specific face model for each patient and running simulations using FE approaches. This paper introduces an original method for the nearly automatic generation of patient-specific FE meshes. It starts from a reference model of the face and applies a geometric transformation to it, under some constraints, to generate a model tailored to the patient's face.

2. Methods

2.1 Reference Finite Element face model

The starting point of our methodology is a reference face model that has been developed in our group from 3D MR and CT scan images of a male adult subject (Nazari *et al.* 2011; Picard *et al.* 2023). This model is based on a FE mesh made of 131K quadratic tetrahedral elements and 31K nodes, which accounts for the geometry and the anatomy of the bones (mandible and maxilla) and soft tissue. The mesh includes two layers that represent the skin, which covers the external face surface, and the hypodermis. The soft tissue constitutive material is described by a second order Yeoh hyperelastic law. Muscular structures are included within the hypodermis layer. These muscles have an anisotropic mechanical behavior defined the law proposed by Nazari *et al.* (2022) and implemented as a Usermat in ANSYS APDL. Contacts are also modeled between bones and soft tissues, allowing sliding movements in certain areas, as well as sliding contacts between tissues at the lip junction.

2.2 Nearly automatic generation of patient-specific FE face meshes

To match the reference mesh to the patient's anatomy, the following nearly automatic process is defined:
1) Bone structures are geometrically deformed to match patient's anatomy using a non-rigid registration algorithm based on medical images (CT scan) using the Elastix library (Klein *et al.* 2009). Two types of transformations are applied. The first one is based on 3D BSpline non-rigid transformations, which only use the positions of anatomical landmarks manually selected by the surgeon on the patient CT images (these landmarks were predefined on the reference CT scan). The second transformation, also relying on BSpline functions,

To cite this article: Marie-Charlotte Picard, Pascal Perrier, Mohammad Ali Nazari, Michel Rochette, Rodolphe Lartizien, Georges Bettega, Yohan Payan, Automatic generation of patient-specific face finite element meshes, *Multidisciplinary Biomechanics Journal*, 2024, vol. 1, p. 339-341, DOI: 10.46298/mbj.14538.

utilizes the intensities of the voxels in each CT scan. The combination of both transformations generates a 3D displacement field, which is then applied to the FE meshes of the reference bone structures to obtain those of the patients. Each bony component, maxilla and mandible, is transformed separately, with two distinct registrations applied to the CT scan zones of interest delimited by binary masks.

2) Soft tissues are generated in 2 steps. First, an image-based non-rigid Elastix registration is applied between the reference CT scan and the patient CT scan, in order to transform the 3D soft tissue volume of the reference subject into the corresponding 3D volume in the patient. Then, a mechanical numerical simulation is run in Ansys APDL, based on the FE meshes (bones and soft tissues) obtained after this first registration, in order to keep the inner surface of the tissues in perfect contact with the bone surfaces. This simulation is essential since (1) applying the image-based registration directly to the mesh modeling the soft tissues in the reference face model could induce excessive distortions of the elements within the mesh, and (2) image-based registration could induce a loss of information on bone/tissue contacts, due to the lack of grey-scale accuracy in the CT scans as well as to artifacts in CT scans (lead teeth, braces, etc.) often observed in patients who undergo orthognathic surgery. In this simulation, the facial mesh obtained after the Elastix registration is deformed by applying a displacement to each node of the bone structures, thus repositioning them at their real location, known from the previous step. During this simulation, a sliding contact is imposed between bones and soft tissue to ensure proper connection. Finally, the nodes located on the external surface of the face are also moved to align them with their actual position, determined by a registration based on the region of interest between air and tissue (positioned on the CT images).

3. Results and discussion

Pre-operative CT data from 5 patients were utilized to create their respective FE meshes. Figure 1a illustrates the disparities observed between the surface of the FE meshes of these patients and their actual external face surfaces as reconstructed from CT scans (3DSlicer software). The color-map displayed represents the Euclidean distance between the nodes of the patient-specific FE mesh and the nodes of the reconstructed surface. When the reference FE mesh is deformed to match that of the patient, there is a risk of element quality degradation, which can compromise the accuracy of future numerical simulations and even lead to numerical errors. To quantify mesh quality, various geometric criteria are available in Ansys APDL and Workbench. In particular, we focus on a global metric proposed in Workbench, called *Element Quality*, which ranges from 0 (poor quality) to 1 (excellent quality). In fig. 1b, the quality of the elements in the patient FE meshes decreases in comparison with the quality of the reference mesh. This was expected because of the 3D geometric transformations. Nevertheless, it should be noted that mesh qualities remain very good (with only 1% of elements below a quality of 0.15), which will allow performing FE analyses based on these patient-specific meshes.

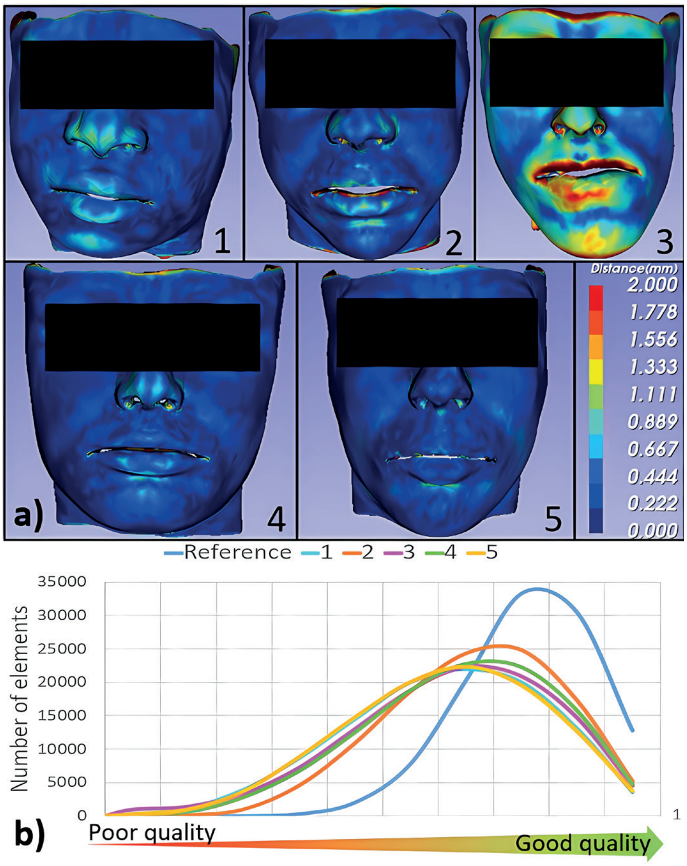

Figure 1. (a) Color map representing the discrepancies between the generated FE meshes of 5 patients and the actual external face geometry of these patients as reconstructed for their CT exams (mm); **(b)** Histograms of elements qualities in patient FE meshes, compared with the reference FE mesh.

4. Conclusion

This paper has introduced a fast process for the nearly automatic generation of patient-specific FE meshes. After manually positioning anatomical landmarks on each patient's CT scan, 5 meshes were created in just a few minutes. The results show that (1) the discrepancies between the generated meshes and the measured external face surface are smaller than one millimeter in the region of interest, and (2) the meshes are of sufficient quality to perform FE analyses. The next stage of this project will involve the simulation of bone cutting and repositioning, followed by a quantitative comparison between the predictions provided by the FE models and the surgical outcomes.

Conflict of Interest

None

References

Nazari M., Perrier P. & Payan Y. (2022). Interwoven muscle fibers: a 3D two-fiber muscle active model. *CMBBE*, 25:sup1, S226-S228.

Klein, S., Staring, M., Murphy, K., Viergever, M. A., & Pluim, J. P. (2009). *Elastix: a toolbox for intensity-based medical image registration. IEEE transactions on medical imaging*, 29(1), 196-205.

Nazari M., Perrier P., Chabanas M. & Payan Y. (2011). *Shaping by stiffening: a modeling study for lips. Motor Control*, Vol. 15(1), pp. 141-168.

Picard M.C., Cotton M., Briot N., Nazari M., Poisbleau D., Perrier P., Chagnon G., Rochette M. & Payan Y. (2023). *Biomechanical model of the human face with a perspective of surgical assistance. CMBBE*, 26:sup1, S202-S204.